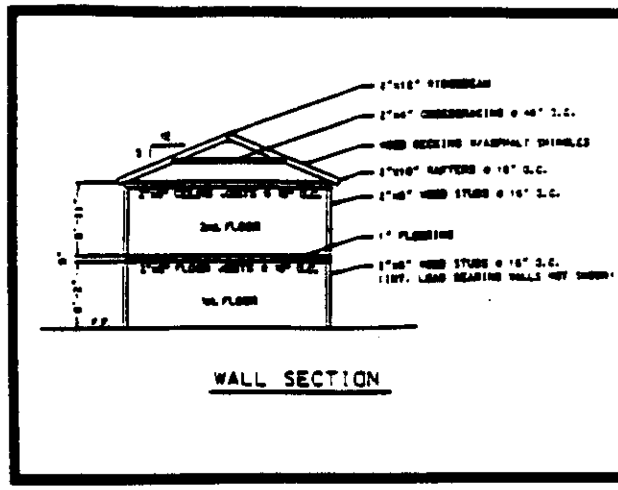
Protective Design - Mandatory Center of Expertise
Technical Report 92-3

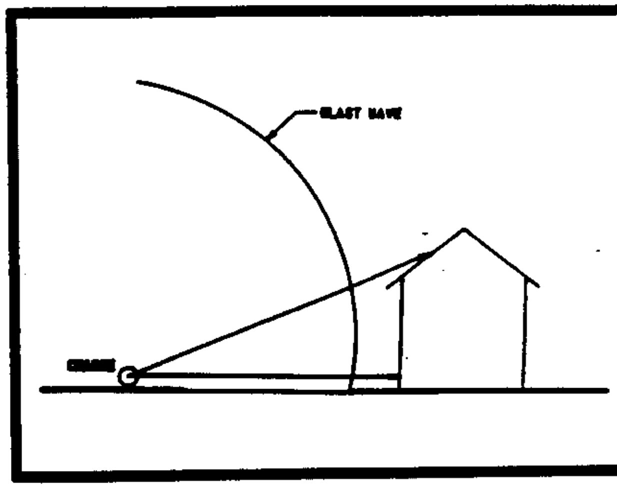


Facility And Component Explosive Damage Assessment Program

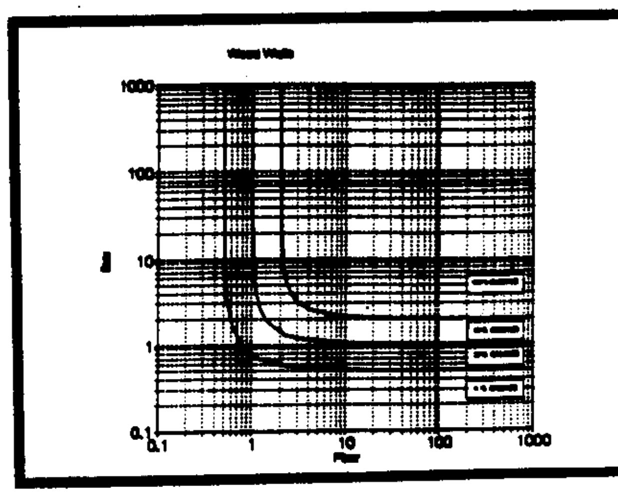
(FACEDAP)

User's Manual

Version 1.2



SwRI Project No. 06-5145-001
Contract No. DACA 45-91-D-0019



Prepared for:

Department of the Army
Corps of Engineers, Omaha District
CEMRO-ED-ST
215 N. 17th Street
Omaha, Nebraska 68102-4978

Modified May 1994

Table of Contents

List of Figures

List of Tables

Table of Contents (Continued)

Table of Contents (Continued)

Table of Contents (Continued)

Table of Contents (Continued)

Table of Contents (Continued)

List of Figures

List of Tables

THIS PAGE INTENTIONALLY LEFT BLANK

1.0 INTRODUCTION

This manual discusses and illustrates usage of the FACEDAP (Facility And Component Explosive Damage Assessment Program) computer code, which calculates blast damage to a building from an external explosive threat. This manual is intended to be read in conjunction with the FACEDAP Programmer's Manual[1], the FACEDAP Theory Manual[2], the IOSUB User's Manual[3], and the ESHELL User's Manual[4]. The FACEDAP Programmer's Manual discusses the basic architecture of the FACEDAP program and general programming information. The FACEDAP Theory Manual discusses and assesses the theoretical approach used by the FACEDAP program to determine building damage from blast loading. The IOSUB User's Manual and ESHELL User's Manual describes subroutines in these software packages which are used in the FACEDAP program to control screen display and to run the FACEDAP driver using a minimum amount of computer memory.

The FACEDAP program is intended to be a tool for quickly determining the approximate structural damage to conventional buildings and building components caused by a given external explosive threat (e.g., no gas pressure loads are considered). Non-structural components, such as windows and doors, are not currently considered by the program. This procedure, which has been developed based on available test data and on basic dynamic structural response theory, is intended to predict damage without the built-in conservatism that is usually present in design procedures. Structural component damage is calculated in terms of four damage levels (0%, 30%, 60%, and 100% damage) that have been correlated with the levels of protection used by the U.S. Army Corps of Engineers as follows[4]: 1) the 0% damage level is similar to the High Level of Protection; 2) the 30% damage level is similar to the Medium Level of Protection; 3) the 60% damage level is similar to the Low Level of Protection; and 4) the 100% damage level is similar to Collapse.

The program can be used to determine blast damage to an individual component or to an entire building. Building damage is based on the calculated damage level of each component in the building. The percentage of building damage is calculated by "weighting" each calculated component damage level with a weighting factor, summing the weighted damage of all building components, and then dividing this sum by the value corresponding to total failure of all building components and converting this ratio to a percentage. Cascading failure, where failure of a supporting component causes failure of all supported components, is considered in the summation algorithm. Building repairability and reusability are also considered in similar summation processes. A building level of protection is calculated by the program equal to the lowest level of protection calculated for any of the building components. The component damage and building vulnerability calculation procedures are discussed in more detail in the Theory Manual[2].

Because of assumptions used in the methodology and because of limited validation, there are several recommended limitations on the usage of the FACEDAP program. **First, the FACEDAP program should not be used for final design or for any case where high accuracy is required.** This caution is necessary because the pressure-impulse diagrams, which predict building component blast damage for the twenty-four different structural components considered in the methodology, are based on limited data in some cases, and on simple dynamic structural response theory and a number of assumptions in other cases. Neither the pressure-impulse diagrams, nor the summation procedure used to get percentage of building damage, have been fully validated yet to determine the bounds on their accuracy. Secondly, the explosive charge should be at a scaled standoff of at least 3.0 ft/lb$^{1/3}$ away from the building and located near the ground surface. This will help ensure that assumptions used to calculate the blast load on each building component are applicable.

Thirdly, the pressure-impulse diagrams only predict damage to components during flexural or buckling mode response and do not consider any damage due to shear failure. Although the available test data has not shown that conventionally designed components tend to fail in shear under blast loads that are applied at scaled standoffs greater than 3.0 ft/lb$^{1/3}$, no thorough study has been made of how this limitation affects the FACEDAP program. Some conservatism can be built into the calculated building damage if conservative strength properties are input for building components.

1.1 Overview of FACEDAP Program

The FACEDAP program consists of a driver, FACEDAP.EXE, three primary processors; 1) the Preprocessor, 2) the building damage Analysis program, and 3) the Postprocessor, as well as minor executables and subroutines which retrieve existing input files and set default configuration values. These are illustrated in Figure 1. The executables (*.EXE files) in Figure 1 are "child programs" which are run, one at a time, as designated by input from the user into the driver program. The child programs pass information to each other by writing and reading files from the hard disk. In general, each child program reads a file containing the required input information in the default data directory on the hard disk, reads additional information input by the user as required, performs various calculations using the input information, displays some calculated results to the screen, and writes an output file that is used as input by one of the other child programs.

The primary program modules in Figure 1 are the Preprocessor, the Analysis module, and the Postprocessor. The FACEDAP Preprocessor reads the input charge weight and location and detailed information defining the individual structural component or the building which is exposed to blast loading. Since the FACEDAP program calculates blast damage using logic which considers damage to each building component, the building analysis option requires all structural components of the building exposed directly or indirectly to significant blast load to be input into the Preprocessor. In order to reduce the amount of work involved in this input, the Preprocessor generates building components within building wall/roof areas based on an input "master" component, generates dependencies between building components which are required in the building damage calculation scheme, and reads component endpoint locations in terms of user-defined local coordinate systems in each in wall and roof area. These features require that the building is divided into large, planar wall/roof areas which are defined by four input corner points. All subsequent building input and output is defined within one of the user-defined wall/roof areas. This allows the user to more easily "manage" the relatively large amounts of detailed building information which must be input without the aid of a graphical display. The cross sectional properties and material properties of each building component must also be defined in detail by the user. The Preprocessor reduces the effort involved in this task by only requiring input of material and cross sectional properties for unique components rather than for each building component. An input set of component properties can be assigned to components throughout the building with the given set of properties. In general, the building analysis part of the Preprocessor has been designed to take advantage of the fact that most buildings are comprised of a relatively small number of "unique" building components which are used repetitively throughout the building construction.

The Preprocessor is also user-friendly in a number of other respects. There are many checks built into the Preprocessor which issue warnings to the user during input if there appears to be an error in the input. The Preprocessor makes extensive use of the pre-coded subroutines in the IOSUB software which display user friendly form-type and spreadsheet-type screens to accept input. The Preprocessor also includes a validation program which checks the user input and provides a specific explanation of each error that is found. A separate section of this manual provides additional explanation of the warning and error messages generated by the program.

If no errors are found in the input by the validation program, the user can run the BDAMA.EXE program in the Analysis module, which calculates individual component or building damage. After the analysis has been completed, the Postprocessor program displays the final calculated blast damage and the results from intermediate calculations used to determine the structural damage. The Postprocessor also makes extensive use of the IOSUB software to display large amounts of component building damage and blast load information on spreadsheet-type screens within each user-defined wall/roof area of the building. The FACEDAP driver also allows the user to retrieve previous input files; establish default directories, printers, and print communications ports; set default screen colors; and shell out to the DOS operating system. These capabilities are included in the boxes marked "Utilities" and "Files" in Figure 1.

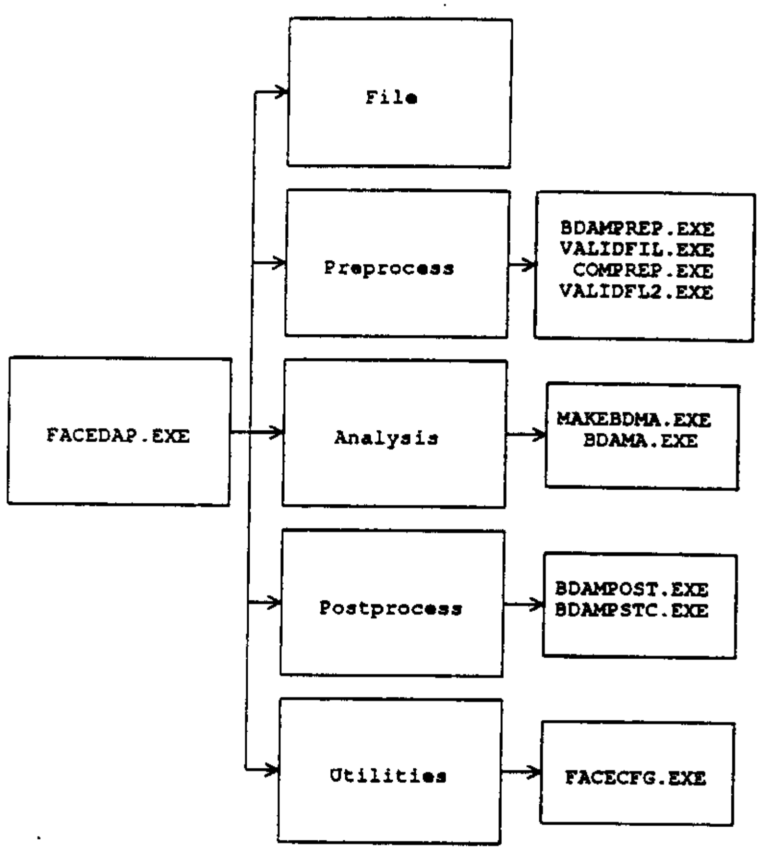


Figure 1. Flow Diagram for FACEDAP Program Driver

Most of the FACEDAP program was written during the current project. However, the BDAMA.EXE program, which is used in the Analysis module of the FACEDAP program was written previously by Southwest Research Institute (SwRI) for the Naval Civil Engineering Laboratory (NCEL) in Port Hueneme, California. The blast damage prediction procedure used by the BDAMA.EXE program to calculate component blast damage has been incrementally developed by SwRI during a number of previous projects. These projects, which are explained in Reference 2, have been funded by NCEL and the U.S. Army Corp of Engineers, Omaha District beginning in 1987 and continuing to this project, which included significant modifications to the damage prediction procedure and to the BDAMA.EXE program. More development and validation of this procedure is envisioned in the future.

The BDAMA.EXE program has also been incorporated into the BDAM program written by NCEL. The current version of BDAM reads all required building component information on a component-by-component basis through interactive input screens and passes this information to the BDAMA.EXE program. The FACEDAP program is similar to BDAM except that the FACEDAP Preprocessor significantly reduces the volume of building component information the user must enter and the Postprocessor displays significantly more information related to calculated blast loads and blast damage for each building component.

2.0 HARDWARE REQUIREMENTS

The hardware requirements for the FACEDAP program are:

1) At least 485 kilobytes of available conventional memory.

2) At least 6 megabytes of hard disk space available for the program executables and the input and output files written to disk during program execution.

3) A math coprocessor is highly recommended because it reduces the program run time by a very significant amount. The program is compiled so that it will search for a math coprocessor and use the coprocessor if it is available but it will also run if there is no coprocessor.

4) A minimum of an 80286 CPU.

5) DOS 3.1 or later

The user can check the size of the largest executable which can be run while any given set of background programs (such as WINDOWS, etc.) are running by entering the DOS command "mem" while these programs are in memory. If the largest executable size is less than 485 kilobytes, the user cannot run all the background programs at the same time the FACEDAP code is executed. The user may not be warned if he/she chooses a module, such as the Preprocessor, which requires more than the available memory. When the Preprocessor is run with insufficient memory, the program may issue an incorrect error message stating that the input file has not been validated. When this happens, or when any other odd or incorrect error messages are displayed, the user should exit the FACEDAP program and use the DOS "mem" command to check the available memory, since insufficient memory can cause some incorrect error messages to be displayed.

3.0 INSTALLATION INSTRUCTIONS

The FACEDAP program is installed from the program floppy disk onto the hard drive of a personal computer (PC) as shown below. For illustration purposes, it is assumed the user is installing to the C hard drive from the B floppy drive. The floppy disk contains a README.DOC file, an archived FACEDAP.ZIP file that contains all the files required to run the FACEDAP program, an archived BUILDING.ZIP file that contains input files for each of the example buildings shown in Section 14.0 of this manual, and the PKUNZIP.EXE file which will unarchive the two archived files. The installation instructions shown below are included in the README.DOC file on the program disk.

1) Create a directory for the FACEDAP program on the computer hard disk. If a directory already exists with a previous version of this program, delete all files.

   >md c:\facedap or >del *.*

2) Copy all the files from the program floppy disk into the program directory (assumed here to be the directory "facedap"). The FACEDAP.ZIP, BUILDING.ZIP, README.DOC, and PKUNZIP.EXE files are contained on the program executable disk. The README.DOC file contains the same instructions printed here.

   >copy b:*.* c:\facedap

3) Unarchive the FACEDAP.ZIP and BUILDING.ZIP files using the PKUNZIP command file.

   >pkunzip facedap *.*
   >pkunzip building *.*

4) Create a directory for the FACEDAP program data files on the computer hard disk.

   >md c:\facedap\data

5) Go to the program data directory (assumed here to be the directory "facedap\data").

   >cd facedap\data

6) Copy the sixteen files unarchived from the BUILDING.ZIP file (which are named EXAMPL1.BLG through EXAMPL13.BLG, EXAMPL2A.BLG, and EXAMPL7A.BLG) from the program directory into the program data directory.

   >copy c:\facedap*.blg *.*

3.0 INSTALLATION INSTRUCTIONS

The FACEDAP program is installed from the program floppy disk onto the hard drive of a personal computer (PC) as shown below. For illustration purposes, it is assumed the user is installing to the C hard drive from the B floppy drive. The floppy disk contains a README.DOC file, an archived FACEDAP.ARC file that contains all the files required to run the FACEDAP program, an archived BUILDING.ARC file that contains input files for each of the example building shown in Section 14.0 of this manual, and the PKXARC.COM file which will unarchive the two archived files. The installation instructions shown below are included in the README.DOC file on the program disk.

1) Create a directory for the FACEDAP program on the computer hard disk. If a directory already exists with a previous version of this program, delete all files.

   >md c:\facedap or >del *.*

2) Copy all the files from the program floppy disk into the program directory (assumed here to be the directory "facedap"). The FACEDAP.ARC, BUILDING.ARC, README.DOC, and PKXARC.COM files are contained on the program executable disk. The README.DOC file contains the same instructions printed here.

   >copy b:*.* c:\facedap

3) Unarchive the FACEDAP.ARC and BUILDING.ARC files using the PKXARC command file.

   >pkxarc facedap *.*
   >pkxarc building *.*

4) Create a directory for the FACEDAP program data files on the computer hard disk.

   >md c:\facedap\data

5) Go to the program data directory (assumed here to be the directory "facedap\data").

   >cd facedap\data

6) Copy the fifteen files unarchived from the BUILDING.ARC file (which are named EXAMPL1.BLG through EXAMPL13.BLG, EXAMPL2A.BLG, and EXAMPL7A.BLG) from the program directory into the program data directory.

   >copy c:\facedap\exampl*.blg *.*

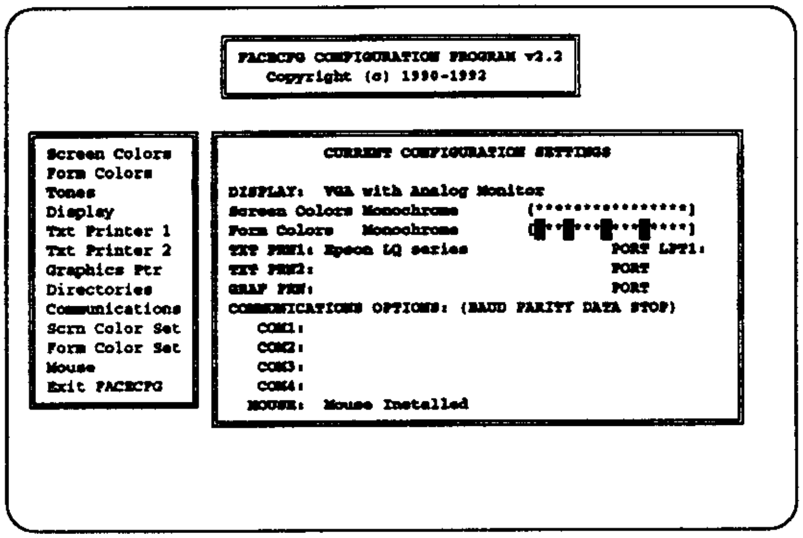
7) Return to the program directory and delete the files copied into the program data directory.

>cd facedap
>del *.blg

8) Run the interactive configuration program named FACECFG.EXE.

>facecfg

This causes the following screen to appear:



9) Use the arrow keys and enter key to select the options shown below on the configuration program screen.

a. Set mouse, if you have a mouse.

b. Select printer type and port.

c. Check directories - all directories except the data directory should be set to the program directory name (c:\facedap in this example). The data directory should be set to the default program data directory name (c:\facedap\data in this example).

d. Select screen colors if default colors are not acceptable.

10) Run the FACEDAP program

>facedap

4.0 REQUIRED FILES

Table 1 shows a list of the files in the FACEDAP.ARC file which are required to run the FACEDAP code.

Five types of files are required: 1) the configuration file (FACE.BSI); 2) program data files (*.DAT); 3) executable files (*.EXE); 4) form files (*.FRM); and 5) printer files (*.PRN). The configuration file is used by FACECFG to create the FACE.CFG file. The data files contain information required by the various executable files. The form files define the structure of various input and output screens displayed by the program. The printer files contain the printer codes of various printers which may be used by the print program, PR2.EXE.

Thirteen example input files (*.BLG files) are also included in the BUILDING.ARC file on the floppy disk with the FACEDAP program. These files have input for thirteen conventionally constructed example buildings which are shown in Section 14.0 of this manual. Each example building is defined by a number (e.g. Example Building No. 1). The files are entitled EXAMPL1.BLG through EXAMPL13.BLG, where the number near the end of the filename refers to the number of the example building defined by the file input.

5.0 INPUT AND OUTPUT FILES

Table 2 shows a list of the files created by the executables in the FACEDAP program during execution and read later by other executables. As mentioned previously, the executables shown in Figure 1 run one at a time and pass information to each other through files written to the hard disk in the default data directory. This directory is set by running the FACECFG executable and selecting the *Directories* option or by invoking this executable with the *Utilities* option in the main menu of the FACEDAP program. The files marked *.BLG, *.ERR, *.PST, and *.REP are stored with a problem specific root name designated by the user and can be retrieved at a later time with the FACEDAP program. The *.BLG file is the primary file where user input is stored. The *.ERR file stores error messages written by the validation program (VALIDFIL.EXE in Figure 1) and the *.PST file stores output building damage, component damage, and blast load information written by the BDAMA.EXE program. The BDAMA.IN file in Table 2 stores user input in the format required by the BDAMA.EXE program. Previous input is retrieved in the FACEDAP program as explained in Section 7.2 by selecting the *.BLG file with the root name of interest.

Table 1. Files Required to Run the FACEDAP Program

FILENAME	FILE TYPE	REQUIRED BY
FACE.BSI	CONFIGURATION FILE	FACECFG
MAINTITL.DAT	DATA FILE	FACEDAP
OPTIONTB.DAT	DATA FILE	BDAMPREP BDAMPOST
SPRDHEAD.DAT	DATA FILE	BDAMPREP BDAMPOST
BDAMA.EXE	EXECUTABLE	N/A
BDAMPOST.EXE	EXECUTABLE	N/A
BDAMPREP.EXE	EXECUTABLE	N/A
FACECFG.EXE	EXECUTABLE	N/A
FACEDAP.EXE	EXECUTABLE	N/A
MAKEBDMA.EXE	EXECUTABLE	N/A
PR2.EXE	EXECUTABLE	N/A
VALIDFIL.EXE	EXECUTABLE	N/A
BUILDDAM.FRM	FORM FILE	BDAMPOST
LOADDEF.FRM	FORM FILE	BDAMPREP
LOADPOST.FRM	FORM FILE	BDAMPOST
PRBTITLE.FRM	FORM FILE	BDAMPREP
EPSFX.PRN	PRINTER DATA FILE	PR2
EPSLQ.PRN	PRINTER DATA FILE	PR2
EPSRX.PRN	PRINTER DATA FILE	PR2
IBM.PRN	PRINTER DATA FILE	PR2
OKIDATA.PRN	PRINTER DATA FILE	PR2
HPLJ.PRN	PRINTER DATA FILE	PR2
HPPAINT.PRN	PRINTER DATA FILE	PR2
PS.PRN	PRINTER DATA FILE	PR2
QMS410.PRN	PRINTER DATA FILE	PR2

Table 1. Files Required to Run the FACEDAP Program

FILENAME	FILE TYPE	REQUIRED BY
FACE.BSI	CONFIGURATION FILE	FACECFG
DISCLAIM.DAT	DATA FILE	FACEDAP
MAINTITL.DAT	DATA FILE	FACEDAP
OPTIONTB.DAT	DATA FILE	BDAMPREP BDAMPOST
OPTION2.DAT	DATA FILE	COMPPREP
SPRDHEAD.DAT	DATA FILE	BDAMPREP BDAMPOST
SPRDHED2.DAT	DATA FILE	COMPPREP
BDAMA.EXE	EXECUTABLE	N/A
BDAMPOST.EXE	EXECUTABLE	N/A
BDAMPREP.EXE	EXECUTABLE	N/A
BDAMPSTC.EXE	EXECUTABLE	N/A
COMPPREP.EXE	EXECUTABLE	N/A
FACECFG.EXE	EXECUTABLE	N/A
FACEDAP.EXE	EXECUTABLE	N/A
MAKEBDMA.EXE	EXECUTABLE	N/A
PR2.EXE	EXECUTABLE	N/A
VALIDFIL.EXE	EXECUTABLE	N/A
VALIDFL2.EXE	EXECUTABLE	N/A
BUILDAM2.FRM	FORM FILE	BDAMPOST
BUILDAMC.FRM	FORM FILE	BDAMPSTC
COMPDEF.FRM	FORM FILE	COMPPREP
LOADCMPP.FRM	FORM FILE	BDAMPSTC
LOADCOMP.FRM	FORM FILE	COMPPREP
LOADDEF.FRM	FORM FILE	BDAMPREP
LOADPOST.FRM	FORM FILE	BDAMPOST
PRBTITLE.FRM	FORM FILE	BDAMPREP
EPSFX.PRN	PRINTER DATA FILE	PR2
EPSLQ.PRN	PRINTER DATA FILE	PR2
EPSRX.PRN	PRINTER DATA FILE	PR2
IBM.PRN	PRINTER DATA FILE	PR2
OKIDATA.PRN	PRINTER DATA FILE	PR2
HPLJ.PRN	PRINTER DATA FILE	PR2
HPPAINT.PRN	PRINTER DATA FILE	PR2
PS.PRN	PRINTER DATA FILE	PR2
QMS410.PRN	PRINTER DATA FILE	PR2

Table 2. FACEDAP Program Input and Output File Names

EXECUTABLE FILE NAME	INPUT FILES	OUTPUT FILES
FACECFG.EXE	FACE.BSI	FACE.CFG
FACEDAP.EXE	FACE.CFG, *.BLG DISCLAIM.DAT MAINTITL.DAT	N/A
BDAMPREP.EXE	FACE.CFG, *.BLG OPTIONTB.DAT SPRDHEAD.DAT PRBTITLE.FRM LOADDEF.FRM	*.BLG *.REP
COMPPREP.EXE	FACE.CFG OPTION2.DAT SPRDHED2.DAT PRBTITLE.FRM LOADCOMP.FRM COMPDEF.FRM	COMP*.BLG *.REP
VALIDFIL.EXE	FACE.CFG, *.BLG	*.BLG *.ERR
VALIDFL2.EXE	FACE.CFG COMP*.BLG	COMP*.BLG *.ERR
MAKEBDAM.EXE	FACE.CFG, *.BLG	BDAMA.IN
BDAMA.EXE	BDAMA.IN, *.BLG	*.PST
BDAMPOST.EXE	FACE.CFG OPTIONTB.DAT SPRDHEAD.DAT BUILDAM2.FRM LOADPOST.FRM *.BLG, *.PST	*.REP
BDAMPSTC	FACE.CFG BUILDAMC.FRM LOADCMPP.FRM COMP*.BLG COMP*.PST	COMP*.REP
PR2	FACE.CFG *.PRN, *.*	N/A

6.0 INPUT/OUTPUT SCREENS IN FACEDAP

Five basic types of input/output screens are utilized in the FACEDAP program. These five types are pull down menus, vertical menus, multi row and column menus, forms and spreadsheets. All input/output to the screen is controlled by pre-coded subroutines in the IOSUB software. Some of these subroutines have been modified in the FACEDAP code.

Table 2. FACEDAP Program Input and Output File Names

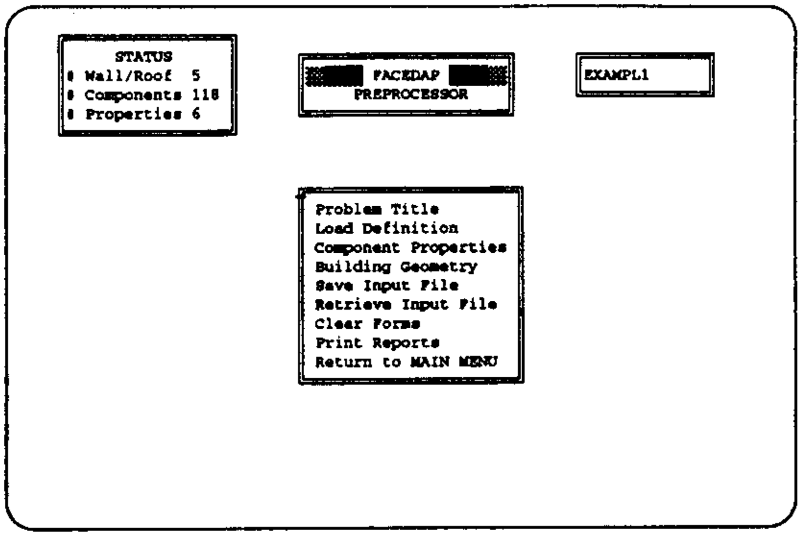
EXECUTABLE FILE NAME	INPUT FILES	OUTPUT FILES
FACECFG.EXE	FACE.BSI	FACE.CFG
FACEDAP.EXE	FACE.CFG MAINTITL.DAT *.BLG	N/A
BDAMPREP.EXE	FACE.CFG OPTIONTB.DAT SPRDHEAD.DAT PRBTITLE.FRM LOADDEF.FRM *.BLG	*.BLG *.REP
VALIDFIL.EXE	FACE.CFG *.BLG	*.BLG *.ERR
MAKEBDAM.EXE	FACE.CFG *.BLG	BDAMA.IN
BDAMA.EXE	BDAMA.IN *.BLG	*.PST
BDAMPOST.EXE	FACE.CFG OPTIONTB.DAT SPRDHEAD.DAT BUILDDAM.FRM LOADPOST.FRM *.BLG *.PST	*.REP
PR2	FACE.CFG *.PRN *.*	N/A

6.0 INPUT/OUTPUT SCREENS IN FACEDAP

Five basic types of input/output screens are utilized in the FACEDAP program. These five types are pull down menus, vertical menus, multi row and column menus, forms and spreadsheets. All input/output to the screen is controlled by pre-coded subroutines in the IOSUB software. Some of these subroutines have been modified in the FACEDAP code.

6.1 Vertical Menu

The vertical menus are utilized throughout the Preprocessor and Postprocessor. The main menu of the Preprocessor, which follows, illustrates this type of menu.



An option can be selected from a vertical menu using one of several methods. The following table summarizes the keystrokes with can be used to highlight and select an option.

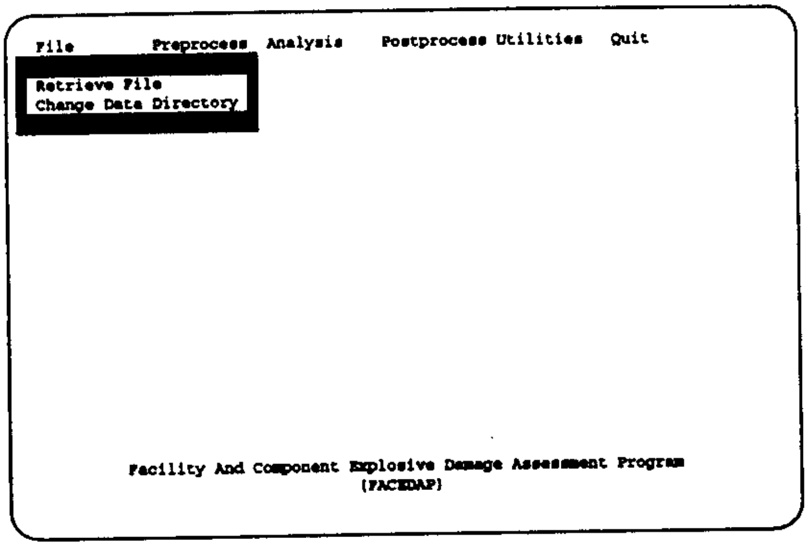
Table 3. Keystrokes Used in Vertical Menus

KEY	FUNCTION
↑	Move to previous row of menu and highlight (circular wrap)
↓	Move to next row of menu and highlight (circular wrap)
First letter of menu item	Highlights the line containing the pressed. If pressed again finds next occurrence. (circular wrap)
←	Select current highlighted row

If more than one option begins with the same letter, the first input of the letter highlights the first occurrence of an option beginning with the input letter, the second input of the letter finds the second occurrence, etc. An end-around wrap occurs after the last occurrence has been reached. An option in the menu can also be both highlighted and selected be clicking on the option with the left mouse button. The return option at the end of each vertical menu in the program is the primary means of exiting to the previous menu. In some cases, the Esc key can be used to escape from a vertical menu to the previous menu.

6.2 Pull Down Menu

The FACEDAP Main Menu is a Pull Down menu. This menu initially displays a horizontal list of primary menu options on the top line of the screen. A vertical submenu of options is associated with most of the primary options. The screen with the Main Menu from the FACEDAP program follows. The vertical submenu for the first option of the Main Menu is also shown.



The primary options along the top line are accessed by movement of the ← and → cursor keys or by clicking on the option with a mouse. These actions automatically cause a submenu of options associated with the selected primary option to appear, except for cases where no further user input is required. The vertical submenus function in the same manner as the vertical menu just described in Section 6.1. The ↵ key or the mouse is used to select an option off the vertical submenu. The ↵ key is also used to select a highlighted option on the primary horizontal menu if the option does not have a submenu. The user can move out of a vertical submenu by using the ← and → cursor keys or the mouse and select another primary option along the top line of the screen.

6.3 Multi Row and Column Menu

The Multi Row and Column Menu provides a list of options in tabular form, i.e., several columns of data and one or more rows. This type of menu is typically used to display options with user-defined names, such as the wall/roof areas which is in the following screen.

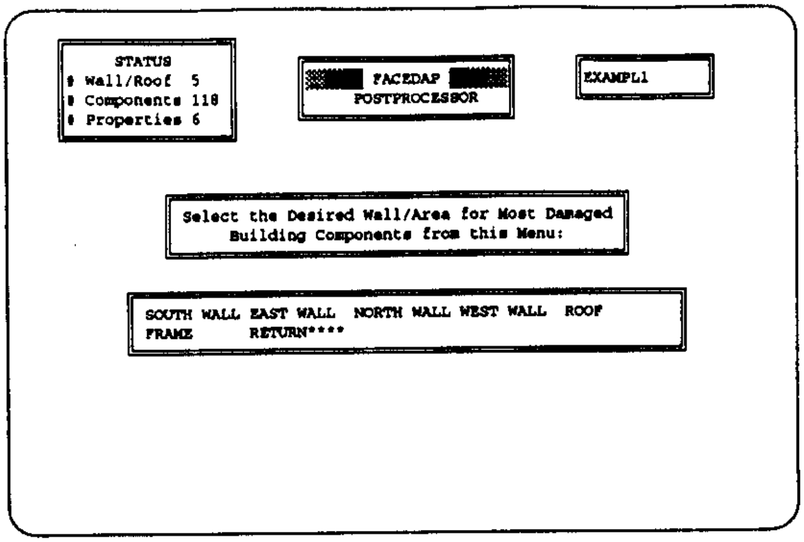


Table 4 summarizes the keystrokes that can be used to select an option in a Multi Row and Column Menu.

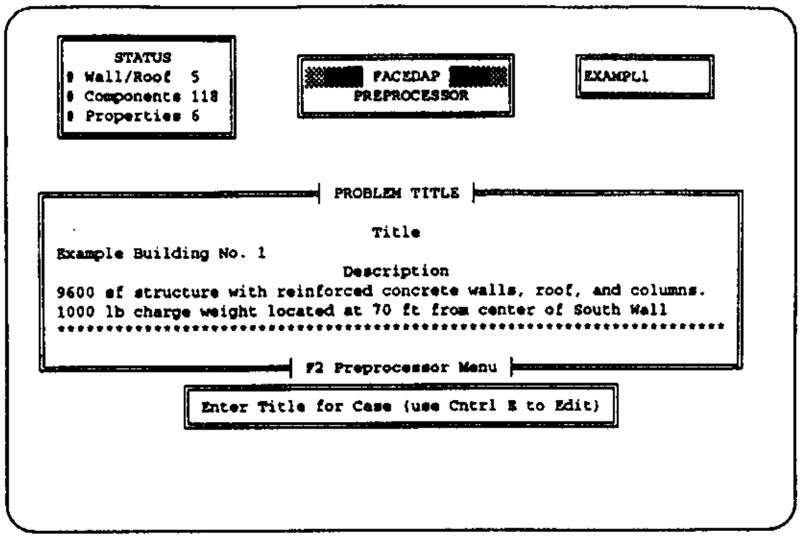
The ← and → cursor keys provide circular movement on the current row of the table. These keys are used to highlight an option. The first letter of an option can be used to highlight the option as explained in Section 6.1. The left mouse button may also be used to highlight and select and option. The ↑ and ↓ keys cycle within a column. If the total number of items for display exceeds the reserved menu space, "PgUp" and "PgDn" appear at the top and bottom of the menu. The PgUp, PgDn, Home, and End keys are used to move between screens of large menus. The Esc allows escape from the current menu to the previous menu. This type of menu also includes a return option, as shown in the preceding example screen, which is the primary means of returning to the previous menu.

6.4 Form-Style Screens

Form-style screens are used where a small amount of data is required for either input or output. An example of an input form from the Preprocessor follows after Table 4.

Table 4. Keystrokes Used in Multi Row and Column Menus

KEY	FUNCTION
←	Move left to previous column of current row in current window (circular wrap)
→	Move right to next column of current row in current window (circular wrap)
↑	Move up current column to previous row in current window (circular wrap)
↓	Move down current column to next row in current window (circular wrap)
PgUp	Display previous menu items in window. Only functional when more rows are in menu than can fit in window.
PgDn	Display next menu items in window. Only functional when more rows are in menu than can fit in window.
Home	Display first rows of menu in window. Only functional when more rows are in menu than can fit in window.
End	Display last rows of menu in window. Only functional when more rows are in menu than can fit in window.
↵	Select current highlighted option
Esc	Abort current menu selection and return to previous menu



Movement through the form is accomplished by the cursor keys or the [↵] key. The following table summarizes the keystrokes.

Table 5. Keystrokes Used in Form-Style Input Screens

KEY	FUNCTION
[↑]	Move to previous field on form
[↓]	Move to next field on form
[↵]	Move to next field on form
[Ctrl-E]	Invokes the line editor (see table which follows for more information)
[Space Bar] followed by [↵]	Blank current field. To abort the field deletion, press [Esc] after the [Space Bar] instead of [↵].
[F2]	Exit form

A help message is displayed at the bottom of the form for the current field. Editing of a given field is achieved by re-typing the line or using the edit capability of the field. [Ctrl-E] invokes the line editor. The table which follows indicates the available keys in the line editor and their function.

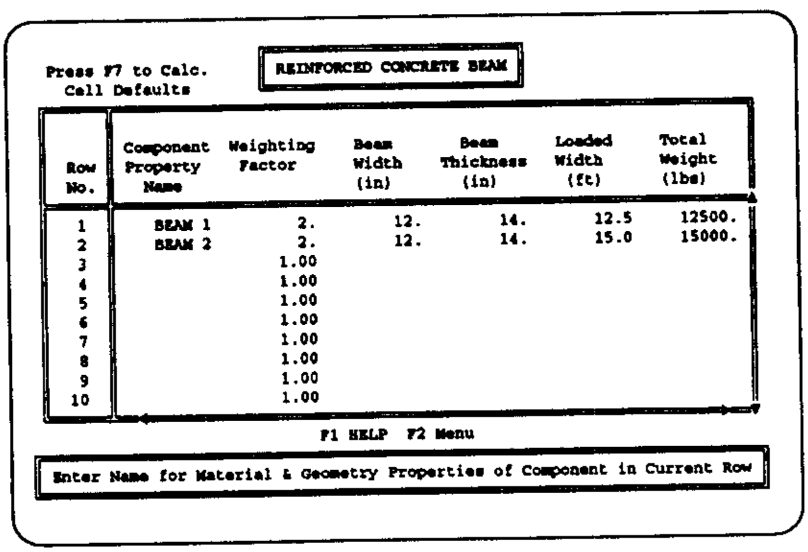
Table 6. Keystrokes Used While in Edit Mode

KEY	FUNCTION
[←]	Moves one character to the left
[→]	Moves one character to the right
[Home]	Moves to beginning of line
[End]	Moves to end of line
[Ins]	Toggle for insert/overtype
[Del]	Delete current character
[←]	Destructive backspace

In the Postprocessor, the forms are used for display purposes only and therefore the fields are read-only. **The user exists from form screens by pressing [F2] or by clicking the left mouse button on the F2 area of the form.**

6.5 Spreadsheet-Style Screens

Spreadsheet-style input screens are used where a large amount of data is required for either input or output. Data is organized into rows and columns. Each column has a header and a column-specific help message, which appears at the bottom of the spreadsheet as the user moves across columns. An example of this type of input/output screen follows. The help message for the first column in the spreadsheet is at the bottom of the screen.


Press F7 to Calc.
Cell Defaults

REINFORCED CONCRETE BEAM

Row No.	Component Property Name	Weighting Factor	Beam Width (in)	Beam Thickness (in)	Loaded Width (ft)	Total Weight (lbs)
1	BEAM 1	2.	12.	14.	12.5	12500.
2	BEAM 2	2.	12.	14.	15.0	15000.
3		1.00				
4		1.00				
5		1.00				
6		1.00				
7		1.00				
8		1.00				
9		1.00				
10		1.00				

F1 HELP F2 Menu

Enter Name for Material & Geometry Properties of Component in Current Row

A summary of the keystrokes used to move in the spreadsheet screens and input data follow. These may be viewed anytime on the spreadsheet by pressing [F1].

Table 7. Keystrokes Used in Spreadsheet-Style Input/Output Screens

KEY	FUNCTION
↵	Move right to next field; Move to 1st field next row when on last column
↑	Move up one row
↓	Move down on row
→	Move right one field
←	Move left one field
PgUp	Page up
PgDn	Page down
Ctrl-Right	Page right
Ctrl-Left	Page left
Ctrl-D	Delete current row and move remaining rows up a row
Ctrl-I	Insert a row after the current row
Home	Move to 1st column of current row
End	Move to last column of current row
F2	Return to previous menu
F8	Duplicate cell from previous row. Does not duplicate cells displaying parameters defined by the program.
Alt-F8	Duplicate previous row. Does not duplicate cells displaying parameters defined by the program.
Space Bar followed by ↵	Blank current cell. If the current cell is a character type the contents will be deleted when the Space Bar is pressed followed by ↵. To abort the cell deletion, press Esc after the Space Bar instead of ↵.
Ctrl-E	Edits current field. See Section 6.4 for a table of key functions. Not allowed on option type fields.

The spreadsheet screens are versatile screens which have many general features as shown in the table above. Some spreadsheet have special functions which can be used only for input or output of a particular group of parameters. These special functions are described in the following paragraphs. A more specific description of some special features is presented in the section of the manual where the input parameters affected by the special function are described.

In many spreadsheet screens, the (Space Bar) key is used as an option key which displays a pop-up menu of acceptable input values. This type of input is used for data entry when the permissible input values are limited to a small set of specific parameters. Input into these columns is accomplished by pressing the (Space Bar) key, which causes a vertical or multi row and column menu to pop-up on the screen with the acceptable input values, and selecting one of the values in the menu using the keystrokes described in Section 6.1 or 6.3. The help message for spreadsheet columns requiring this type of input states "Press the space bar". No direct user entry is allowed into these columns. If the user begins typing, a message appears stating "The key you pressed is not valid here -- try again". If this message appears, the user must leave the cell where the incorrect type of data entry was attempted using the (→) or (←) keys and then return to the cell of interest and press the (Space Bar) for the menu of permissible input values. In some spreadsheet screens, the (Space Bar) is used to display the unabbreviated version of a component type name. The building component type names used in the FACEDAP program are too long to be displayed in full and therefore abbreviated versions of these names are shown in some spreadsheet screens. The help message for these columns tells the user than the (Space Bar) key can be used for this purpose.

On some spreadsheets, certain columns are initialized with default values. The user can accept the default value or, if the column accepts direct user entry, overwrite the default value by typing in a new value. If the column only accepts data from a pop-up menu, a new value may be selected using the (Space Bar) as described above.

In the *Component Properties* spreadsheet screens (which are described in Section 8.5.2), input into some columns can be defined with formulas programmed into the Preprocessor. These columns are identified with an "*" in the column header. The formulas, which are called default formulas, use other input values which are in the same row as the cursor and in columns to the left of the given column. If required data has not been provided in these columns, a message appears stating that required data is 0 or undefined. The default formula calculates an input value and writes it to the column where the cursor is located when the user presses (F7). The use of default formulas is an option and is NOT automatically invoked by the program. The user can simply enter input directly into this column without invoking the (F7) key or they can overwrite an input value calculated with the (F7) key. If any values in a spreadsheet row to the left of a column with a calculated input are redefined, the user must use the (F7) to recalculate the input. It is the user's responsibility to maintain current calculated values in the spreadsheet. The default formulas are shown in Section 8.5.2.

In the *Component Geometry* spreadsheet screen in the Preprocessor (which is described in Section 8.6.3), the function keys (F9) and (F10) are assigned specific tasks. The (F9) key is used to generate components from a "Master" component. (F10) invokes a spreadsheet which allows viewing/editing of the generated components. The Dependency spreadsheet screens use (F9) to generate the dependencies for the current wall or roof area. The user can make edits to the generated component geometries and dependencies, but this should be done after the user is confident that all other input is correct. The FACEDAP program will only automatically generate components and dependencies if the given components or dependencies have not been previously generated. This prevents the program from overwriting user edits to generated component geometries or dependencies. Therefore, the user must either; 1) regenerate components any time a change is made to their "Master" component and regenerate dependencies with F9 any time any component geometries are changed or, 2) propagate the effect of any changes into previously generated component geometries and dependencies with edits. Since the latter process is not usually simple, it is best to delay any editing of generated component geometries or dependencies until the user is confident that all other input is correct.

Special functions that are available on a spreadsheet screen are typically displayed in the upper lefthand and righthand corners of spreadsheet screen. They may also be called out in the help message or in the column header if they are column specific. Mouse support is not functional on the spreadsheets in this version of the FACEDAP program. The mouse can be used on any secondary menus which "pop-up" from the spreadsheet. However, movement on the spreadsheet and exiting the spreadsheet must be done using the keyboard.

7.0 FACEDAP DRIVER PROGRAM

The FACEDAP program is run by typing FACEDAP while in the program directory. The FACEDAP program directory is specified in the Configuration Program, FACECFG. The user can always check all facets of the program configuration, including the designated program directory, by typing FACECFG in the directory where this executable resides. After the command FACEDAP is entered, the title screen appears as shown below.

```
          Facility And Component Explosive Damage Assessment Program
                                  (FACEDAP)

      Developed by:  Southwest Research Institute (SwRI)
                     P.O. Drawer 28510
                     San Antonio, Texas 78228-0510
                     Charles J. Oswald, Program Manager

              For:   U.S. Army Corp of Engineers
                     Omaha District
                     215 North 17th Street
                     Omaha, Nebraska 68102-4978
                     Engineering Division Special Projects ED-5
                     Edward Conrath, Technical Monitor

  Acknowledgement:   The FACEDAP Computer Program was developed
                     using the Blast Damage Assessment Model
                     computer program (BDAMA), which was written
                     by SwRI for the Naval Civil Engineering
                     Laboratory in Port Hueneme, California.

Press any key to continue.
```

By pressing ↵, or any other key, the FACEDAP Main Menu appears.

prevents the program from overwriting user edits to generated component geometries or dependencies. Therefore, the user must either; 1) regenerate components any time a change is made to their "Master" component and regenerate dependencies with F9 any time any component geometries are changed or, 2) propagate the effect of any changes into previously generated component geometries and dependencies with edits. Since the latter process is not usually simple, it is best to delay any editing of generated component geometries or dependencies until the user is confident that all other input is correct.

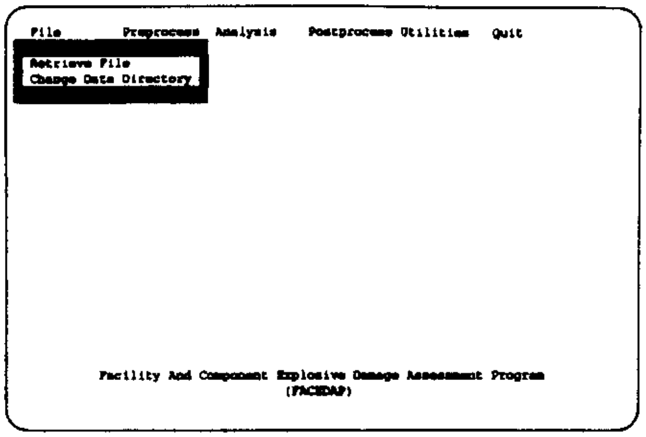
Special functions that are available on a spreadsheet screen are typically displayed in the upper lefthand and righthand corners of spreadsheet screen. They may also be called out in the help message or in the column header if they are column specific. Mouse support is not functional on the spreadsheets in this version of the FACEDAP program. The mouse can be used on any secondary menus which "pop-up" from the spreadsheet. However, movement on the spreadsheet and exiting the spreadsheet must be done using the keyboard.

7.0 FACEDAP DRIVER PROGRAM

The FACEDAP program is run by typing FACEDAP while in the program directory. The FACEDAP program directory is specified in the Configuration Program, FACECFG. The user can always check all facets of the program configuration, including the designated program directory, by typing FACECFG in the directory where this executable resides. After the command FACEDAP is entered, the title screen appears as shown below.

```
        Facility And Component Explosive Damage Assessment Program
                  (FACEDAP - VERSION 1.2 UPDATED 5/20/94)

    Developed by:  Southwest Research Institute (SwRI)
                   P.O. Drawer 28510
                   San Antonio, Texas 78228-0510
                   Charles J. Oswald, Program Manager

            For:   U.S. Army Corp of Engineers
                   Omaha District
                   215 North 17th Street
                   Omaha, Nebraska 68102-4978
                   Engineering Division Special Projects ED-5
                   Edward Conrath, Technical Monitor

 Acknowledgement:  The FACEDAP Computer Program was developed
                   using the Blast Damage Assessment Model
                   computer program (BDAMA), which was written
                   by SwRI for the Naval Civil Engineering
                   Laboratory in Port Hueneme, California.

Press any key to continue.
```

By pressing ↵, or any other key, the FACEDAP Main Menu appears.



Refer to Section 6.2 for a description on how to make selections from the main menu. The screen figure above shows the Main Menu of the FACEDAP program across the top of the screen. As the figure shows, the Main Menu consists of six options. A brief overview of each option follows.

7.1 File Option

The *File* option allows the user to select a previously defined input file. If the desired file is in the default program data directory (defined with the FACECFG executable as described in Section 5.0), it is selected by first choosing the *Retrieve* Option from the *File* submenu, and then using the arrow keys or mouse to designate the file of interest on the multi row and column menu which pops up on the screen. All *.BLG files in the default program data directory are displayed when the *Retrieve* Option is selected. These files contain all the basic building and charge location information which the user inputs into the Preprocessor. The selected file can then be modified or viewed with the Preprocessor option, or it can be used in the Analysis or Postprocessor options. The *Change Data Directory* option enables the user to change the default program data directory. This provides the same service as changing the Data Directory with the Configuration Program, FACECFG, or with the Utilities option in the main menu.

There are two separate types of input files (i.e. *.BLG files), one for input files defining single components and another for input files defining entire buildings. As discussed in Section 1.0 and Section 8, the FACEDAP program can be used to determine blast damage to either single structural components or entire buildings. Input files which define single components begin with the four letter string **"COMP"**, and will be referred to as COMP*.BLG files. Input files which define buildings begin with **any string of letters except "COMP"**. This format allows the program to check whether a retrieved input file contains input which is compatible with the type of analysis that is selected by the user in the preprocessor. The user should only retrieve an input file which will be compatible with the type of analysis that will be selected in the preprocessor.

7.2 Overview of Preprocess Option

The *Preprocess* Module serves three functions. First, it provides access to the Preprocessor, where an input file can be created or modified. This option is implemented by selecting *Input Definition* option from the *Preprocess* submenu. Input definition is described in detail in Section 8.0. If a file was selected with the *Retrieve* option prior entering the Preprocessor, that file will automatically be read into the Preprocessor for user editing and viewing. If no file was selected, the last file handled by the Preprocessor will automatically be retrieved. The Preprocessor also provides the user the means to save and print input files, retrieve other input files, and clear all input. The retrieve capability in the *Input Definition* option is very similar to the *Retrieve* option on the main menu.

Secondly, the *Input Definition* option from the *Preprocess* submenu allows the user to define the type of analysis, either single component analysis or building analysis, that will be performed. The Analysis module will perform the type of analysis defined in the Preprocessor.

Thirdly, the *Preprocess* Module provides access to the *File Validation* option. This option checks a Preprocessor input file for errors, which are categorized as warning and fatal errors. This option is described in detail in Section 9.0. All error messages are displayed to the screen and are automatically written to a file with the user-designated input file save name and the suffix .ERR. The contents of this file can be printed with a print option that is automatically presented to the user when they exit the *File Validation* option. Any fatal errors prevent the file from being "validated". Unless a file is successfully validated it cannot be run through the *Analysis* Module. If the user proceeds directly to the *File Validation* option without selecting a file using the Main Menu *Retrieve* option or saving an input file in the Preprocessor, the user will be provided a list of available input files in the default data directory.

7.3 Overview of Analysis Option

When the *Analysis* option in the main menu is selected, the program calculates blast damage to the single structural component or to the building defined by the user based on the information input into the *Preprocessor* and validated by the *File Validation* Module. The *Analysis* option will read in the input file which was most recently validated. The *Analysis* Module can also be run directly on a retrieved file if the file has been validated previously. If the *Analysis* option is selected prior to any input file definition or file retrieval, a menu is provided listing available files from the default data directory for user selection. These files can only be analyzed for building blast damage if they have been previously validated.

7.4 Overview of Postprocess Option

The *Postprocess* option displays calculated component or building damage information and blast load information. In the case of a building analysis, the damage and blast load that were calculated for each component in the building are also displayed if the appropriate options are selected. The *Postprocess* option will display the blast damage calculated for the latest input file submitted to the *Analysis Module*. The *Postprocess* option also allows the user to designate output information for printing. The information selected for printing will be sent to the printer after the user has exited the *Postprocess* option and returned to the main menu. The user should make certain that a printer and port have been defined prior to selecting any print options. See Sections 7.5 and 3.0 for information on defining printers and ports.

7.5 Utilities Option

The *Utilities* option provides access to the Configuration program, FACECFG, and the DOS shell. Within the *Configuration* option, the user can change or assign printers, communication ports, default program and data directories, screen colors and all other information controlled by the FACECFG program. The input screen for the *Configuration* option is shown in Section 3.0. The *DOS Shell* option enables the user to temporarily leave the FACEDAP program by "shelling" to DOS. Return to FACEDAP is achieved by typing EXIT.

8.0 PREPROCESSOR

In the next sections the Preprocessor, Analysis, and Postprocessor modules are described in detail. These are the major program modules which read in all required input, perform the blast damage analysis, and output the calculated damage information. The Preprocessor is described in this section. As mentioned in Section 7.3, it is invoked by selecting the *Input Definition* option under the *Preprocess* option in the FACEDAP main menu.

The Preprocessor provides an user-friendly method for defining the type of analysis (i.e., single component analysis or complete building analysis) and the structure and explosive threat to be analyzed. The basic input procedure is as follows: 1) define the type of analysis (single component analysis or building analysis); 2) define the explosive charge weight and location; 3) define the geometry and material properties of the single component or building components to be analyzed; 4) define the location of each structural component in a building analysis, its component type (i.e., steel beam), and the name of the set of cross sectional and material properties that define its properties; and 5) define the "dependencies" of each component in a building analysis which describe how building components support, and are supported by, each other. Only one single component or building can be analyzed at a time.

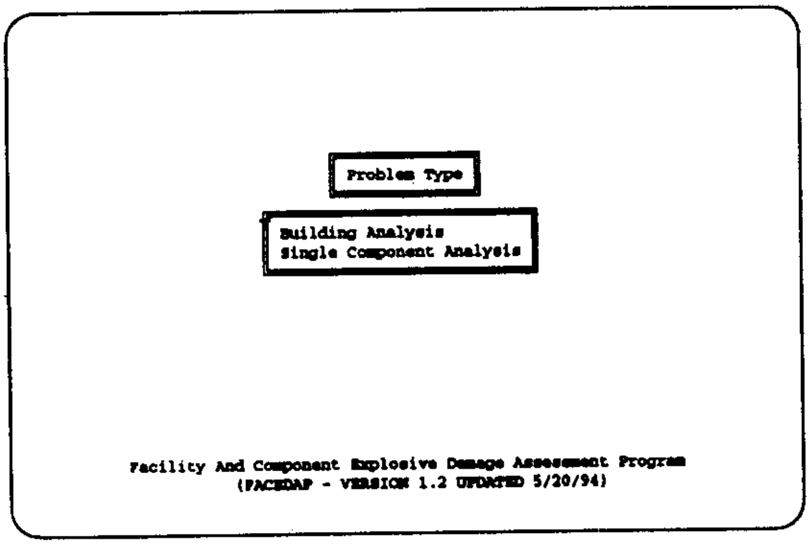
Input into the Preprocessor is facilitated by the user-friendly input screens in this module. Input which defines each set of component properties and the location of building components is read by spreadsheet-type input screens which have many of the capabilities of a typical spreadsheet including limited copying capabilities, the capability to insert and delete lines, and the ability to calculate pre-programmed default material property inputs with a function key. The keystrokes used for each type of input screen are discussed in Section 6.0 and the user is encouraged to return to this section during input to the Preprocessor when necessary. Help messages giving some detailed explanation of each input are shown at the bottom of all input screens. The Preprocessor module also checks for errors or unexpected values in user input and displays a message to the screen whenever an error is found.

There are several other sections of the User's Manual which supplement this section. As mentioned above, Section 6.0 explains the four different types of input screens which accept user input in the Preprocessor. Section 12.0 contains an explanation of all error and warning messages in the Preprocessor. Section 13.0 shows the input of an example problem. Section 14.0 has descriptions of thirteen conventional buildings which have been input into the files EXAMPL1.BLG through EXAMPL13.BLG in the BUILDING.ZIP file on the program disk with the FACEDAP executables. These files can be unarchived as described in Section 3.0. The user can retrieve these files into the program and use the Preprocessor print option to print out the input and then use this information as an additional example problem.

Finally, the user should be aware that input cannot be entered into the Preprocessor in a random fashion. Because the Preprocessor "participates" in the input by issuing the user many prompts, by making a number of checks and, in some cases, by generating input for the user, the user must enter input in a structured manner. **Input definition should be entered in the order shown on the main Preprocessor menus in Sections 8.0.1.1 and 8.2.**

8.0.1 Selecting Single Component or Building Type of Analysis

When the *Input Definition* option under the *Preprocess* option in the FACEDAP Main Menu is selected, the following menu is displayed. This is the Problem Type menu.



The user should select either the Single Component Analysis option, which will determine blast damage to a single structural component, or the Building Analysis option, which will determine blast damage to an entire building. The Building Analysis option calculates damage to each structural component in a building, including secondary damage caused by failure of supporting components, and therefore requires much more input by the user. The Single Component Analysis option is useful in cases where previous experience and engineering judgement make it possible to select a few "representative" building components for use in making a first-cut determination of building damage. This option is also useful for focusing on only those structural components in a building which directly protect a given asset against a given explosive threat.

The problem type selected by the user should be compatible with the most recent retrieved input file or with any input currently in the preprocessor. Usually this only requires that the most recent retrieved input file is compatible with the desired type of analysis as explained in Section 7.1. However, if the user has been defining building input in the Preprocessor and decides to switch to a single component analysis (or vice versa), the following sequence should be invoked prior to selecting the new problem type: 1) the previous input should be saved; and then, 2) either a compatible input file (i.e. a COMP* file as explained in Section 7.1 if the new problem type will be single component) should be retrieved with the *Retrieve* option in the Preprocessor, or the building analysis input should be cleared with the *Clear Forms* option. The *Retrieve* and *Clear Forms* options are explained in Sections 8.8 and 8.9.

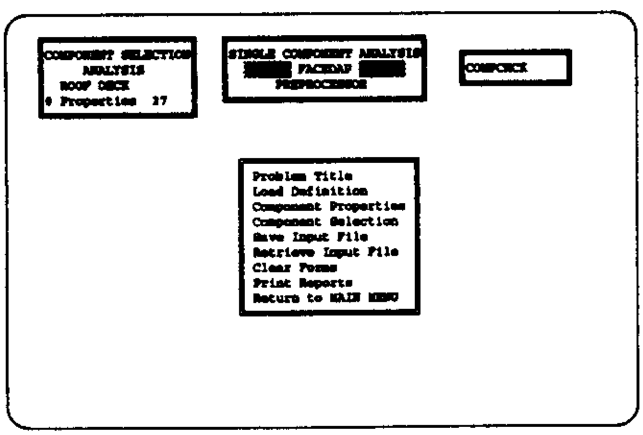
If the most recent retrieved input file is incompatible with the selected type of analysis, the program will issue a explanatory message and the incompatible file will not be read into the Preprocessor. Instead, the Preprocessor will read the name of the last compatible input file that was processed from a session file stored on the hard drive by the FACEDAP program and then it will read this input file into the Preprocessor. If no filename is stored in the session file, the Preprocessor will have blank input screens.

8.0.2 Preprocessor Input for Single Component Analysis

The Single Component Preprocessor provides a user friendly manner for defining a single component and explosive threat as required by the *Analysis* Module. The basic input procedure is as follows: 1) define the explosive charge weight, the standoff between the charge and component, and the type of blast pressure applied to the component (i.e. fully reflected or free-field blast pressure); 2) define the span length and cross sectional properties of the component(s) of interest; 3) select the component to be analyzed, assuming that the user may want to define a number of single components and then select them one-by-one for analysis. Only one component may be analyzed for blast damage at a time in the *Analysis Module.* Refer to Table 8, Section 8.1.1 for a table showing the types of structural components which can be analyzed with the FACEDAP program. The following description of the Single Component input refers to sections of the manual describing Building Analysis input because the Single Component Analysis option was not originally part of the FACEDAP program. As the user will probably notice, the flow of the manual favors the Building Analysis option for this reason.

8.0.2.1 Main Single Component Preprocessor Menu

When the *Input Definition* option under the *Preprocess* option in the FACEDAP Main Menu is selected, followed by selection of *Single Component Analysis* problem type, the following menu is displayed. This is the Main Single Component Preprocessor Menu.



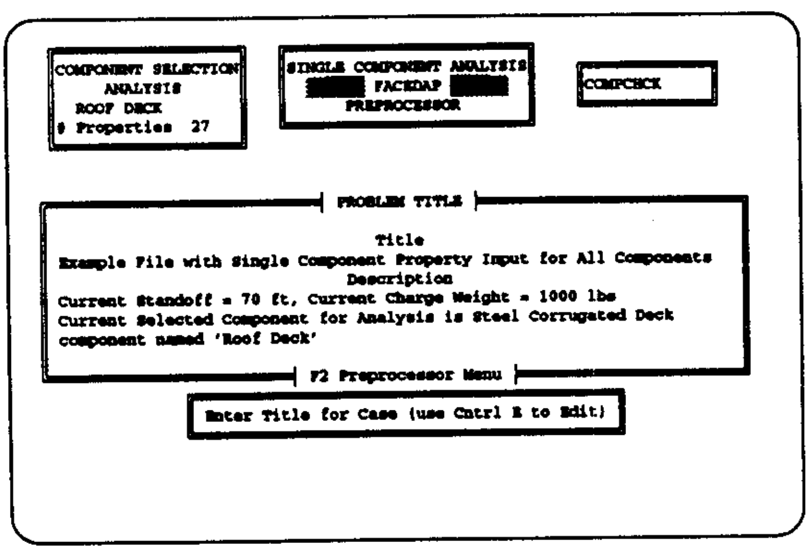
As explained in Section 6.1, the user can use the first letter or the arrow keys to highlight options on this menu and then ⏎ to select the option. Alternatively, the left mouse button can be used to both highlight and select an option.

This menu is used to define the single component to be analyzed and the explosive threat by selecting the options off the menu in a top-to-bottom sequence. It is strongly suggested that **input definition should be entered in the order shown on the main Preprocessor menu.** Some of the input, particularly the *Component Selection* section, must be compatible with the most current input into other sections preceding it in the menu.

The preceding screen print also shows a box in the top right hand corner. This box indicates the name of the name of the component currently selected for analysis with the *Component Selection* option, as well as the total number of component properties currently defined under the *Component Properties* option. It is always displayed by all screens which pertain to Single Component Analysis input.

8.0.2.2 Problem Title

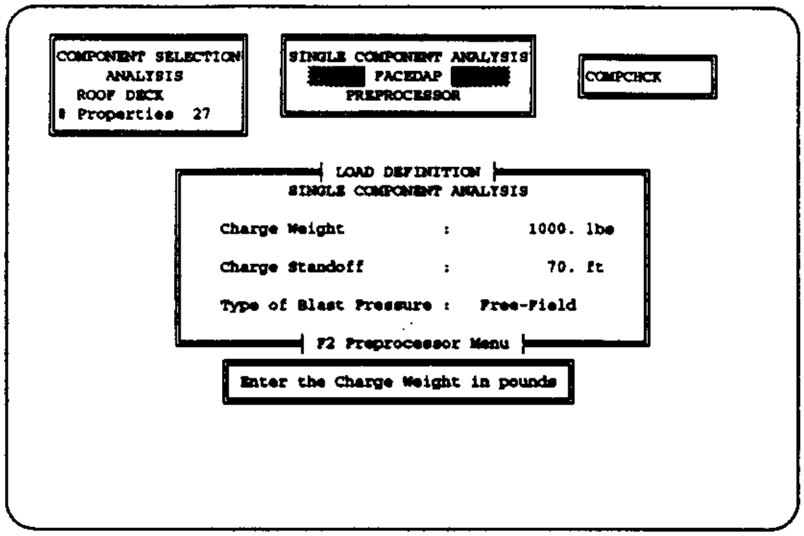
The first option on the Main Single Component Preprocessor Menu, *Problem Title*, defines the problem title and description. A one line title and a three line problem description can be specified with this option. Each line has a maximum of 70 characters. The problem title specified here will appear on the Error Report of the Validation Program and the Preprocessor and Postprocessor Print Reports. The following screen illustrates the form screen provided for entry of this information.



Line editing is invoked with the (Ctrl-E) key. The cursor keys and ⏎ traverse the form. More detailed information on entering data into this form screen is provided in Section 6.4. Pressing (F2) or clicking the left mouse button on the F2 area of the form, returns control to the Single Component Preprocessor Main Menu. **The (F2) key is typically the escape key off all form and spreadsheet screens in the FACEDAP program.**

8.0.2.3 Load Definition

Load Definition is the second option on the Main Single Component Preprocessor Menu. Input into the *Load Definition* section defines the charge weight, the charge standoff and the blast pressure type. The input is entered into a form-type input screen. Section 6.4 describes the keystrokes used to traverse this form screen. The F2 key returns control to the Single Component Preprocessor Main Menu. A sample *Load Definition* form follows. A warning message is issued on exiting the *Load Definition* if any fields are left blank. The input charge weight and standoff must be greater than zero.



The standoff is the distance from the center of the explosive threat to the center of the component. The blast load is assumed to be uniform over the component equal to the value calculated at the center of the component. Also, the blast damage calculation procedure in the *Analysis* module assumes that the blast load is applied from a surface burst of the explosive. This means that it is assumed that the distance between the explosive charge and the ground surface is small compared to the distance from the charge to the component. The type of blast pressure must be defined as either free-field or fully reflected by the user. If the angle of incidence between the explosive charge and the unit normal from the surface at the center of the component is less than 45 degrees, selection of fully reflected blast pressure is recommended. Otherwise, selection of free-field blast pressure is recommended. See Section 3.0 of the Theory Manual[2] for an illustration showing the angle of incidence and more discussion of free-field and reflected blast pressures.

It is recommended that the user only analyzes cases where the scaled standoff is between 3.0 $ft/lb^{1/3}$ and 100.0 $ft/lb^{1/3}$. The scaled standoff is the distance between the charge and the component (in the units of feet) divided by the cube root of the charge weight (in the units of pounds). The minimum recommended scaled standoff helps ensure that the blast load applied to the component is uniform over the whole component, which is the assumption in the blast damage calculation procedure in the *Analysis* module. It will also help ensure that the program is not used to analyze cases where local shear failure or local damage (such as spalling or breach of concrete components) occurs and that the load distribution over the component is relatively uniform. Only flexural response or buckling response is considered in the program. The maximum recommended scaled standoff is the upper bound value for the curve-fit equations that are used in the *Analysis* module to calculate blast loads on the center point of the component. The program will allow input of cases where the scaled standoff is as low as 1.0 $ft/lb^{1/3}$ but the user should be aware the blast damage of the component may not be accurately calculated if the scaled standoff is less than 3.0 $ft/lb^{1/3}$. The validation module will issue a fatal error message, which precludes entry into the Analysis module, if the scaled standoff is less than 1.0 $ft/lb^{1/3}$. A warning message will be issued if the scaled standoff is less than 3.0 $ft/lb^{1/3}$.

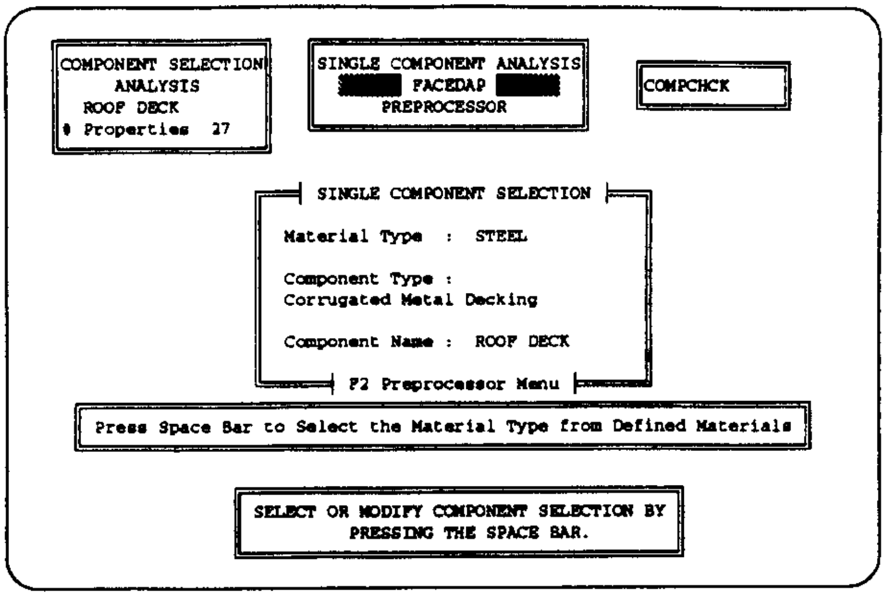
8.0.2.4 Component Properties

The user should select the *Component Properties* option in the Main Single Component Preprocessor Menu to define the geometry and material properties of single component(s) of interest. Multiple components can be defined, and then selected of interest one at a time based on user defined component property set names, with the *Component Selection* option for analysis by the *Analysis Module*. A user may want to always save Single Component input into the same file so that they can build up a large set of defined component property sets for later reference. The copying capabilities incorporated into the Component Properties input screens (see Section 6.5) can be used to define a new component property set using existing input. However, a maximum of 150 components can be defined in any input file. Also, a maximum of 100 component properties of any given component type can be defined in an input file.

The component property input screens are accessed by first selecting the type of material and then the desired component type from multi-row input screens which are displayed after the *Component Properties* option is selected. This procedure is identical to that described in Section 8.5 for input of component properties for a Building Analysis. Therefore, the user is referred to Section 8.5 for a complete description of Component Property input subject to the following exceptions for Single Component input. First, the weighting factor is not a required component property input for Single Component Analysis. Secondly, span length input is required. The input span length should be equal to the distance between the points where the component is supported against the applied blast pressures.

8.0.2.5 Component Selection

Following the input of the component properties, the user must select the component property set defining the component which will be analyzed in the *Analysis Module*. If only one component property set was defined in the *Component Properties* phase, this property set is assumed to be the selected set. If however, more than one component property set is defined, the user must select the desired set using the *Component Selection* option. Choosing *Component Selection* from the Main Single Component Preprocessor Menu causes the following screen to be displayed.



As the screen indicates, the user selects a given component property set by first selecting the type of material used to construct the component to be analyzed (i.e., concrete, steel, masonry, or wood), then selecting the type of the component (e.g. reinforced concrete beam), and finally selecting the user defined name of the property set defining the component of interest. This name was assigned in the Component Properties definition. Pressing the (Space Bar) or clicking the right mouse button, results in a pop-up list of available choices for each particular item. This multi-phase input procedure allows the program to prompt the user for the desired component property set name rather than requiring the user to write down the name or to remember the exact spelling. Information should only be input by selecting off the pop-up lists. Also, the user should always select, or at least check, each of the three parameters defining the selected component property set in a sequential manner starting at the top of the screen. The Component (Property Set) Name, the Component Type, and the Material Type must always be compatible (e.g. the selected component type must always be constructed with the selected material and the selected component name must be defined on the component property input screen of the selected component type). Warning messages are issued if a "mismatch" occurs. Although the errors are only warnings within the Preprocessor, these errors will prove FATAL in the Validation Process. Thus, these errors should be corrected as soon as they are detected.

8.0.2.6 Save, Retrieve, Clear, Print and Return Options

The *Save, Retrieve, Clear, Print* and *Return* options on the Single Component Preprocessor *Main Menu* are similar enough to those in the Building Analysis portion of the Preprocessor so that the user is referred to Sections 8.7 through 8.11 for a discussion of these options subject to the considerations in the following paragraph. As mentioned previously, the Single Component Analysis option was not originally part of the FACEDAP program, but added at a later time and, therefore, reference is made here to previously written discussion in the Building Analysis section.

The user is once again reminded that all input files storing information related to Single Component analysis must begin with the four letter string "COMP" (i.e., a COMP* file). Therefore, the user should save all single component input into a file beginning with these four letters and retrieve only such files into the preprocessor when the Single Component Analysis option has been invoked. Retrieval of any other type of file results in a warning message and the retrieval of a default file (the last file which was saved when the Single Component Analysis option was invoked). No conversion feature has been developed at this time for converting a building file into a single component file.

8.1 Preprocessor Input for Building Analysis

The required input for a building analysis can be quite labor intensive, especially for a large building with many components. Therefore, the Preprocessor has been designed to reduce the amount of required user input to a minimum level. It does this in part by taking advantage of the fact that most buildings are comprised of a relatively small number of typical building components which are used repetitively throughout the building construction. For example, the Preprocessor generates the information defining components used repeatedly in the building based on the endpoints and properties of a "master" component. It also allows the user to assign a single set of cross sectional and material properties to multiple components throughout the building rather than requiring separate input for each component. In addition, the Preprocessor reads the endpoints and corner points defining the location of components in terms of two local coordinates defined within each wall and roof area, rather than requiring input in terms of the global three-dimensional coordinate system, and it generates dependencies between building components which are required in the building damage calculation scheme. The dependencies define how building components support, and are supported by, each other.

All the following sections of the Users Manual which describe the Preprocessor refer to input for the Building Analysis option, even though this is not specifically stated in each section. As mentioned previously, the Single Component option was not originally part of the FACEDAP program. Therefore, the following sections were written assuming that the Building Analysis option was the only option and they have not been updated.

8.1.1 Types of Structural Components Which can be Analyzed

Only components on the exterior of the building which are directly loaded by a blast load, or components which directly support exterior components (such as roof columns), should be input. Lower level interior columns can also be damaged by the blast load applied to the roof but since these columns are generally at least as strong as the columns at roof level, they can be considered adequate if the roof level columns do not fail. Also, only components which can be categorized as one of the twenty-four component types shown in Table 8 can be analyzed with the program. If an exterior component in the building is not one of the twenty-four different types considered by the program, then the user must use another method to analyze the blast damage to the particular component or the component must be idealized as an "equivalent" component which is considered by the program. An understanding of structural dynamics and the theory used by the FACEDAP program is necessary in order to determine equivalent properties for a component type other than those shown in the following table. The FACEDAP theory manual[2] discusses the theoretical basis of the code.

Table 8. Structural Components Considered in the FACEDAP Program

Concrete Components	Steel Components	Masonry Components	Wood Components
R/C Beams	Steel Beams	One-Way Unreinforced Masonry	Wood Stud Walls
R/C One-Way Slabs	Metal Stud Walls	Two-Way Unreinforced Masonry	Wood Roofs
R/C Two-Way Slabs	Open Web Steel Joists (bending response)	One-Way Reinforced Masonry	Wood Beams
R/C Exterior Column (bending response)	Corrugated Metal Deck	Two-Way Reinforced Masonry	Wood Exterior Columns
R/C Interior Column (buckling response)	Steel Exterior Columns (bending response)	Masonry Pilasters	Wood Interior Columns
R/C Frames (lateral frame sway)	Steel Interior Columns (buckling response)	-	-
Prestressed Beams	Steel Frames (lateral frame sway)	-	-

As the table above shows, doors and windows are not analyzed by the FACEDAP program. These components can be analyzed separately, using computer programs such as that in Reference 5, or they can be simplistically assumed to fail if the calculated blast pressure exceeds a nominal amount such as 2 psi. Window and door failure can cause significant injury to building inhabitants.

8.1.2 Preparation of Component Property Input

Once all the relevant components in the building have been identified, these components should be grouped in terms of their component type. Then, the cross sectional and material properties of the components within each group should be examined and the components with identical cross sectional and material properties should be separated into subgroups. These subgroups are the unique material and cross sectional property sets which must be input into the FACEDAP program. A descriptive component property name should be assigned to each subgroup which the user can refer to during program input. In some cases total weight is a required component property input and, in these cases, two components with identical cross sections and material properties but different length cannot have the same component property set. The specific cross sectional and material property inputs required for each component type are shown in table format and by example in Section 8.5.2.

8.1.3 Preparation of Component Geometry Input

An general description of component property definition is given here and specific input instructions are given in Section 8.6. Component geometry is defined in a multi-step process where the building is first divided into large planar wall/roof areas by the user, secondly the geometry of

Table 8. Structural Components Considered in the FACEDAP Program

Concrete Components	Steel Components	Masonry Components	Wood Components
R/C Beams	Steel Beams	One-Way Unreinforced Masonry	Wood Stud Walls
R/C One-Way Slabs	Metal Stud Walls	Two-Way Unreinforced Masonry	Wood Roofs
R/C Two-Way Slabs	Open Web Steel Joists (bending response)	One-Way Reinforced Masonry	Wood Beams
R/C Exterior Column (bending response)	Corrugated Metal Deck	Two-Way Reinforced Masonry	Wood Exterior Columns
R/C Interior Column (buckling response)	Steel Exterior Columns (bending response)	Masonry Pilasters	Wood Interior Columns
R/C Frames (lateral frame sway)	Steel Interior Columns (buckling response)	-	-
Prestressed Beams	Steel Frames (lateral frame sway)	-	-

As the table above shows, doors and windows are not analyzed by the FACEDAP program. These components can be analyzed separately, using computer programs such as that in Reference 5, or they can be simplistically assumed to fail if the calculated blast pressure exceeds a nominal amount such as 2 psi. Window and door failure can cause significant injury to building inhabitants.

8.1.2 Preparation of Component Property Input

Once all the relevant components in the building have been identified, these components should be grouped in terms of their component type. Then, the cross sectional and material properties of the components within each group should be examined and the components with identical cross sectional and material properties should be separated into subgroups. These subgroups are the unique material and cross sectional property sets which must be input into the FACEDAP program. A descriptive component property name should be assigned to each subgroup which the user can refer to during program input. In some cases total weight is a required component property input and, in these cases, two components with identical cross sections and material properties but different length cannot have the same component property set. The specific cross sectional and material property inputs required for each component type are shown in table format and by example in Section 8.5.2.

8.1.3 Preparation of Component Geometry Input

An general description of component property definition is given here and specific input instructions are given in Section 8.6. Component geometry is defined in a multi-step process where the building is first divided into large planar wall/roof areas by the user, secondly the geometry of building components within these areas are defined, and finally the dependencies between components are defined. Large, planar wall and roof areas are defined in the first step. Usually, each of the four walls and the roof of a building are designated by the user as separate wall/roof areas. One edge of each wall/roof area is designated by the user as the local x axis for that wall/roof area and one corner is designed as (0,0) in the local wall/roof area coordinate system. The local y axis is perpendicular to the x axis and is positive in the direction into the wall/roof area. This defines the local coordinate system of the wall/roof areas. The user inputs the endpoints or corner points of all building components within each wall/roof area in terms of the local coordinate system. This allows the user to input the endpoints and corner points of component directly off typical engineering drawings or plan and elevation sketches without considering the global coordinate system.

During preparation of building component input, the user should identify and name each wall and roof area (i.e., South Wall) which will be used. Next, the user should identify the corner of each wall and roof area which will be the (0,0) point in the local wall/roof coordinate system and the side of the wall or roof intersecting the (0,0) corner which will be the local x axis. As mentioned previously, the Preprocessor will generate the end points or corner points of a group of components which are evenly spaced within a wall/roof area based on the input of one component in the group called the "Master" component. Therefore, the user should examine the components within each wall and roof area for groups of similar components offset at a typical spacing.

8.1.4 Preparation of Building Geometry and Charge Location Input

Finally, the user should set up the global coordinate system which will be used during input and define the global coordinates of the corners of the building and the center of the explosive charge. The global coordinates of the building corners are used to define the global coordinates of each corner of the designated wall/roof areas. *There are only two rules which govern the choice of a global coordinate system; 1) the global X and Y axes must lie in the plane of the ground surface, and 2) the global Z axis must have its zero point at the ground surface and be positive upward from the ground.* Within these two restrictions, any convenient point (such as the point on the ground beneath the charge) can be designated (0,0,0) in the global coordinate system and the global coordinates of the corners of the building and the center of the explosive charge can then be chosen to cause the desired location of the charge relative to the building.

8.1.5 Summary of Problem Preparation

In summary, the user should perform the following tasks, as described above, before entering the Preprocessor.

- Identify all building components which will be input into the FACEDAP program
- Separate the components by component type. Only the twenty-four component types in Table 8 can be analyzed by the program.

- Identify groups of components within each component type with identical cross sectional and material properties and assign a short (10 letter maximum) descriptive name to each subgroup. The set of cross sectional and material properties which are relevant for each component type is defined in Section 8.5.2.

- Divide the building into large, planar wall/roof areas (usually one area per wall and one for the roof) and assign a short, descriptive name to each area (i.e., South Wall).

- Identify a convenient corner of each wall/roof area which will be (0,0) in the local coordinate system of the wall/roof area and identify which side of the wall or roof intersecting the chosen corner will be the local x axis. A convenient approach is to choose the bottom side of wall areas (along the ground surface) as the local x axis and to choose the side of roof areas closest to the explosive charge as the local x axis. It is highly recommended that the user chooses a system that can be applied as uniformly as possible to all walls or roofs since this will significantly reduce input error. Figure 2 in Section 8.6.2 illustrates how a building can be divided into wall/roof areas.

- Identify groups of similar components within each wall/roof area which are spaced at a typical spacing in either the local x or local y direction. The component in these groups which is closest to the (0,0) point in the local coordinate system can be defined during input as a "Master" component and used to generate the required geometry input for all the other components in the group.

- Choose a convenient global coordinate system keeping in mind the two restrictions discussed in Section 8.1.4 and define the global coordinates of the center of the explosive charge and the corners of the building.

8.1.6 Maximum Limits on Input Building Component Data

Care should be exercised during input to make sure that none of the program limits are exceeded. Table 9 shows all the program limitations.

Table 9. Maximum Values for Key FACEDAP Program Inputs

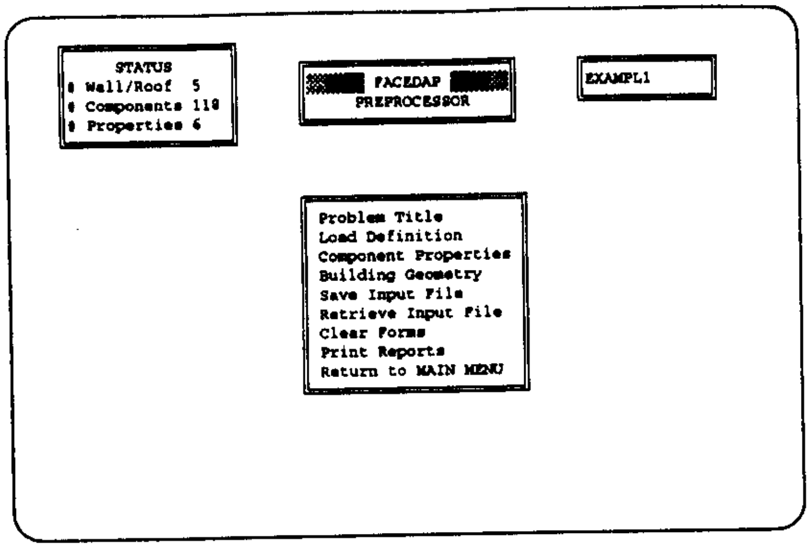
INPUT VALUE DESCRIPTION	MAXIMUM VALUE
Total Components	370
Sets of Component Properties (Total)	150
Component Property Sets per Component Type	100
Wall/Roof Areas	50
"Master" Components defined within a Wall/Roof Area	150
Number of Generated Components from a "Master" Component	50
Total Dependency Pairs (Independent Component, Dependent Component)	200

If any of the maximum limits on total values shown above are exceeded, it is recommended that input be limited to "representative" building components, rather than including all building components. It is not generally necessary to input every component in a large building. The user can input only a portion of the secondary components (i.e., cladding components, girts, etc.) in a large structure (every third such component, for example) AND increase the weighting factors (see Section 8.5.2) for these components accordingly (by a factor of three if every third component is input). The damage to all components will be represented in the overall building damage calculations since; 1) the increased weighting factor causes the "weighted" damage to the input component to be equal to the sum of the damage to all the components which it represents, and 2) the input secondary components are representatively spaced around the structure and therefore they should sustain the same damage as the components they are representing. The weighted component damage, which is the product of the calculated component damage level and the component weighting factor, is used in the building damage summation algorithm to determine the percentage of building damage. It is also important that "pairs" of secondary components are entered together. For example, if only representative roof panels and roof joists are entered, the joists which support the input panels should be entered, so that component dependencies are correctly accounted for. The concepts referred to here, such as dependencies, weighting factors, and the building damage calculation, are discussed in the following sections of the manual. Also, this approach is used to enter the secondary roof components in Example Building No. 1 in Section 14.0. The input file (EXAMPL1.BLG) for this example building is in the BUILDING.ARC file on the program disk. See Section 3.0 for instructions on accessing this file.

A more time consuming alternative is to increase the parameter values and array dimensions in the source code which control the maximum limit of interest and then recompile and relink the program. This alternative is discussed in the FACEDAP Programmer's Manual[1].

8.2 Main Preprocessor Menu

When the *Input Definition* option under the *Preprocess* option in the FACEDAP Main Menu is selected, the following menu is displayed. This is the main Preprocessor menu.



As explained in Section 6.1, the user can use the first letter or the arrow keys to highlight options on this menu and then ↵ to select the option. Alternatively, the left mouse button can be used to both highlight and select an option.

If a file was retrieved with the *File* option in the main FACEDAP program menu, the Preprocessor automatically reads the retrieved file. If no file was selected, the Preprocessor reads the name of the last input file that was processed from the session file, BDAM.SES, and then reads this input file. All Preprocessor files have the extension .BLG. This file is both the input and output file of the Preprocessor. If no filename is stored in the BDAM.SES file, the Preprocessor will have blank input screens. As explained in Sections 8.8 and 8.9, all screens can be blanked or an input file can be retrieved using options in the main Preprocessor menu.

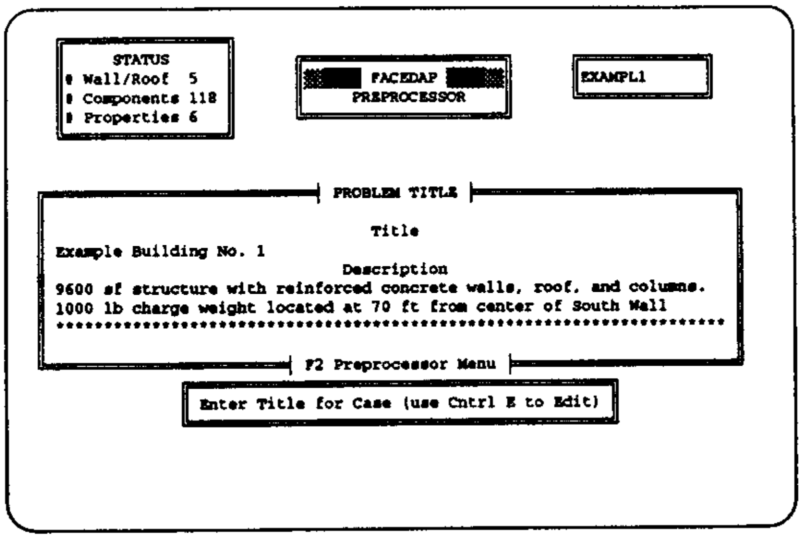
In addition to the menu which appears on the screen, a Status box is shown. Table 10 below describes each of the fields in this box.

Table 10. Description of Status Box

FIELD TITLE	DESCRIPTION
Wall/Roof	Total number of Wall/Roof Areas defined in the building
Components	Total number of Building Components defined in the building
Properties	Total number of Component Property sets defined in the building

8.3 Problem Title

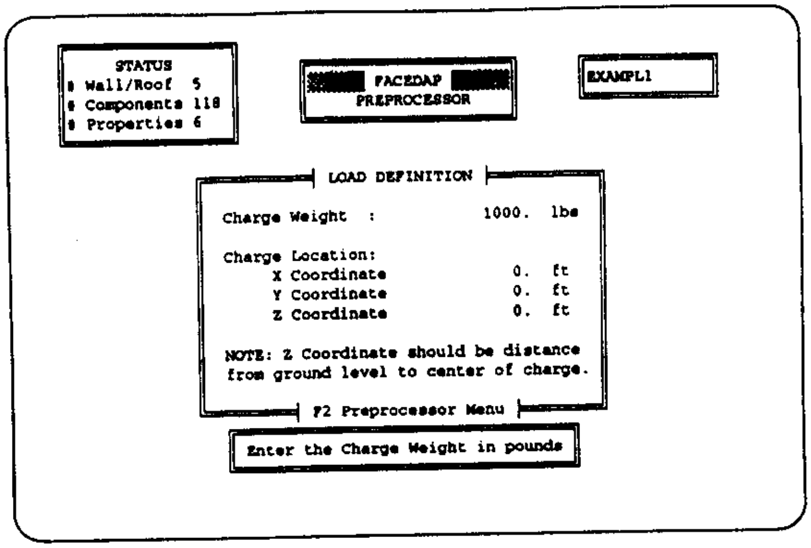
The first option on the main Preprocessor menu, *Problem Title*, defines the problem title and description. A one line title and a three line problem description can be specified with this option. Each line has a maximum of 70 characters. The problem title specified here will appear on the Error Report of the Validation Program and the Preprocessor and Postprocessor Print Reports. The following screen illustrates the form screen provided for entry of this information.



Line editing is invoked with the [Ctrl-E] key. The cursor keys and [↵] traverse the form. More detailed information on entering data into this form screen is provided in Section 6.4. Pressing [F2] or clicking the left mouse button on the F2 area of the form, returns control to the Preprocessor Main Menu. **The [F2] key is typically the escape key off all form and spreadsheet screens in the FACEDAP program.**

8.4 Load Definition

Load Definition is the second option on the main Preprocessor menu. Input into the *Load Definition* section defines the charge weight and its location in the global coordinate system. The input is entered into a form-type input screen. The global coordinate system and the factors affecting the choice of the charge location are discussed in Section 8.1.4. Section 6.4 describes the keystrokes used to traverse this form screen. The [F2] key returns control to the Preprocessor Main Menu. A sample *Load Definition* form follows. A warning message is issued on exiting the *Load Definition* if any fields are left blank.



There are several restrictions on the input charge location. First, the input charge location must be **outside** the building which is analyzed for blast damage and it must not be directly over the building roof. Secondly, it is recommended that the user only analyzes cases where the scaled standoff between the charge and the building is at least 3.0 ft/lb$^{1/3}$. The scaled standoff is the minimum distance between the charge and the building (in the units of feet) divided by the cube root of the charge weight (in the units of pounds). The minimum recommended scaled standoff helps ensure that the blast load applied to each building component is uniform over the whole component, which is the assumption in the blast damage calculation procedure in the *Analysis* module. It will also help ensure that the program is not used to analyze cases where shear failure or local damage (such as spalling or breach of concrete components) occurs. Only flexural response or buckling response is considered in the program. The program will allow input of cases where the scaled standoff is as low as 1.0 ft/lb$^{1/3}$ but the user should be aware the blast damage of building components closest to the explosive may not be accurately calculated if the scaled standoff is less than 3.0 ft/lb$^{1/3}$. Also, the blast damage calculation procedure in the *Analysis* module assumes that the blast load is applied from a surface burst of the explosive. This means that it is assumed that the distance between the explosive charge and the ground surface is very small compared to the distance from the charge to the building. The validation module will issue a fatal error message, which precludes entry into the Analysis module, if the scaled standoff is less than 1.0 ft/lb$^{1/3}$ or if the charge location is inside the building or above the building roof. A warning message will be issued if the scaled standoff is less than 3.0 ft/lb$^{1/3}$.

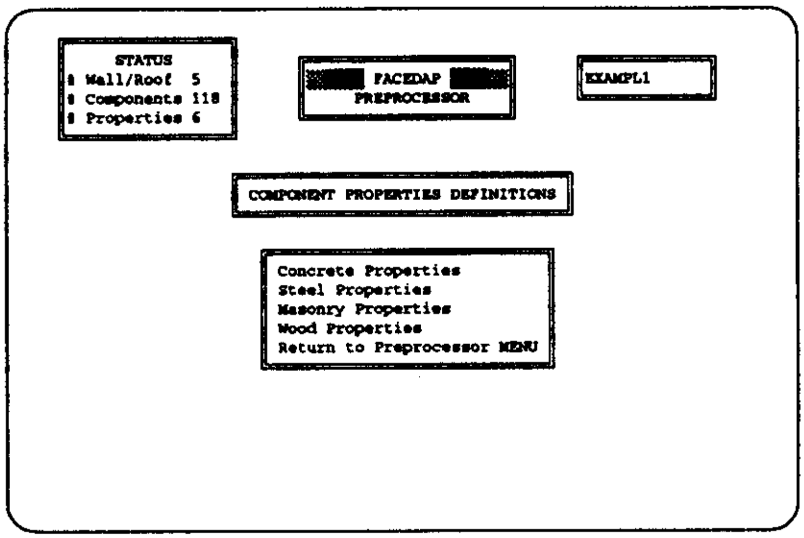
8.5 Component Properties

Component Properties is the third option on the main Preprocessor menu. Input into the *Component Properties* section defines the cross sectional and material properties of all components in the building in terms of unique component property sets which are common to one or more building components. Input is entered into separate spreadsheet screens for each component type. The preparation and organization of building components required prior to input of component properties is described in Section 8.1.2. As this section explains, the user must group the building components by component type and then identify groups of components within each component type with identical cross sectional and material properties. The user must refer to the tables in

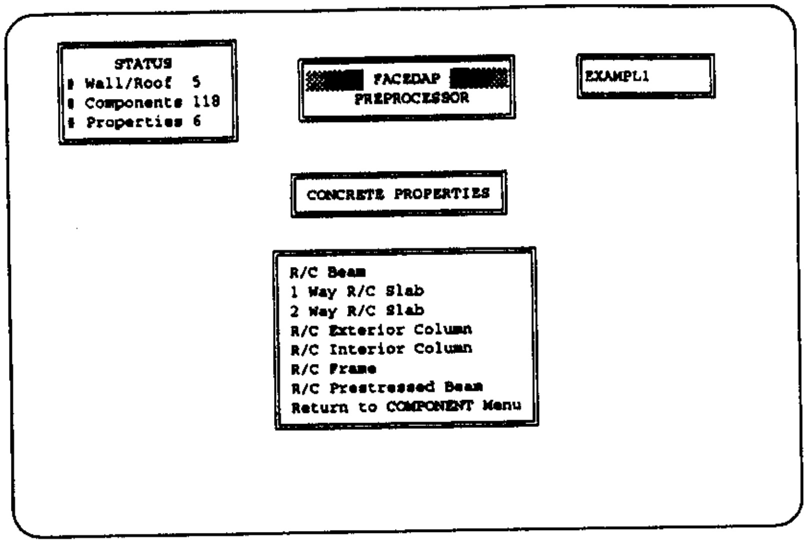
Section 8.5.2 to see the particular property inputs required for each component type. Each of these subgroups is assigned a short, descriptive name called the Component Property Name. A set of cross sectional and material properties which is common to several building components is entered only once into the Preprocessor and the designated Component Property Name is assigned to all the components in the building with this set of properties. A maximum of 100 component property sets can be defined for a given component type and a maximum of 150 component property sets can be defined for a building.

8.5.1 Selecting Component Types for Component Property Input

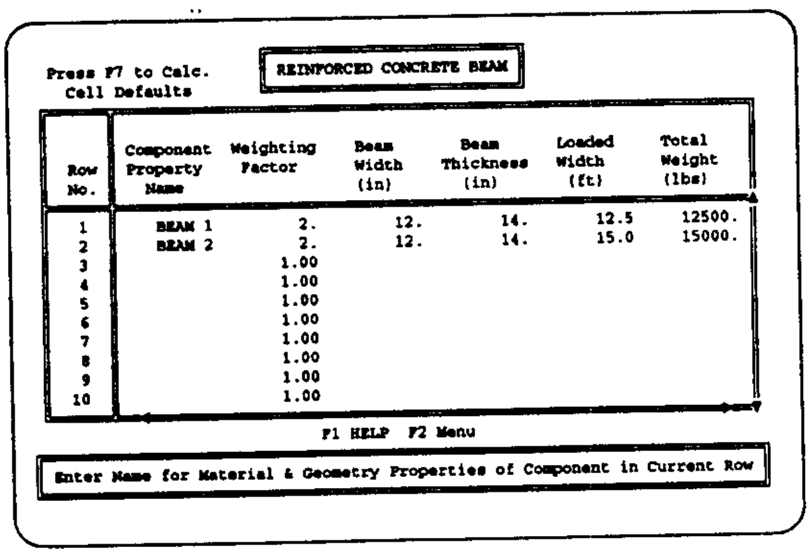
After the Component Properties option is selected, a series of submenus are displayed. The first submenu lists the four material types which the 24 component types are constructed from; concrete, steel, masonry and wood. A screen showing this submenu follows. Section 6.2 discusses how a vertical menu, such as that which follows, is used.



A selection of any of the four options on this menu results in the appearance of a second submenu. This submenu has a list of all component types constructed from the selected material type. The secondary menu for the *Concrete* material type follows. Table 8 shows the 24 component types in columns under the material type they are constructed with.



After the desired component type is selected, a spreadsheet appears which has a column for each component property input required for the selected component type. A separate spreadsheet, with a different set of required inputs, exists for each component type. The input for each unique set of properties for the selected component type are input into separate rows of the spreadsheet. The Component Property Name input into the first column of each row is assigned to the property set input into that spreadsheet row. A sample of the Component Property input screen for the component type "Concrete Beam" follows.



Several special features of the Components Properties input screens require some explanation. First, many input properties have **default values** which automatically are written to the screen by the Preprocessor. For example, the Weighting Factor in Column 2 of the preceding screen has a default value of 1.0 which is automatically written to the screen. These values can be overwritten by simply placing the cursor in that data field and entering the correct input value. Secondly, the F7 key can be used to **calculate input properties** when there is an asterisk in the column header for the input value. The formulas used for these calculations (which are called default formulas) are shown for each applicable component type in Section 8.5.2. The default formulas use input values in the same row of the spreadsheet as the cursor and in columns to the left of the column of interest. Therefore, all columns to the left of a column should be defined by the user before the F7 key is used to calculate the value. Finally, the (Space Bar) key is used to access pop-up menus for user selection in columns with help messages which say "Press the Space Bar". Input into these columns is only permitted when a selection is made from the pop-up menu or when the F8 key is used to copy input down from the previous row.

Refer to Section 6.5 for more discussion of spreadsheet type input screens. There are several cautions the user should keep in mind during input of component properties. First, **Component Property Names assigned within a spreadsheet must be unique within that spreadsheet.** Secondly, the first row containing a blank entry for the Component Property Name marks the end of data for that spreadsheet. Any rows and data entered after this blank entry will be ignored.

8.5.2 Description of Required Component Property Input for Each Component Type

Subsections containing descriptions and examples of all required component property input values for each of the twenty-four component types follow. These sections also include the default formulas which can be used to calculate the values of input properties in spreadsheet columns that have an asterisk in the headers. *It is very important to look carefully at the required units for each component property input. All inputs MUST be in terms of these units.* All input properties must be positive, nonzero values except as noted for wood walls and wood roofs. Also, many input properties are dependent on other input properties. For example, required input cross sectional properties, such as the moment of inertia and section modulus, are functions of the cross section width and depth. Therefore, the input moment of inertia and section modulus should be consistent with the input cross sectional dimensions. The user is responsible for ensuring that all input values are self-consistent. Values calculated with the F7 key will be consistent with other input values since they are calculated directly from these values as explained previously.

Several component property inputs some require additional discussion. First, possible **support conditions** generally include **simple and fixed supports.** This refers to cases where component support rotations are prevented (fixed), or are not prevented (simple supports). Generally, if a component is continuous over a support, or is rigidly connected to a similar component which is either very stiff or also exposed to blast load, then the supports of the component can be considered fixed. Most other support conditions are assumed to be simply supported, which is the more conservative assumption (results in greater blast damage). For interior columns, the user must also decide whether **sidesway** (lateral movement of the top of a column relative to its base) is prevented or not. Sidesway is prevented in buildings that have concrete, masonry, or metal stud shear walls in two perpendicular directions and a concrete roof or diagonal cross bracing across the roof. It is also prevented in buildings with lateral cross bracing in the walls and cross bracing in the plane of the roof. Buildings with moment resisting frames allow sidesway in at least one direction. Usually, roof level interior columns are conservatively considered as simply supported.

Secondly, the effect of **arching and tension membrane** response can be considered for some concrete and steel components. Less blast damage occurs during these types of response than during purely flexural response. However, these types of response only occur if the loading conditions and support conditions prevent in-plane support movement. Arching can usually be assumed to occur during the response of a concrete or masonry panel to blast loading if the panel is enclosed or framed by column and beam components and there is a very tight fit between the component and the surrounding columns and beam. Tension membrane response in steel beams can be assumed if a beam is continuous across, or is attached to, a substantial supporting member. It is well known that light metal girts and purlins spanning between girder or column sections will develop tension membrane response during blast loading. It is not known with the same certainty whether large steel structural members, such as girders and columns, will develop significant tension membrane response before sustaining the maximum damage level of "100% damage" (a ductility ratio of 15) calculated by the code. This concern is discussed more in Reference 2. It is generally recommended that large steel frame members should be analyzed assuming no tensile membrane response for the present. It is more conservative (maximizes calculated blast damage) to assume no arching or tension membrane response.

Thirdly, the example component property inputs for each component type which follow in this section suggest that **loaded width of primary structural members** (columns and beams) is always equal to the spacing between these components and the **total weight of primary structural members** includes the weight of all secondary, or supported components within the loaded width. These example inputs are estimates rather than well known values. This approach assumes that the dynamic response of the supported components is very quick compared to that of the primary member. This will cause the mass of supported components to deflect with the primary component and it will cause the reaction force from the supported component to be applied to the primary component very quickly. This approach also assumes that the supported components does not yield or "filter" any of the blast load which is applied directly to it. None of these assumptions is completely true for most cases, but the inaccuracies in the input loaded width and component weight tend to offset each other when this approach is used. For the intended use of the FACEDAP program, which is to determine the approximate blast damage to buildings, this approach is recommended because of its simplicity and because it does not err on the side of overconservatism as some simplified design approaches typically do. If a conservative estimate of blast damage to primary structural members is desired, the user should only include 20% of the weight of all secondary, or supported components within the loaded width in the input component weight and keep the loaded width equal to the spacing between members. This is the approach recommended for design of concrete beams supporting a concrete slab in Reference 6.

Finally, a **weighting factor** must be assigned to each input component property set. The weighting factor is used in the summation routine which determines the percentage of building damage in the Analysis module. The percentage of building damage output by the FACEDAP program is a weighted average of the damage to all the building components. It is calculated by summing the "weighted" damage of all building components and dividing by the sum of the component weighting factors. The weighted damage of each component is the product the calculated component damage level (0%, 30%, 60%, or 100% damage) expressed as a fraction and the user defined component weighting factor. This is discussed in more detail in Section 11.1. The weighting factor is used primarily to cause blast damage to major building components to influence the calculated building damage more than an equal level of damage to minor components. Any scheme of assigning positive, non-zero weighting factors to building components which, in the user's judgement, will correctly influence the determination of overall building damage is valid. A scheme which is commonly used is to assign a weighting factor of 1.0 to cladding components (such as wall and roof panels), a factor of 2.0 to stringers, girts and other secondary beams which support cladding components, a factor of 3.0 to primary beams and girders, and a factor of 4.0 to columns. Weighting factors of wall panels which support roof components are typically increased to 2.0 because these components are functioning more as primary components in this case. Since the weighting factor is assigned as a component property, the user must verify that a common weighting factor is valid for all components within the subgroup of components which will be assigned the given set of component properties. Otherwise, two component property sets with different weighting factors must be input.

The weighting factor can also be used to account for the presence of building components which are not explicitly input into the program. This is particularly important when buildings contain hundreds of secondary components, such as roof panels, roof joists, wall panels, etc., and only a portion of these components can be input without exceeding the limitations of the FACEDAP code in Table 9. In this case, "representative" secondary components are input, every third roof panel for example, and the weighting factors of these components are increased to account for the component which are not input. For example, if every third roof panel is input, the weighting factor of the input roof panel should be multiplied by three to account for the fact that only every third roof panel is entered into the program. This will cause the weighted damage of the input roof panel to be equal to the sum of the weighted damage to the three panels it is representing assuming that all three panels sustain the same damage level. This is a good assumption as long as all panels which are represented by the input panel have the same properties and span length as the input panel and all three panels are at nearly the same standoff from the explosive charge. This situation is discussed in more detail in Section 8.1.6.

8.5.2.1 Component Property Input for Reinforced Concrete Beam

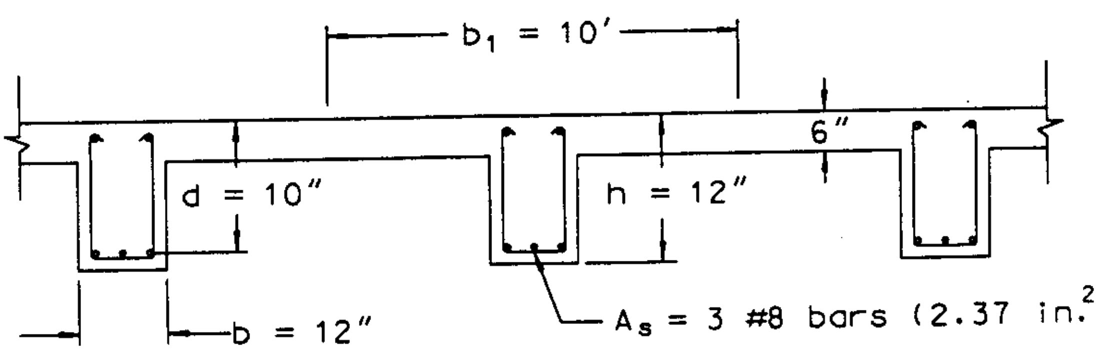
Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Beam Width (b)	in	real	Beam Width	12 in
Beam Thickness (h)	in	real	Beam Thickness	12 in
Loaded Width (b_1)	ft	real	Loaded Width	10 ft
Total Weight (W)	lb	real	Total Weight in Pounds of Section + Supported Components	see equation below figure
Concrete Compressive Strength	psi	real	Compressive Strength of the Concrete (f'c)	4,000 psi
Steel Yield Strength	psi	real	Yield Strength of the Steel Reinforcement	60,000 psi
Depth to Tensile Steel (d)	in	real (default formula)	Depth to Tensile Steel Reinforcement from Loaded Side	10 in
Area of Tensile Steel (A_s)*	in^2	real (default formula)	Area of Tensile Steel Reinforcement within Section Width	2.37 in^2
Moment of Inertia (I_{eff})	in^4	real (default formula)	Moment of Inertia of Cracked Cross Section	1,150 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple

* SEE GENERAL NOTES 1 AND 2 AT END OF COMPONENT DESCRIPTIONS

Default Formulas

$$d = h - 2 \qquad A_s = (b)(d)(0.02)$$

$$I_{eff} = \frac{bd^3(5.5\rho + 0.083)}{2} \qquad \rho = \frac{A_s}{(bd)}$$



$W = [(10\text{ ft})(0.5\text{ ft}) + (12\text{ in})(12\text{ in} - 6\text{ in})/144]\,(L)\,(150\text{ lb/ft}^3)$
where L = span length (ft)

8.5.2.2 Component Property Input for One-Way Reinforced Concrete Slabs

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Section Width (b)	in	real	Section Width (Used for all Section Property Calculations)	12 in
Slab Thickness (h)	in	real	Slab Thickness	6 in
Concrete Compressive Strength	psi	real	28 Day Compressive Strength of the Concrete (f'c)	4,000 psi
Steel Yield Strength	psi	real	Yield Strength of the Steel Reinforcement	60,000 psi
Depth to Tensile Steel (d)	in	real (default formula)	Depth to Tensile Steel Reinforcement	4.5 in
Area of Tensile Steel (A_s)*	in^2	real (default formula)	Area of Tensile Steel Reinforcement within Section Width	0.16 in^2
Concrete Density	lb/ft^3	real	Weight Density of Concrete	150 lb/ft^3
Moment of Inertia (I_{cr})	in^4	real (default formula)	Moment of Inertia of Cracked Cross Section Within Width	54 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Fixed

* SEE GENERAL NOTES 1 AND 2 AT END OF COMPONENT DESCRIPTIONS

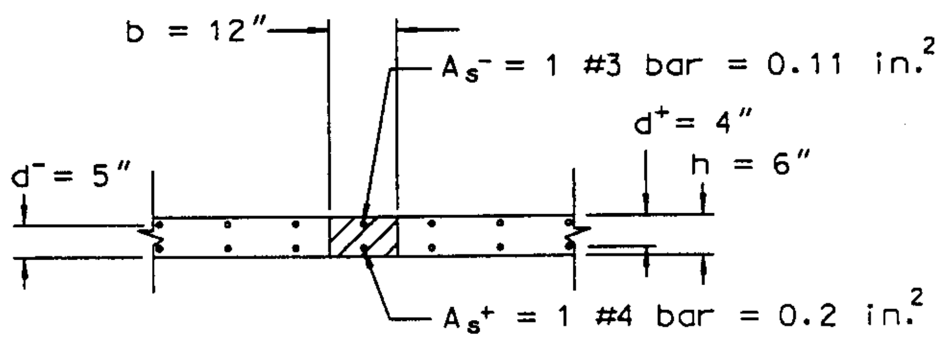
Default Formulas

$d = h - 2$

$A_s = (b)(d)(0.02)$

$$I_{cr} = \frac{bd^3 (5.5\rho + 0.083)}{2}$$

$$\rho = \frac{A_s}{(bd)}$$



8.5.2.3 Component Property Input for Two-Way Reinforced Concrete Slabs

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Section Width (b)	in	real	Section Width (Used for all Section Property Calculations)	12 in
Slab Thickness (h)	in	real	Slab Thickness	6 in
Concrete Compressive Strength	psi	real	28 Day Compressive Strength of the Concrete (f'c)	4,000 psi
Steel Yield Strength	psi	real	Yield Strength of the Steel Reinforcement	60,000 psi
Depth to Tensile Steel (d)†	in	real (default formula)	Depth to Tensile Steel Reinforcement	4 in
Area of Tensile Steel (A_s)*	in^2	real (default formula)	Area of Tensile Steel Reinforcement within Section Width	0.16 in^2
Concrete Density	lb/ft^3	real	Weight Density of Concrete	150 lb/ft^3
Moment of Inertia (I_{eff})	in^4	real (default formula)	Moment of Inertia of Cracked Cross Section Within Width	39 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple
Arching Response	-	option	Arching (Compression Membrane) Response Will/Will Not Occur	No arching

* SEE GENERAL NOTE 3 AT END OF COMPONENT DESCRIPTIONS

† SEE GENERAL NOTE 4 AT END OF COMPONENT DESCRIPTIONS

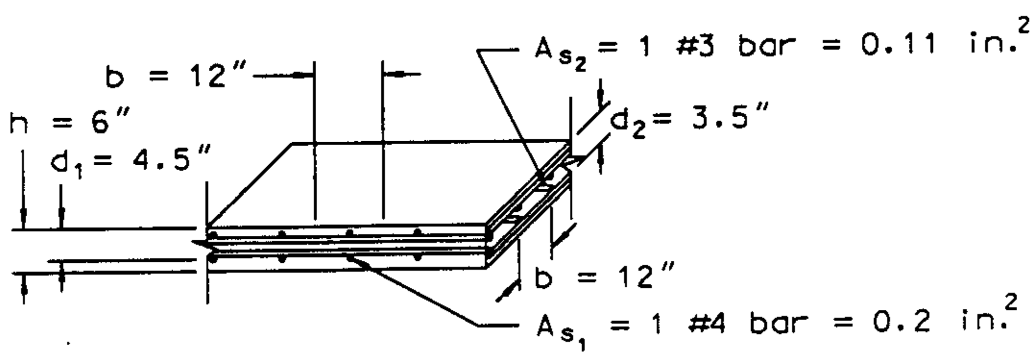
Default Formulas

$$d = h - 2$$

$$A_s = (b)(d)(0.02)$$

$$I_{eff} = \frac{bd^3(5.5\rho + 0.083)}{2}$$

$$\rho = \frac{A_s}{(bd)}$$



8.5.2.4 Component Property Input for Reinforced Concrete Exterior Columns

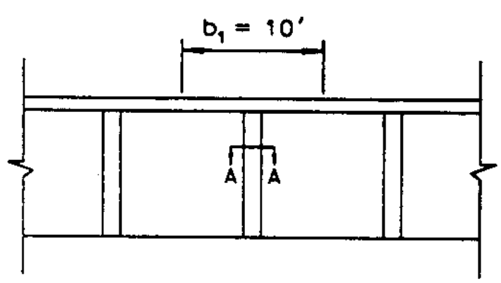
Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Column Width (b)	in	real	Width of Column Cross Section	12 in
Column Thickness (h)	in	real	Thickness of Column Cross Section (Perpendicular to Loaded Area)	14 in
Loaded Width (b_l)	ft	real	Width of Area Supported by Component Which is Loaded by Blast	10 ft
Total Weight (W)	lb	real	Weight of Component Plus Attached Components Within Loaded Width	see equation below figure
Concrete Compressive Strength	psi	real	28 Day Compressive Strength of the Concrete (f'c)	4,000 psi
Steel Yield Strength	psi	real	Yield Strength of the Steel Reinforcement	60,000 psi
Depth to Tensile Steel (d)	in	real (default formula)	Depth to Tensile Steel Reinforcement	12 in
Area of Tensile Steel (A_s)*	in^2	real (default formula)	Area of Tensile Steel Reinforcement in Column	2.37 in^2
Moment of Inertia (I_{cr})	in^4	real (default formula)	Moment of Inertia of Cracked Column Cross Section Resisting Lateral Load	1,799 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Fixed

* SEE GENERAL NOTES 1 AND 2 AT END OF COMPONENT DESCRIPTIONS

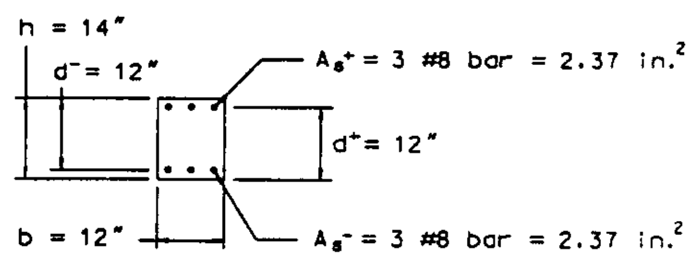
Default Formulas

$d = h - 2$ $\qquad A_s = (b)(d)(0.02)$

$$I_{cr} = \frac{bd^3(5.5\rho + 0.083)}{2} \qquad \rho = \frac{A_s}{(bd)}$$



ELEVATION



A - A

$W = [(10\ ft - 1\ ft)(t) + (12\ in)(14\ in)/144](L)(150\ lb/ft^3)$

where L = column height (ft)

t = (concrete) wall panel thickness (ft)

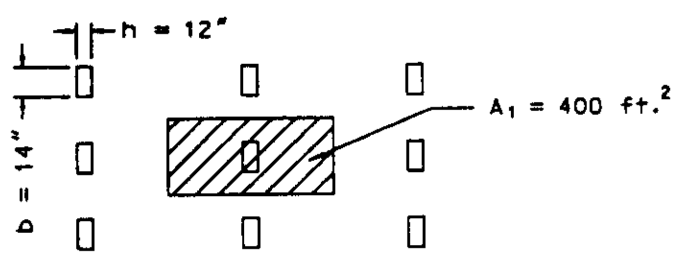
8.5.2.5 Component Property Input for Reinforced Concrete Interior Columns

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Smaller Column Dimension (h)	in	real	Smaller Column Cross Section Dimension	12 in
Larger Column Dimension (b)	in	real	Larger Column Cross Section Dimension	14 in
Column Height	ft	real	Column Height Between Lateral Supports	-
Loaded Area (A_1)	ft^2	real	Loaded Area Supported by Column	400 ft^2
Supported Weight per Area (W)	lb/ft^2	real	Weight Per Unit Area of Supported Area	see equation below figure
Concrete Compressive Strength	psi	real	28 Day Compressive Strength of the Concrete (f'c)	4,000 psi
Minimum Moment of Inertia (I)	in^4	real (default formula)	Moment of Inertia of Cross Section About Weak Bending Axis	2,016 in^4
Boundary Condition	-	option	Possible Boundary Conditions Shown Below	-

Default Formula $I = \frac{bh^3}{12}$

Boundary Conditions

End Support Conditions	Side Sway
Fixed-Simple	No
Fixed-Simple	Yes
Fixed-Fixed	No
Fixed-Fixed	Yes
Simple-Simple	No
Simple-Simple	Yes



PLAN VIEW

W = (t) (150 lb/ft^3) for flat concrete slab roof
t = roof slab thickness (ft)

8.5.2.6 Component Property Input for Reinforced Concrete Frames

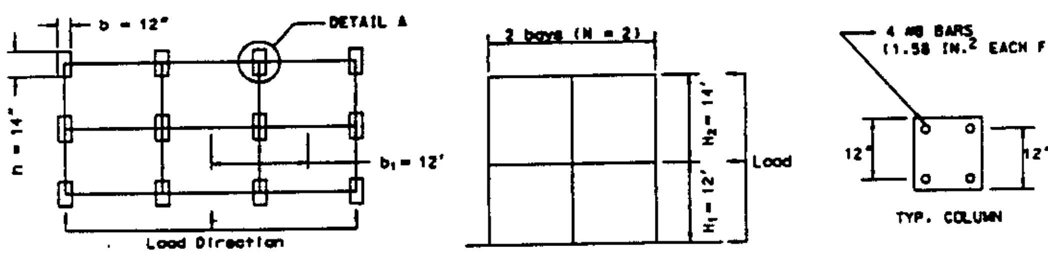
Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Loaded Width (b_l)	ft	real	Width of Wall Area Supported by Exterior Column of Frame	12 ft
Average Column Width (b)	in	real	Average Width Across Cross Section of Frame Columns	12 in
Average Column Thickness (h)	in	real	Average Thickness of Frame Columns (Perpendicular to Loaded Area)	14 in
Total Weight (W)	lb	real	Effective Weight Supported by Frame	see equation below figure
Number of Bays (N)	-	integer	Number of Bays in the Frame (Must be Less Than 15)	2
Single Story Height (H)	ft	real	Average Story Height in Frame	13 ft
Number of Stories	-	option	Number of Stories in Frame (2 Story Maximum)	2
Concrete Compressive Strength	psi	real	28 Day Compressive Strength of Concrete in Frame Columns (f'c)	4,000 psi
Steel Yield Strength	psi	real	Yield Strength of Steel Reinforcement in Frame Columns	60,000 psi
Column Depth to Tensile Steel (d)	in	real (default formula)	Average Depth to Tensile Steel Reinforcement in Frame Columns	12 in
Column Area of Tensile Steel (A_s)*	in^2	real (default formula)	Average Area of Tensile Steel in Frame Columns	1.58 in^2
Column Moment of Inertia (I_{eff})	in^4	real (default formula)	Average Moment of Inertia (Cracked Section) of Frame Columns	1,486 in^4

* SEE GENERAL NOTES 1 AND 2 AT END OF COMPONENT DESCRIPTIONS

Default Formulas

$$d = h - 2 \qquad A_s = (b)\,(d)\,(0.02)$$

$$I_{eff} = \frac{bd^3\,(5.5\rho + 0.083)}{2} \qquad \rho = \frac{A_s}{(bd)}$$



PLAN — ELEVATION — DETAIL A

W = roof weight + 1/3 (wall and column weight) within Loaded Width

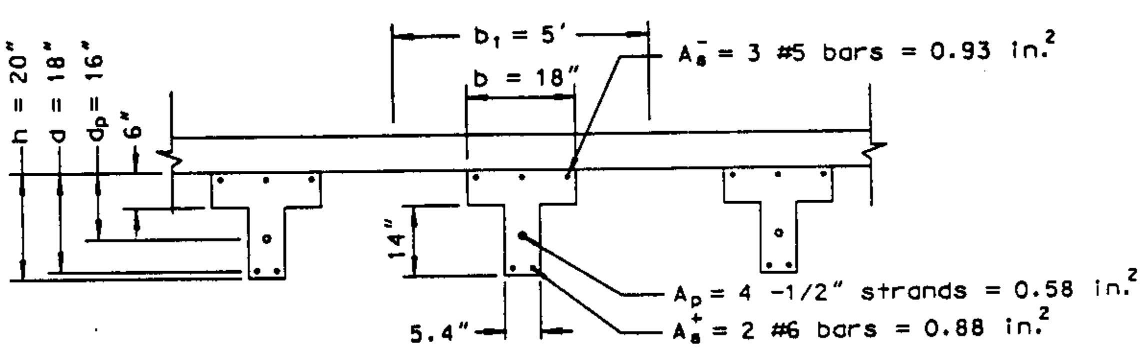
$W = (t_r)\,(12\text{ ft})\,(24\text{ ft})\,(150\text{ lb/ft}^3) + 1/3\,[(2\,(t_w)\,(12\text{ ft})\,(26\text{ ft})] + [3\,(12\text{ in})\,(14\text{ in})\,(26\text{ ft})/144]]\,(150\text{ lb/ft}^3)$

t_r = (concrete) roof slab thickness (ft)

t_w = (concrete) wall slab thickness (ft)

8.5.2.7 Component Property Input for Reinforced Concrete Prestressed Beams

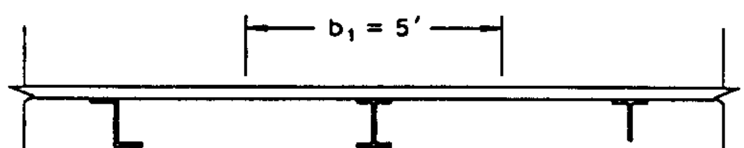
Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Beam Flange Width (b)	in	real	Beam Flange in Compression	18 in
Loaded Width (b_l)	ft	real	Width of Area Supported by Component Which is Loaded by Blast	5 ft
Total Weight (W)	lb	real	Total Weight of Component Plus Weight of Any Supported Components	see equation below figure
Concrete Compressive Strength	psi	real	28 Day Compressive Strength of the Concrete (f'c)	4,000 psi
Steel Yield Strength	psi	real	Yield Strength of the Steel Reinforcement	60,000 psi
Prestress Steel Ultimate Strength	psi	real	Ultimate Strength of the Prestressing Steel	250,000 psi
Depth to Tensile Steel (d)	in	real	Depth to Tensile Reinforcement (If Unknown: Beam Thickness - 2")	16 in
Depth to Prestress Steel (d_p)	in	real	Depth to Prestress Steel in Maximum Moment Area (If Unknown: 80% Beam Thickness)	11 in
Area of Tensile Steel (A_s^+)	in^2	real	Area of Non-Prestressed Tensile Steel (If Unknown: 0.5% Beam Area)	0.88 in^2
Area of Compression Steel (A_s^-)	in^2	real	Area of Non-Prestressed Compression Steel (If Unknown: 0.5% of Beam Area)	0.93 in^2
Area of Prestress Steel (A_p)	in^2	real	Area of Non-Prestressed Compression Steel (If Unknown: 0.5% Beam Area)	0.58 in^2
Moment of Inertia	in^4	real	Uncracked Moment of Inertia of Cross Section	6,005 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple



W = [(5 ft) (t) + A_b] (L) (150 lb/ft^3)
L = span length (ft)
A_b = cross sectional area of beam (ft^2)
t = (concrete) slab thickness (ft)

8.5.2.8 Component Property Input for Steel Beams

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Loaded Width (b_1)	ft	real	Width of Area Supported by Component Which is Loaded by Blast	5 ft
Total Weight (W)	lb	real	Total Weight of Component Plus Weight of Any Supported Components	see equation below figure
Steel Yield Strength	psi	real	Yield Strength of Beam	36,000 psi
Plastic Section Modulus (Z)	in^3	real	Plastic Section Modulus	38.8 in^3
Moment of Inertia (I)	in^4	real	Moment of Inertia of Cross Section	170 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple
Tension Membrane Response (Y or N)	-	option	Designate if Tension Membrane Response Will (Yes) or Will Not (No) Occur	No



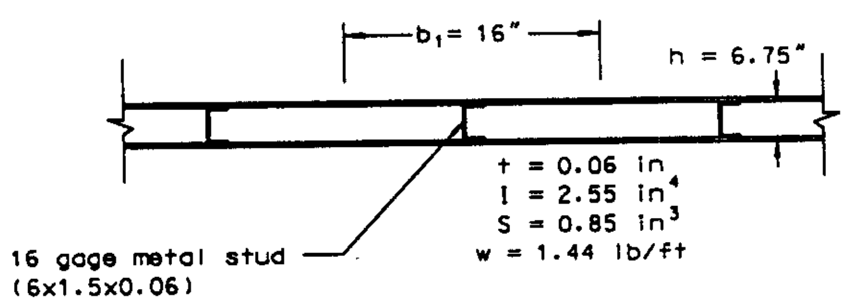
$W = [(W_p)(5\ ft) + (33\ lb/ft)](L)$
W_p = areal weight of paneling and insulation supported by beams (lb/ft^2)
L = span length (ft)

8.5.2.9 Component Property Input for Metal Stud Walls

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Section Width (b_1)	in	real	Section Width (Used for all Section Property Calculations)	16 in
Wall Thickness (h)	in	real	Total Wall Thickness	6.6 in
Total Weight (W)	lb	real	Total Weight of Component Plus Attached Components Within Loaded Width	see equation below figure
Yield Strength	psi	real	Yield Strength of Metal Stud	50,000 psi
Elastic Section Modulus (S)	in^3	real (default formula)	Elastic Section Modulus of Metal Stud	0.85 in^3
Moment of Inertia (I)	in^4	real (default formula)	Moment of Inertia of Stud	2.55 in^4

Default Formulas $S = 0.08\,(h + h^2/10)$ $I = 0.036\,(h^2 + h^3/10)$

Note: These default formulas assume a metal stud thickness equal to 0.06", a section depth equal to (0.9) (h), an unstiffened flange width equal to 1.5", and a "C" or "Z" shaped section.



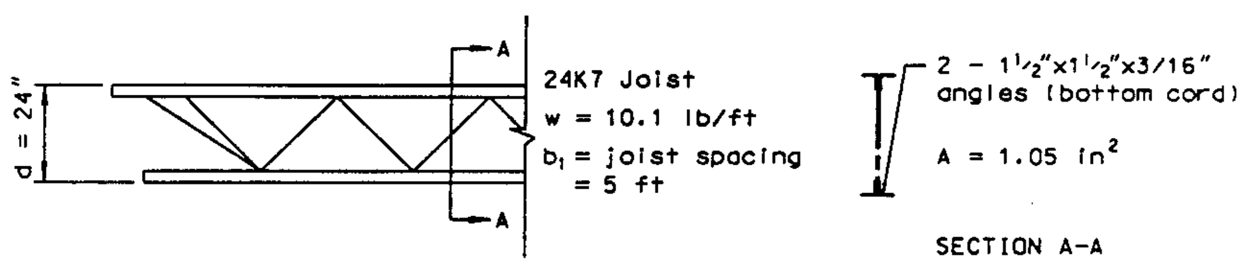
W = $[W_p\,(1.33\text{ ft}) + (1.44\text{ lb/ft})]\,(L)$
W_p = areal weight of paneling and insulation supported by studs (lb/ft^2)
L = span length (ft)

8.5.2.10 Component Property Input for Steel Open Web Joists

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Loaded Width (b_l)	ft	real	Width of Area Supported by Component	5 ft
Joist Depth (d)	in	real	Distance Between Top and Bottom Chord of Joist	24 in
Area of Bottom Chord (A)†	in^2	real	Cross Sectional Area of Joist Bottom Chord at Midspan	1.05 in^2
Total Weight (W)	lb	real	Total Weight of the Joist and Deck Within the Loaded Width	see equation below figure
Yield Strength	psi	real	Yield Strength of Bottom Chord	50,000 psi
Bending Stiffness *	lb/in	real	Joist Bending Stiffness	1.06E4 lb/in for 30 ft span

* SEE GENERAL NOTE 6 AT END OF COMPONENT DESCRIPTIONS

† SEE GENERAL NOTE 5 AT END OF COMPONENT DESCRIPTIONS



W = [(t) (5 ft) (150 lb/ft³) + (10.1 lb/ft)] (L)

t = (concrete) roof slab thickness

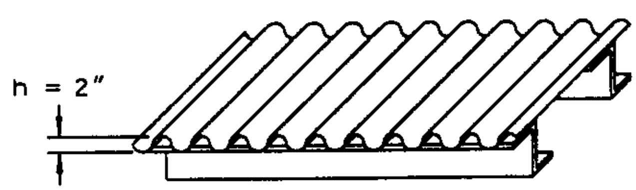
L = span length (ft)

8.5.2.11 Component Property Input for Steel Corrugated Decking

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Decking Uncoated Thickness (t)	in	real	Uncoated Thickness of Decking	0.023 in
Rib Height (h)	in	real	Height of Corrugations	2 in
Total Weight (W)	lb	real	Weight of Decking and Any Attached Material Within 1 ft Width	see equation below figure
Yield Strength	psi	real	Yield Strength of Decking	40,000 psi
Elastic Section Modulus (S)	in^3/ft	real (default formula)	Elastic Section Modulus of Decking Per Foot	0.29 in^3/ft
Moment of Inertia (I)	in^4/ft	real (default formula)	Moment of Inertia of Decking Cross Section Per Foot	0.343 in^4/ft
Boundary Condition	-	option	Fixed or Simple Supports	Fixed

Default Formulas $S = 4.5\ ht$ $I = 2.4\ th^2$

Note: These default formulas assume an effective flange width 0.375 times the section width. This is a good approximation based on comparisons with actual section properties reported in manufacturer's literature. A 12" section width is assumed.

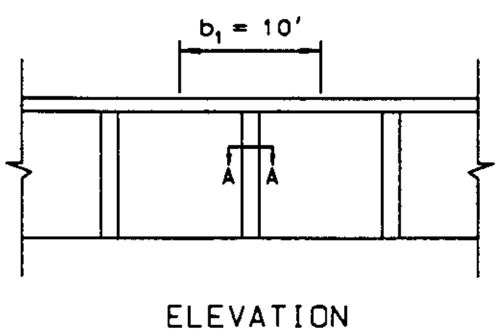


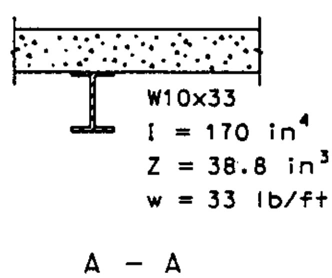
Vulcraft 2VL22 Deck
t = 0.023 in (22 gage)
I = 0.343 in^4/ft
S = 0.29 in^3/ft
w = 1.7 lb/ft^2

W = $[(1.7\ lb/ft^2) + (W^1)]\ (1\ ft)\ (L)$
W^1 = areal weight of insulation, etc., attached to decking (lb/ft^2)
L = span length (ft)

8.5.2.12 Component Property Input for Steel Exterior Columns

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Loaded Width (b_1)	ft	real	Width of Area Supported by Component Which is Loaded by Blast	10 ft
Total Weight (W)	lb	real	Total Weight of Component Plus Weight of Any Supported Components	see equation below figure
Yield Strength	psi	real	Yield Strength of Column	36,000 psi
Plastic Section Modulus (Z)	in^3	real	Plastic Section Modulus of Column	38.8 in^3
Moment of Inertia (I)	in^4	real	Moment of Inertia of Column Cross Section Resisting Lateral Load	170 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple
Tension Membrane Response (Y or N)	-	option	Designate if Tension Membrane Response Will (Yes) or Will Not (No) Occur	No





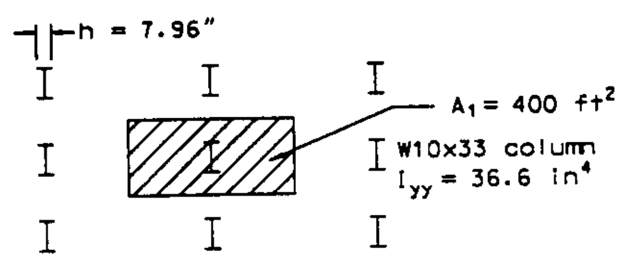
$$W = [(10\ ft)(t)(150\ lb/ft^3) + 33\ lb/ft](L)$$
$$t = \text{(concrete) wall slab thickness (ft)}$$
$$L = \text{span length (ft)}$$

8.5.2.13 Component Property Input for Steel Interior Columns

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Smaller Column Dimension (h)	in	real	Column Thickness About Weak Axis	7.96 in
Column Height	ft	real	Column Height Between Lateral Supports	-
Loaded Area (A_1)	ft^2	real	Loaded Area Supported by Column	400 ft^2
Supported Weight per Area (W)	lb/ft^2	real	Weight Per Unit Area of Supported Area	see equation below figure
Yield Strength	psi	real	Yield Strength of Column	36,000 psi
Minimum Moment of Inertia (I_{yy})	in^4	real	Column Moment of Inertia About the Weak Bending Axis	36.6 in^4
Boundary Condition	-	option	Possible Boundary Conditions Shown Below	-

Boundary Conditions

End Support Conditions	Side Sway
Fixed-Simple	No
Fixed-Simple	Yes
Fixed-Fixed	No
Fixed-Fixed	Yes
Simple-Simple	No
Simple-Simple	Yes

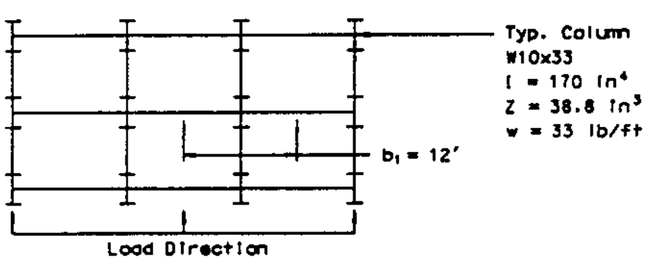


PLAN VIEW

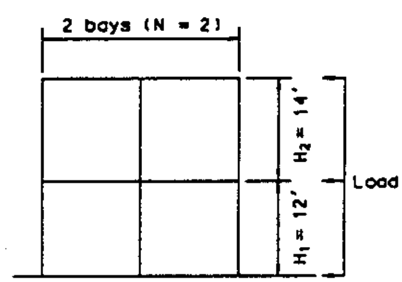
W = (t) (150 lb/ft^3)
t = (concrete) roof slab thickness

8.5.2.14 Component Property Input for Steel Frames

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Loaded Width (b_1)	ft	real	Width of Wall Area Supported by Exterior Column of Frame	12 ft
Total Weight (W)	lb	real	Effective Weight Supported by Frame	see equation below figure
Number of Bays (N)	-	integer	Number of Bays in the Frame (Must be Less Than 15)	2
Single Story Height (H)	ft	real	Average Story Height	13 ft
Number of Stories	-	option	Select Number of Stories in Frame (2 Story Maximum)	2
Steel Yield Strength	psi	real	Yield Strength of Frame Columns	36,000 psi
Column Plastic Section Modulus (Z)	in^3	real	Average Plastic Section Modulus of Frame Columns	38.8 in^3
Column Moment of Inertia (I)	in^4	real	Average Moment of Inertia of Frame Columns Resisting Lateral Load	170 in^4



PLAN



ELEVATION

W = roof weight + 1/3 (wall and column weight) within Loaded Width

$W = (t_r)\,(12\text{ ft})\,(24\text{ ft})\,(150\text{ lb/ft}^3) + 1/3\,[[2\,(t_w)\,(12\text{ ft})\,(26\text{ ft})\,(150\text{ lb/ft}^3)] + [3\,(33\text{ lb/ft})\,(26\text{ ft})]]$

where t_r = (concrete) roof slab thickness (ft)

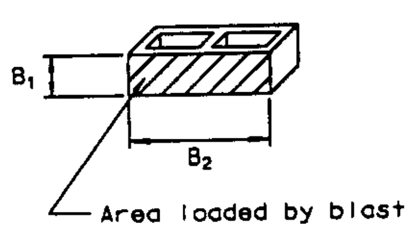
t_w = (concrete) wall thickness (ft)

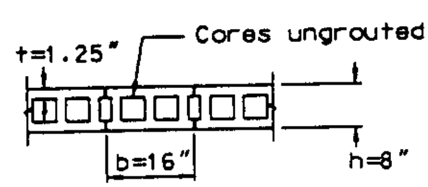
8.5.2.15 Component Property Input for Masonry One-Way Unreinforced Walls

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Section Width (b)	in	real	Section Width (Used for all Section Property Calculations)	16 in
Wall Thickness (h)	in	real	Wall Thickness	8 in
Masonry Compressive Strength	psi	real	Compressive Strength of Masonry Wall	1,350 psi
Masonry Tensile Strength	psi	real	Tensile Strength of Masonry Wall	200 psi
Weight/Unit Loaded Area (W)	lb/ft^2	real	Weight per Unit of Surface Area Loaded by Blast Pressure - Accounting for Voids	see equation below figure
Masonry Shell Thickness (t)	in	real	Masonry Shell Thickness (Usually 1.25" for CMU, 0.75" for Thickness)	1.25 in
Section Modulus (S)	in^3	real	Elastic Section Modulus Within Section Width	135 in^3
Moment of Inertia (I)	in^4	real	Moment of Inertia of Cross Section Within Section Width	455 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple
Arching Response	-	option	Designate if Arching (Compression Membrane) Response Will/Will Not Occur	No arching

Default Formulas $S = t\,b\,(h-t)$ $\qquad I = \dfrac{tb\,(h-t)^2}{2}$

Note: These formulas assume the wall is ungrouted. If it is known that the wall is grouted input $S = \frac{th^2}{6}$, $I = \frac{th^3}{12}$





CROSS-SECTION

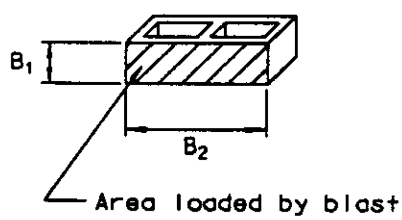
$W = W_B/[(B_1)(B_2)]$

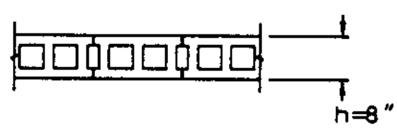
W_B = weight of block (including grout if voids grouted) (lb)

B_1, B_2 = dimensions of block area loaded by blast (ft)

8.5.2.16 Component Property Input for Masonry Two-Way Unreinforced Walls

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name		char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor		real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Wall Thickness (h)	in	real	Wall Thickness	8 in
Masonry Compressive Strength	psi	real	Compressive Strength of Masonry Wall	1,350 psi
Masonry Tensile Strength	psi	real	Tensile Strength of Masonry Wall	200 psi
Weight/Unit Loaded Area (W)	lb/ft^2	real	Weight per Unit of Surface Area Loaded by Blast Pressure - Accounting for Voids	see equation below figure
Boundary Condition	-	option	Fixed or Simple Supports	Simple





CROSS-SECTION

$W = W_B/[(B_1)(B_2)]$

W_B = weight of block (including grout if voids grouted) (lb)

B_1, B_2 = dimensions of block area loaded by blast (ft)

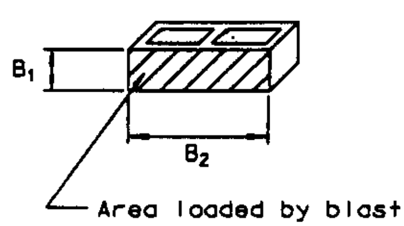
8.5.2.17 Component Property Input for Masonry One-Way Reinforced Walls

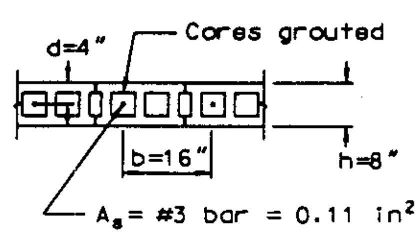
Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Section Width (b)	in	real	Section Width (Used for all Section Property Calculations)	16 in
Wall Thickness (h)	in	real	Wall Thickness	8 in
Masonry Compressive Strength	psi	real	Compressive Strength of Masonry Wall	1,350 psi
Steel Yield Strength	psi	real	Yield Strength of the Steel Reinforcement	60,000 psi
Depth to Tensile Steel (d)	in	real (default formula)	Depth to Tensile Steel Reinforcement	4 in
Area of Tensile Steel (A_s)*	in^2	real (default formula)	Area of Tensile Steel Reinforcement Within Section Width	0.11 in^2
Weight/Unit Loaded Area (W)	lb/ft^2	real	Weight per Unit of Surface Area Loaded by Blast Pressure	see equation below figure
Moment of Inertia (I_{cr})	in^4	real (default formula)	Moment of Inertia of Cracked Cross Section Within Section Width	.344 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple

* SEE GENERAL NOTES 1 AND 2 AT END OF COMPONENT DESCRIPTIONS

Default Formulas $d = (0.5)(h)$ $A_s = (b)(d)(0.002)$

$$I_{cr} = \frac{bh^3}{24} + 0.0025\,(b)(d)^3$$





CROSS-SECTION

$W = W_B/[(B_1)(B_2)]$

W_B = weight of block (including grout if voids grouted) (lb)

B_1, B_2 = dimensions of block area loaded by blast (ft)

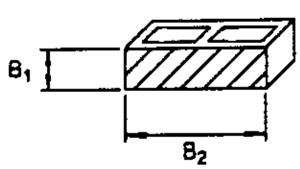
8.5.2.18 Component Property Input for Masonry Two-Way Reinforced Walls

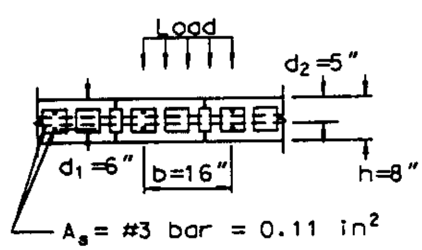
Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Section Width (b)	in	real	Section Width (Used for all Section Property Calculations)	16 in
Wall Thickness (h)	in	real	Wall Thickness	8 in
Masonry Compressive Strength	psi	real	Compressive Strength of Masonry Wall	1,350 psi
Steel Yield Strength	psi	real	Yield Strength of the Steel Reinforcement	60,000 psi
Depth to Tensile Steel (d)†	in	real (default formula)	Depth to Tensile Steel Reinforcement	5.5 in
Area of Tensile Steel (A_s)*	in^2	real (default formula)	Area of Tensile Steel Reinforcement Within Section Width	0.11 in^2
Weight/Unit Loaded Area (W)	lb/ft^2	real	Weight per Unit of Surface Area Loaded by Blast Pressure	see equation below figure
Moment of Inertia (I_{eff})	in^4	real (default formula)	Moment of Inertia of Cracked Cross Section Within Section Width	348 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple

* SEE GENERAL NOTE 3 AT END OF COMPONENT DESCRIPTIONS
† SEE GENERAL NOTE 4 AT END OF COMPONENT DESCRIPTIONS

Default Formulas $d = (0.5)(h)$ $A_s = (b)(d)(0.002)$

$$I_{eff} = \frac{bh^3}{24} + 0.0025\,(b)(d)^3$$





CROSS-SECTION

W = $W_B/[(B_1)(B_2)]$
W_B = weight of block (including grout if voids grouted) (lb)
B_1, B_2 = dimensions of block area loaded by blast (ft)

8.5.2.19 Component Property Input for Masonry Pilasters

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Pilaster Width (b)	in	real	Width of Pilaster Cross Section	16 in
Pilaster Thickness (h)	in	real	Thickness of Pilaster Cross Section (Perpendicular to Loaded Area)	16 in
Loaded Width (b_l)	ft	real	Width of Area Supported by Pilaster Which is Loaded by Blast	12 ft
Total Weight (W)	lb	real	Total Weight of Pilaster Plus Attached Components Within Loaded Width	see equation below figure
Masonry Compressive Strength	psi	real	Compressive Strength of Pilaster	1,350 psi
Steel Yield Strength	psi	real	Yield Strength of the Steel Reinforcement	60,000 psi
Depth to Tensile Steel (d)	in	real (default formula)	Depth to Tensile Steel Reinforcement	12 in
Area of Tensile Steel (A_s)	in^2	real (default formula)	Area of Tensile Steel Reinforcement	1.32 in^2
Moment of Inertia (I_{eff})	in^4	real (default formula)	Moment of Inertia of Cracked Pilaster Cross Section Resisting Lateral Load	1,670 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple

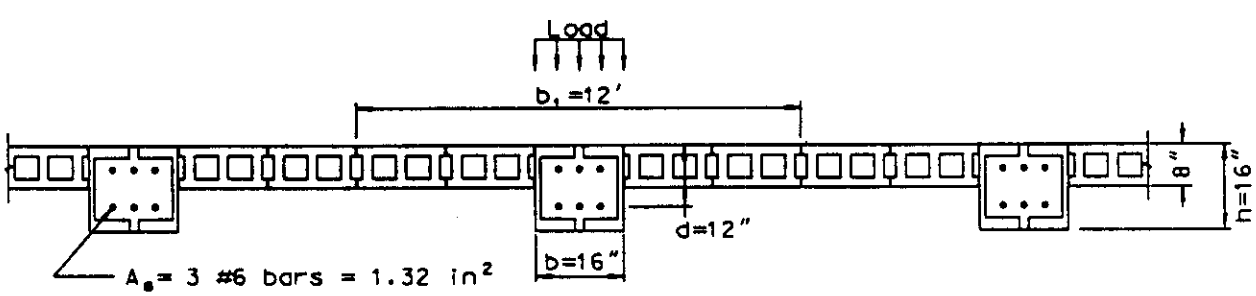
Default Formulas

$$d = (0.8)(h)$$

$$A_s = (b)(d)(0.01)$$

$$I_{eff} = \frac{bd^3(5.5\rho + 0.083)}{2}$$

$$\rho = \frac{A_s}{(bd)}$$



SECTION

W = [W¹ (12 ft) + [(16 in - 8 in) (16 in)/144] (120 lb/ft³)] (L)

W¹ = areal weight of wall per unit area (lb/ft²) - calculate as shown for other masonry components

L = pilaster height (ft)

8.5.2.20 Component Property Input for Wood Walls

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Stud Width (b)	in	real	Actual Wall Stud Width (Usually Nominal Width - 0.5")	1.5 in
Stud Depth (h)	in	real	Actual Wall Stud Depth (Usually Nominal Depth - 0.5")	3.5 in
Loaded Width (b_l)	in	real	Loaded Width (Stud Spacing) Used for all Section Properties	16 in
Total Weight (W)	lb	real	Total Weight of Stud Plus Attached Components Within Loaded Width	see equation below figure
Wood Yield Strength	psi	real	Full Yield Strength of Wall Stud (Approximately 2.5 Times Allowable Design Stress)	-
Modulus of Elasticity	psi	real	Modulus of Elasticity of Stud	1.2E6 psi
Wall Sheathing Thickness (t)	in	real	Average Thickness of Interior and Exterior Wall Sheathing Attached to Stud	0.5 in
Stud + Sheath Moment of Inertia (I)	in^4	real (default formula)	Moment of Inertia of Stud (& Sheathing Each Side if Composite)	~~69~~ in^4 37.4
Boundary Condition	-	option	Fixed or Simple Supports	Simple

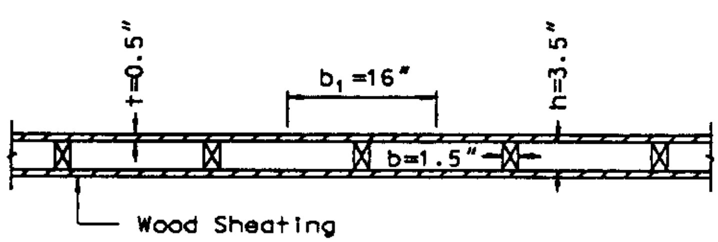
errata

Default Formulas

$$I = \frac{(b_l t/2)(h+t)^2}{2} + \frac{bh^3}{12}$$

Notes: ***The formula above assumes one-~~fourth~~ half of the sheathing span between studs acts compositively with studs***

If sheathing is not composite with wall studs input $I = \frac{bh^3}{12}$ ***and*** $t = 0$



SECTION

$W = [W^1 (16\ in/12) + [(1.5\ in)(3.5\ in)/144](\gamma_w)]L$

W^1 = areal weight of both faces of sheathing and wall insulation (lb/ft^2)

L = wall height (ft)

γ_w = density of wood in stud (lb/ft^3)

8.5.2.21 Component Property Input for Wood Roofs

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Joist Width (b)	in	real	Actual Roof Joist Width (Usually Nominal Width - 0.5")	1.5 in
Joist Depth (h)	in	real	Actual Roof Joist Depth (Usually Nominal Depth - 0.5")	5.5 in
Loaded Width (b_1)	in	real	Loaded Width (Joist Spacing) Used for All Section Properties	24 in
Total Weight (W)	lb	real	Total Weight of Joist + Weight of Attached Components Within Loaded Width	see equation below figure
Wood Yield Strength	psi	real	Full Yield Strength of Joist (Approximately 2.5 Times Allowable Stress)	-
Modulus of Elasticity	psi	real	Modulus of Elasticity of Joist	1.2E6 psi
Roof Decking Thickness (t)	in	real	Wood Thickness of the Roof Decking	0.5 in
Joist + Deck Moment of Inertia (I)	in^4	real (default formula)	Moment of Inertia of Joist (and Deck if Composite)	~~64.8~~ in^4 45
Boundary Condition	...	option	Fixed or Simple Supports	Simple

errata

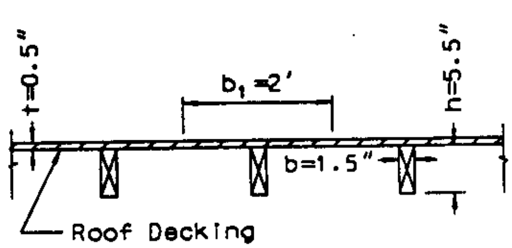
Default Formulas

$$I = (b_1 t/2)\left(\bar{y} - \frac{t}{2}\right)^2 + \frac{h^3 b}{12} + hb\left(t + \frac{h}{2} - \bar{y}\right)^2$$

where
$$\bar{y} = \left[b_1 \frac{t^2}{4} + bh\left(t + \frac{h}{2}\right)\right] / [b_1 t/2 + bh]$$

Notes: ***The formula above assumes ~~one-fourth~~ half of the decking span between studs acts compositively with studs***

If decking is not composite with roof joists, input $I = \frac{bh^3}{12}$ ***and*** $t = 0$



SECTION

W = [[W^1 (2 ft) + [(5.5 in) (1.5 in)/144](γ_w)]](L)

W^1 = areal weight of roof decking and roofing material (lb/ft²)

L = roof joist span (ft)

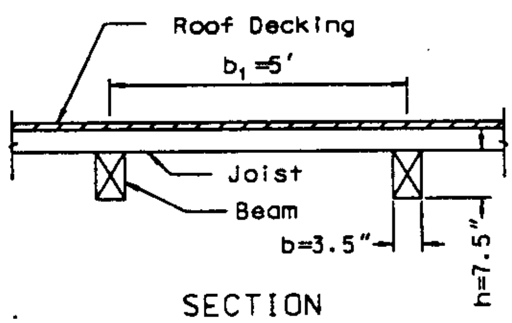
γ_w = density of wood in stud (lb/ft³)

8.5.2.22 Component Property Input for Wood Beams

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Beam Width (b)	in	real	Actual Beam Width (Usually Nominal Width - 0.5")	3.5 in
Beam Depth (h)	in	real	Actual Beam Depth (Usually Nominal Depth - 0.5")	7.5 in
Loaded Width (b_1)	ft	real	Width of Area Loaded by Blast (Beam Spacing)	5 ft
Total Weight (W)	lb	real	Total Weight of Beam + Weight of Supported Components Within Loaded Width	see equation below figure
Wood Yield Strength	psi	real	Full Yield Stress of Beam (Approximately 2.5 Times Allowable Design Stress)	-
Modulus of Elasticity	psi	real	Modulus of Elasticity of Beam	1.2E6 psi
Moment of Inertia (I)	in^4	real (default formula)	Moment of Inertia of Cross Section	123 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple

Default Formulas

$$I = \frac{bh^3}{12}$$



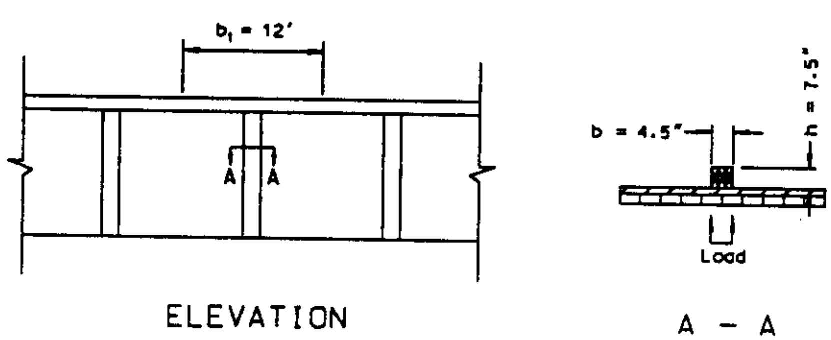
W = $[W^1 (5\ ft) + [(7.5\ in)(3.5\ in)/144](\gamma_w)]/(L)$
W^1 = areal weight of roof decking, roofing material, and roof joists (lb/ft^2)
L = span length of beam (ft)
γ_w = density of wood in beam (lb/ft^3)

8.5.2.23 Component Property Input for Wood Exterior Columns

Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name	-	char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor	-	real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Column Width (b)	in	real	Actual Column Width (Usually Nominal Width - 0.5")	4.5 in
Column Thickness (h)	in	real	Actual Column Depth (Usually Nominal Depth - 0.5")	7.5 in
Loaded Width (b_l)	ft	real	Width of Area Loaded by Blast (Column Spacing)	12 ft
Total Weight (W)	lb	real	Total Weight of Column Plus Attached Components Within Loaded Width	see equation below figure
Wood Yield Strength	psi	real	Full Yield Strength of Column (Approximately 2.5 Times Allowable Design Stress)	-
Modulus of Elasticity	psi	real	Modulus of Elasticity of Column	1.2E6 psi
Moment of Inertia (I)	in^4	real (default formula)	Moment of Inertia of Column Cross Section Resisting Lateral Load	158 in^4
Boundary Condition	-	option	Fixed or Simple Supports	Simple

Default Formulas

$$I = \frac{bh^3}{12}$$



W = [W' (12 ft) + [(4.5 in) (7.5 in)/144](γ_w)](L)

W' = areal weight of wall, stringers, insulation which are laterally supported by column (lb/ft²)

L = column height (ft)

γ_w = density of wood in column (lb/ft³)

8.5.2.24 Component Property Input for Wood Interior Columns

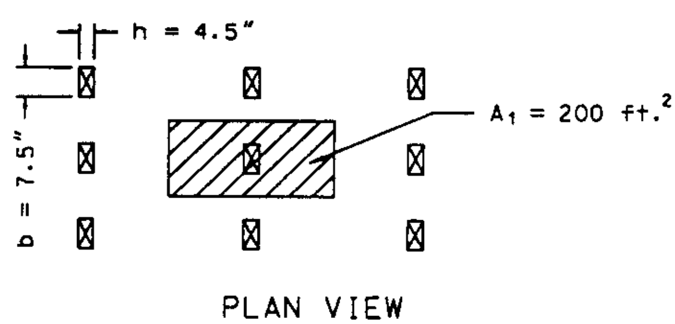
Input Parameter	Units	Parameter Type	Description	Input Value for Example Case Below
Component Property Name		char.	Material & Geometry Properties of Component in Current Row	-
Weighting Factor		real	Component Weighting Factor (> 0) for Overall Building Damage Calculations	-
Smaller Column Dimension (h)	in	real	Smaller Column Cross Section Dimension	4.5 in
Larger Column Dimension (b)	in	real	Larger Column Cross Section Dimension	7.5 in
Column Height	ft	real	Column Height Between Lateral Supports	-
Loaded Area (A_1)	ft^2	real	Loaded Area Supported by Column	200 ft^2
Supported Weight Per Area (W)	lb/ft^2	real	Weight Per Unit Area of Supported Area	-
Wood Yield Strength	psi	real	Full Column Yield Strength (Approximately 2.5 Times Allowable Stress)	-
Modulus of Elasticity	psi	real	Modulus of Elasticity of Column	1.2E6 psi
Minimum Moment of Inertia (I)	in^4	real (default formula)	Moment of Inertia of Cross Section About Weak Bending Axis	57 in^4
Boundary Condition		option	Possible Boundary Conditions Shown Below	-

Default Formula

$$I = \frac{bh^3}{12}$$

Boundary Conditions

End Support Conditions	Side Sway
Fixed-Simple	No
Fixed-Simple	Yes
Fixed-Fixed	No
Fixed-Fixed	Yes
Simple-Simple	No
Simple-Simple	Yes



8.5.2.25 General Notes

1. For simply supported components calculate tensile steel area, A_s, using the midspan reinforcement at the face not loaded by the blast.

2. For components with fixed boundary conditions the tensile steel area, A_s, is calculated by averaging the midspan reinforcement at the inside face and the support reinforcement at the face loaded by the blast. The depth of tensile steel is also calculated as an average of the depth to each steel area.

3. Calculate tensile steel area (A_s) as an average of the tensile steel area in the long and short span directions. When calculating A_s for the long and short spans, reference notes 1) and 2).

4. Use an average depth to tensile steel (d) for long and short span tensile reinforcement.

5. Open web steel joist designation is typically in the form DSW where D is the joist depth in inches, S is the series (for example, K series, DLH series), and W is the chord size designation. An example is a 24K7 joist which has a depth of 24 inches, is a K series joist and has chord members which are classified as chord size 7. The next page shows the size of angles and rounds that are typically used for each chord size classification. The bottom chord area required for input into the FACEDAP program is the area of two of the angles or rounds shown on the next page.

6. The value of K can be estimated as:

   $K = 360\ w_{360}$

   where w_{360} is the uniformly distributed load estimated to cause a midspan deflection of L/360 for the given joist and span where L is the span length. This value is specified by the manufacturer or in Steel Joist Institute literature[8]. Typically, w_{360} is in the units of lb/ft and must be converted to the required units of lb/in by dividing by 12.

Approximate Chord Sizes of Open Web Joists (from Reference 8)

CHORD SIZE	TOP CHORD	BOTTOM CHORD ANGLES	BOTTOM CHORD ROUNDS
1	3/4 x 3/4 x 1/8 or 1 x 1 x 1/8	3/4 x 3/4 x 1/8 or 1 x 1 x 1/8	3/8 Φ
2	1 x 1 x 1/8	3/4 x 3/4 x 1/8 or 1 x 1 x 1/8	1/2 Φ
3	1 1/4 x 1 1/4 x 1/8	1 x 1 x 1/8	9/16 Φ
4	1 1/2 x 1 1/2 x 1/8	1 1/4 x 1 1/4 x 1/8	5/8 Φ
5	1 1/2 x 1 1/2 x 5/32 or 1 3/4 x 1 3/4 x 1/8	1 1/2 x 1 1/2 x 1/8	11/16 Φ
6	1 1/2 x 1 1/2 x 3/16	1 1/4 x 1 1/4 x 3/16	3/4 Φ
7	1 3/4 x 1 3/4 x 3/16	1 1/2 x 1 1/2 x 3/16	13/16 Φ
8	2 x 2 x 3/16 or 2 x 1 1/2 x 3/16 (LLV)	1 3/4 x 1 3/4 x 3/16 or 1 1/2 x 1 1/2 x 3/16	7/8 Φ
9	2 x 2 x 3/16	1 3/4 x 1 3/4 x 3/16	15/16 Φ
10	2 x 1 1/2 x 1/4 (LLV) or 2 1/2 x 2 x 3/16 (LLV)	2 x 2 x 3/16	1" Φ
11	2 x 2 x 1/4	1 3/4 x 1 3/4 x 1/4 or 2 1/2 x 2 x 3/16 (LLH)	------

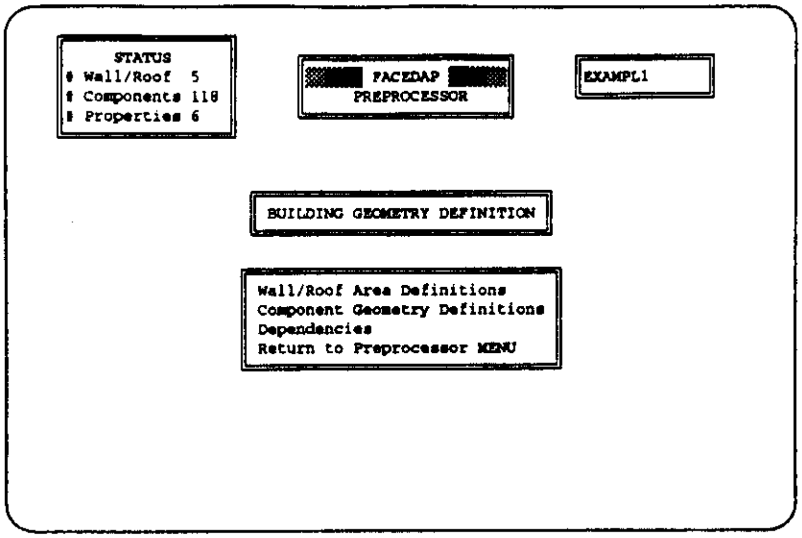
8.5.2.26 Situations Where Required Component Property Parameters are Unavailable

As the previous pages show, a significant amount of input component property data can be required. The user should make a reasonable effort to determine all required component property input values from building drawings or, if possible, from an inspection of the building of interest. However, sometimes some required component property input will not be available. If default values are provided or the required input value can be calculated with a default formula, these options should be used in such cases. If the user has experience with other buildings, where component property information was available, and the user has reason to believe that the buildings were built during the same era for similar purposes, then these component property values can be assumed. Normally, approximate input component property values for a small percentage of building components will not significantly affect the accuracy of the calculated building damage. Possible exceptions include cases where a high weighting factor is assigned to these components or these components sustain 100% damage and, because of the consideration of component dependencies in the *Analysis* module, cause cascading failure of other components. Component dependencies are explained in Section 8.6.4. Therefore, a high weighting factor should be assigned with caution to components if their properties are not known with reasonable certainty. Also, if the properties of primary components which support a number of secondary components are estimated with low confidence, the user should check whether failure of these components is causing cascading failure of the supported components when viewing the output in the Postprocessor.

If the properties of a significant number of building components can only be estimated with low confidence, the FACEDAP code should only be used to get a rough bounds on the possible building damage. For this case, a set of upper bound component properties can be defined during the first FACEDAP analysis run and then the building input can be reanalyzed with lower bound property estimates during a second analysis run.

8.6 Building Geometry

Building Geometry is the fourth input option on the main Preprocessor menu. Input into the *Building Geometry* section defines the location of all the components within the building relative to each other and relative to the explosive charge, and assigns a previously input set of component properties to each component. Because of this last factor, **Component Property definition must precede Building Geometry Definition.** Building Geometry input is divided into three categories in the primary *Building Geometry* menu; *Wall/Roof Area Definition*, *Component Geometry Definition*, and *Dependencies*. A sample of the input screen with this menu follows. These three categories should be selected only in the sequence listed above because the geometry of building components is defined in a step-by-step manner where the information input in each step, or category is used by the program to simplify input of information into the next step. Input for each category is made into spreadsheet-type input screens which are selected using submenus.



The basic flow of the building geometry is as follows. First, the user is required to divide the building into large planar wall and roof areas. These are called wall/roof areas. Usually, the building is divided into five such areas made up of the four walls and the roof. However, the building can also be divided into more wall/roof areas if this will simplify the input. The global coordinates of the four corners of each wall/roof area, and a descriptive name for each area, are input into the *Wall/Roof Area Definition* spreadsheet. The order in which these corners are input is very important as explained in the next section. After this is completed, the user returns to the main *Building Geometry* menu and selects *Component Geometry Definition.* This causes a secondary menu to appear with a list of the names of all the input wall/roof areas. The user selects these wall/roof areas one-by-one and then defines each component in the selected wall/roof area by defining; 1) the component type, 2) the Component Property Name assigned by the user to the set of cross sectional and material properties which describe the properties of the component, and 3) the end point or corner point coordinates of the component in the local coordinate system of the selected wall/roof area. The user may use the capabilities of the Preprocessor to generate similar components within a wall/roof area at uniform spacing as is explained in Section 8.6.3.1.3. When all components in all wall/roof areas have been defined, the user returns to the main *Building Geometry* menu and selects *Dependencies.* This causes the secondary menu with the list of all wall/roof area names to appear again. The user then selects the wall/roof areas one-by-one again and the Preprocessor generates the dependencies, or the list of components which support each component, in the selected wall/roof area. The user should inspect the calculated dependencies, edit them if necessary, and then return to the secondary menu and select another wall/roof area until the dependencies for the components in all wall/roof areas have been generated. When this is completed, then the Building Geometry input is complete and the user should return to the main Preprocessor menu.

8.6.1 Preparation of Input

The information in this section supplements similar discussion in Sections 8.1.3 and 8.1.4. The preceding introductory paragraphs and Section 8.1.3 discuss the division of the building into large planar areas called wall/roof areas. **Wall/roof areas should be chosen so that: 1) all areas of the building exterior and roof are included in one, and only one wall/roof area; 2) all building components are entirely within a single wall/roof area; 3) all the wall/roof areas are** four sided, planar areas; 4) there is at least one **non-vertical wall/roof area (i.e., one roof area); and 5) each input vertical wall/roof area (i.e., each wall area) lies in a plane which includes one edge of an input roof area.** When possible, it is recommended that the four walls and roof are selected as the wall/roof areas for the building since this simplifies input and maximizes the number of components within each wall/roof area. Maximizing the number of components in a wall/roof area increases the number of components which can be generated by the Preprocessor with a single "Master" component. More than one wall/roof area will be required in "H" and "L" shaped roofs and in sloped roofs which peak at the center, due to restriction No. 3 above.

Restrictions No. 4 and 5 in the preceding list are necessary because the Preprocessor associates an Interior Building Node with each component in a given wall area. The Interior Building Node assigned to each wall area is a point inside the building directly behind the given wall area. The point is calculated by the FACEDAP code as a point which is below the building roof area which has an edge that is in the vertical plane of the wall/roof area. This node is used to establish the outward normal from the wall. The direction of the outward normal is used to determine whether components on the wall face the charge, and therefore receives a more intense, "reflected" blast loading, or whether they face in another direction and therefore receive a less intense, "free-field" blast load. In a H shaped building the Preprocessor does NOT check whether another section of the building which is between the charge and a given wall area facing the charge, shields the given wall. If this is the case, the blast damage in the shielded wall will be overestimated.

After the building has been divided into wall/roof areas, the global coordinates of the wall/roof area corner points should be defined. This is discussed in Section 8.1.3. **There are only two rules which govern the choice of a global coordinate system; 1) the global X and Y axes must lie in the plane of the ground surface, and 2) the global Z axis must have its zero point at the ground surface and be positive upward from the ground.** Restriction No. 2 implies that the Z coordinate of the explosive charge and all building corners is equal to their height off the ground. Within these two restrictions, any convenient point (such as the point on the ground beneath the charge) can be designated (0,0,0) in the global coordinate system and the global coordinates of the corners of the wall/roof areas and the charge can then be chosen to cause the desired location of the charge relative to the building. It is recommended that the global X and Y axes be chosen parallel with two perpendicular sides of the building floor plan since this greatly simplifies input.

Finally, all the wall/roof areas should be drawn in plan or elevation view and the global coordinates of each corner should be labeled, and the four corners should be labeled 1 through 4. This is illustrated in Figure 2. The numbering must be sequential along the perimeter of the wall/roof area in either the clockwise or counterclockwise direction. **This numbering of the four corners of each wall/roof area is very important since it establishes the local coordinate system within the wall/roof area.** The user should input the global coordinates of the four wall/roof area corners into the Preprocessor in the numerical order defined by the assigned numbers. **The x axis in the wall/roof area local coordinate system extends from Corner No. 1 to Corner No. 2, positive towards Corner No. 2.** Therefore, the four corners of each wall/roof area should be numbered so that the side of the wall/roof area which the user chooses as the local x axis extends from Corner No. 1 to Corner No 2. **Corner No. 1 is (0,0) in the local coordinate system. The local y axis is perpendicular to the local x axis and positive in the direction towards Corner No. 4.** For a typical rectangular or square wall/roof area, the local y axis will be along the side between Corners No. 1 and 4 and positive in the direction towards Corner No. 4. Some possible "rules" for choosing the local coordinate systems for wall/roof areas follow; 1) number the corners of all wall areas so that the local x direction extends along the side which intersects the ground surface is at the bottom

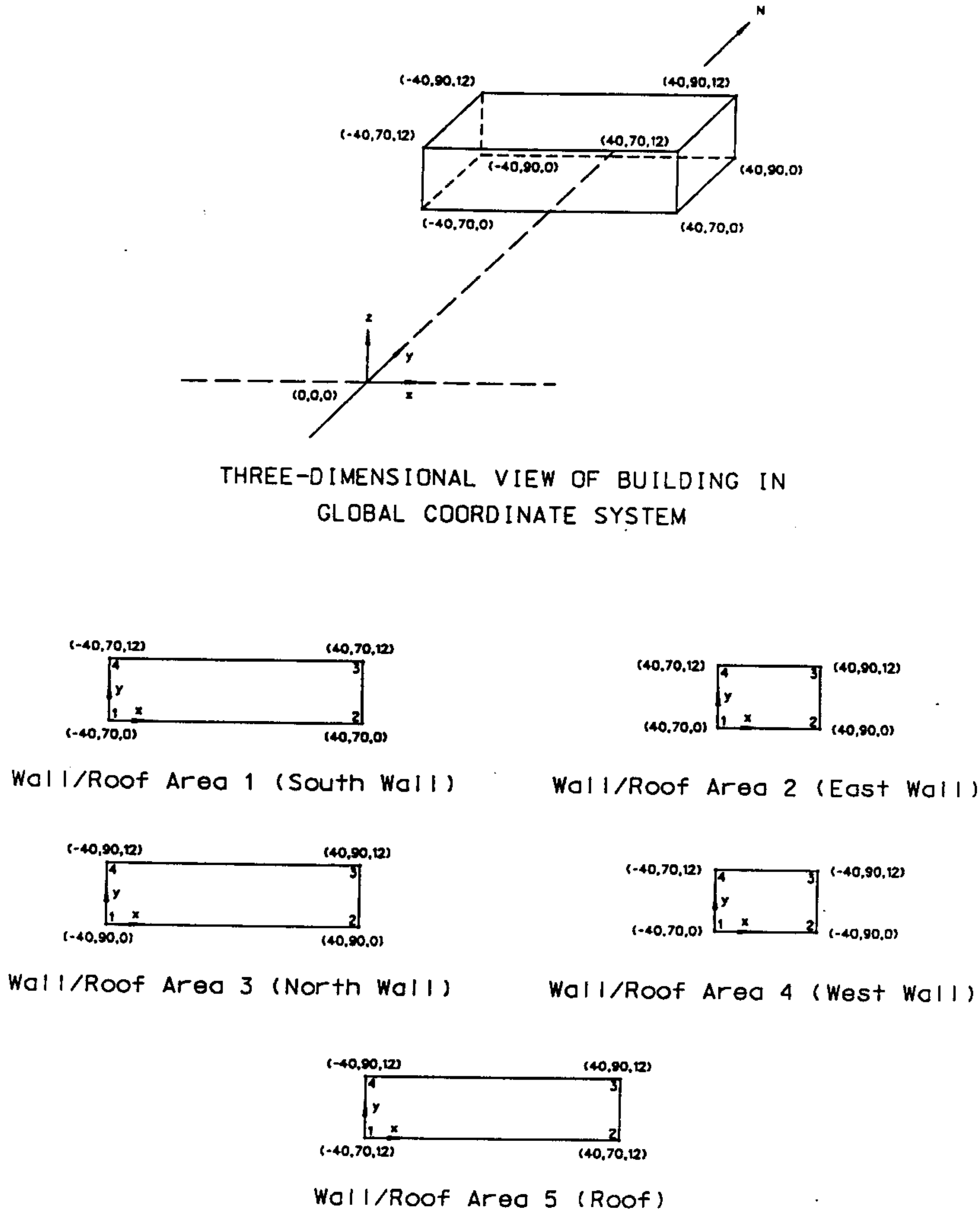


Figure 2. Wall/Roof Areas for 80 ft x 20 ft x 12 ft Building Loaded by an Explosion at a 70 ft Standoff Opposite the Center of the South Wall

of the wall areas is at the bottom of the wall area, 2) number the corners of wall areas on opposite building walls so that the local x and y axes in these walls are parallel and positive in the same direction, and 3) number the corners of all roof areas so that the local x axis is along the side closest to the explosive charge. The user can choose any numbering system which is sequential around the perimeter of the wall/roof area and causes a convenient local coordinate system.

Endwalls and roofs of buildings with a roof which peaks in the middle will need to be divided into two separate wall/roof areas for input into the program. Non-rectangular, trapezoidal walls can be input as single wall areas. Triangular wall areas must be entered as four sided areas with a short fourth side. The use of separate local coordinate systems for each wall/roof area allows the user to input the coordinates of building components directly from plan or elevation views of the building without considering the global coordinate system. *However, the existence of separate local coordinate systems for each wall/roof area can lead to some confusion. Therefore, it recommended that; 1) the user chooses the local coordinate systems of wall/roof areas in as uniform and consistent a manner as possible throughout the building using "rules" such as those recommended above, and 2) the user sketches each wall/roof area as shown in Figure 2.* Each wall/roof area should also be assigned a short, descriptive name as shown in Figure 2.

8.6.2 Wall/Roof Area Definitions

The process of defining input into the *Building Geometry* section of the Preprocessor should begin by selecting the *Wall/Roof Area Definitions* in the main *Building Geometry Definitions* menu. **Wall/roof areas must be defined before any of the other Building Geometry options can be exercised.** After the *Wall/Roof Definition* option has been selected, the Wall/Roof Area Definition spreadsheet is displayed. A portion of this spreadsheet follows.

Press F3 for Info. on Defining Corner Points

WALL/ROOF AREA DEFINITION

Row	Name	Corner Point 1 X1 (ft)	Y1 (ft)	Z1 (ft)	Corner Point X2 (ft)	Y2 (ft)
1	SOUTH WALL	-80.	70.	0.	80.	70.
2	EAST WALL	80.	70.	0.	80.	130.
3	NORTH WALL	-80.	130.	0.	80.	130.
4	WEST WALL	-80.	70.	0.	-80.	130.
5	ROOF	-80.	70.	12.	80.	70.
6						
7						
8						
9						
10						

F1 HELP F2 Menu

Enter a Unique Name of Wall/Roof Area for Future Reference

The building should be divided into wall/roof areas, the global coordinates of the four corners of each wall/roof area should be determined, and the corners should be numbered as discussed in the previous section. **The user should input the global coordinates of the four corner points of each wall/roof into the spreadsheet in the order which defines the desired local coordinate system of each wall/roof area.** The coordinates of all the corner points in each wall/roof area are input into a separate row of the *Wall/Roof Area Definitions* spreadsheet. A short, descriptive name (10 character maximum) should also be input for each wall/roof area. This name is used throughout the input and output to refer to the wall/roof area. The names of the wall/roof areas should be unique and no area should be named FRAME because this is a reserved name. Table 11 gives a detailed description of the *Wall/Roof Area Definitions* spreadsheet input parameters. Please notice that the global coordinates of all corners of wall/roof corners must be input in the **units of feet**. A maximum of 50 wall/roof areas can be input.

Table 11. Wall/Roof Area Definition Input Parameters

COLUMN HEADER	UNITS	VARIABLE TYPE	FIELD TYPE	HELP MESSAGE
Name	N/A	char	user entry	Enter a Unique Name of Wall/Roof Area for Future Reference
X1	ft	real	user entry	Enter X Coordinates for 1st Corner of Wall/Roof Area in GLOBAL Coordinates
Y1	ft	real	user entry	Enter Y Coordinates for 1st Corner of Wall/Roof Area in GLOBAL Coordinates
Z1	ft	real	user entry	Enter Z Coordinates for 1st Corner of Wall/Roof Area in GLOBAL Coordinates
X2	ft	real	user entry	Enter X Coordinates for 2nd Corner of Wall/Roof Area in GLOBAL Coordinates
Y2	ft	real	user entry	Enter Y Coordinates for 2nd Corner of Wall/Roof Area in GLOBAL Coordinates
Z2	ft	real	user entry	Enter Z Coordinates for 2nd Corner of Wall/Roof Area in GLOBAL Coordinates
X3	ft	real	user entry	Enter X Coordinates for 3rd Corner of Wall/Roof Area in GLOBAL Coordinates
Y3	ft	real	user entry	Enter Y Coordinates for 3rd Corner of Wall/Roof Area in GLOBAL Coordinates
Z3	ft	real	user entry	Enter Z Coordinates for 3rd Corner of Wall/Roof Area in GLOBAL Coordinates
X4	ft	real	user entry	Enter X Coordinates for 4th Corner of Wall/Roof Area in GLOBAL Coordinates
Y4	ft	real	user entry	Enter Y Coordinates for 4th Corner of Wall/Roof Area in GLOBAL Coordinates
Z4	ft	real	user entry	Enter Z Coordinates for 4th Corner of Wall/Roof Area in GLOBAL Coordinates

As with the Component Properties, the first blank wall/roof area name is assumed by the code to mark the end of wall and roof area input. Any rows of data input after the first row with a blank name will be ignored. Press F2 to exit from the spreadsheet and return to the *Building Geometry Definitions* Menu.

8.6.2.1 Error Checking

As the Preprocessor returns the user to the *Building Geometry Definitions* Menu, the following error checking is performed.

1) Check if user changed the name of a wall/roof area. If this is true all references within the Preprocessor storage structure are updated with the new name. The user must make any required updates to the name of the "blastward wall" of any reinforced concrete frame or steel frame components previously defined in the *Frame Definitions* spreadsheet (see Section 8.6.3.2).

2) Check if a row was deleted on the spreadsheet. If this is true all references to the deleted wall/roof area (all building components contained within that area) are also deleted. The user is warned of this before a wall/roof area is deleted.

3) Check that all wall/roof areas have unique names.

4) Check that no wall/roof area uses the reserved word FRAME.

5) Check that no two wall/roof areas have four identical corner coordinates.

6) Check that no wall/roof area has two corners with the same coordinates.

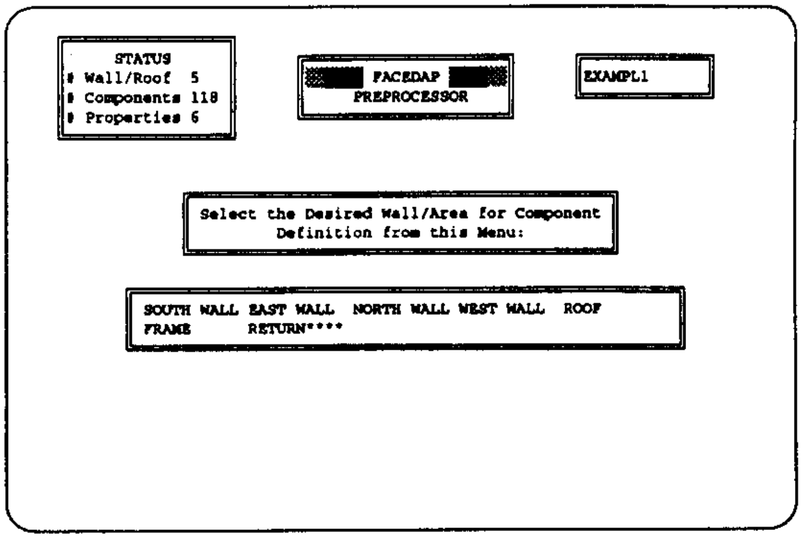
7) Check that each corner of all wall/roof areas matches at least one corner from another wall/roof area within a tolerance of 0.2 ft. This helps to ensure that no overlapping or gaps occur in the defined wall/roof areas.

8) Check that wall/roof areas are planar.

The first two checks are internal to the Preprocessor and no action is required by the user except as stated in the first check. If the other checks determine that an error in the input has occurred, a short descriptive error message is displayed and the user must reenter the *Wall/Roof Area Definitions* spreadsheet and make the necessary corrections. The user should not enter the other two options in the main *Building Geometry Definitions* menu until the errors are corrected. See the alphabetical listing (by the first letter of the error description) of Preprocessor Error Messages in Section 12.2 if an error message is not self-explanatory.

8.6.3 Component Geometry Definitions

After all the wall/roof areas have been defined and any errors have been corrected, return to the main *Building Geometry Definitions* menu and select the *Component Geometry Definitions* option. This causes a submenu with a list of the names of all the wall/roof areas defined in the *Wall/Roof Area Definitions* phase, as well as the option FRAME, to be displayed on the screen. A sample of this menu follows.

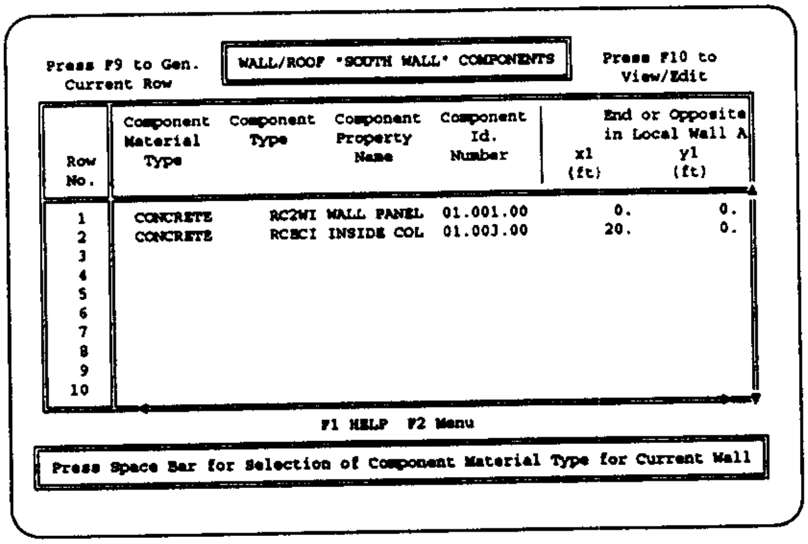


The user should sequentially select each of the wall/roof areas; input information defining all the components located within the selected wall/roof area; and then press F2 to return to the submenu listing all the wall/roof area names. All non-frame components should be entered once into one wall/roof area. If the component lies along a boundary between two wall/roof areas, it can be defined in either of the wall/roof areas but it should not be defined in both wall/roof areas. Reinforced concrete frames and steel frames typically do not lie within a single wall/roof area. Therefore, all frames must be defined using the Frame option shown on the menu with all the named wall/roof areas. Selecting the *RETURN***** option, transfers control back to the main *Building Geometry Definitions* menu. See Section 6.3 Multi Row and Column Menu if necessary for an explanation of how to make selections from this menu. A maximum of 370 building components can be defined in a single building analysis. If there are more than 370 building components, see the discussion in Section 8.1.6.

If the user only wants to calculate the blast damage to a single component, this component can be defined within any convenient wall/roof area where it will receive the desired blast load. This will conclude the building geometry input for this case and the input can then be saved, validated, analyzed, and output as it would be for a case where the building contained more components. The validation module, which is discussed in Section 9.0, will issue warning messages that no components have been defined in some wall/roof areas and no dependencies have been generated, but this will not prevent analysis of the component damage.

8.6.3.1 Component Geometry Definition for Non-Frame Components

When the user selects a named wall/roof area from the preceding menu, the *Component Geometry Definitions* spreadsheet is displayed. A portion of this spreadsheet input screen follows. The selected wall/roof area name is displayed at the top of the input screen for reference. See Section 6.5 for a full explanation of data entry into a spreadsheet-type input screen.



Information which either defines the geometry of a component within the selected wall/roof area, or defines parameters needed by the Preprocessor to generate component geometries, is entered into separate rows of this input screen. The user defines the component geometry of each input component by inputting the location of the end point or corner point coordinates of the component (in the wall/roof local coordinate system), the component type, and the name of the component property set which contains the cross sectional and material properties of the component. This information is entered into the first eight columns of the spreadsheet. Parameters are input into the remaining columns if the component is a "Master" component that is used to generate additional components.

The user can generate components within the selected wall/roof area which are geometrically similar and are offset from each other at a typical spacing. The geometry of all the components in such a group can be generated by: 1) defining the end point or corner point coordinates, Component Type and Component Property Name of the component **closest** to the (0,0) in the wall/roof local coordinate system; 2) identifying this component as a Master component in the ninth column of the *Component Geometry Definitions* spreadsheet; 3) defining the typical spacing between the components, the direction (local x or y direction) of the spacing, and the number of components to be generated (in *addition* to the Master component) in the last three columns of the spreadsheet; and 4) pressing the F9 key while the cursor is in the row of the Master component. The last step causes the Preprocessor to "generate" all the components in the group. A maximum of 50 components can be generated from a master component. All generated components are assigned the same Component Type and Component Property Name as the Master component. The user can change this by editing the generated components. The generated components can be viewed and edited by pressing the F10 key. Any subsequent generation of components (with the F9 key) will overwrite previous user edits. The user is warned of this consequence by the Preprocessor. It is important to regenerate components with the F9 key if any change is made to the Master component after components have initially been generated.

A maximum of 150 components should be input into a Component Geometry Definitions spreadsheet for any given wall/roof area. This will avoid a problem when dependencies (explained in Section 8.6.4) are calculated because only 150 total components per wall/roof area (including generated components) can be considered when calculating dependencies. If a wall or roof area contains too many components, the user can return to the *Wall/Roof Area Definitions* spreadsheet and divide the wall or roof area into two areas.

The preceding discussion is an overview of the *Component Property* input. Table 12, which summarizes all fields on the *Component Geometry Definitions* spreadsheet, follows. This table makes reference to "Master" and "Unique" components. Components on the *Component Geometry Definitions* spreadsheet are **Master** components if they are used to generate other components and they are **Unique** components if they are not used to generate any components. This table and others which follow make reference to "option" fields. These are columns in the spreadsheet where the user should input information by pressing the space bar and making a selection from the pop-up menu of acceptable input values which appears. The Component Property Names entered into Column 3 in the spreadsheet must be names of component property sets previously defined with the Component Property option in the main Preprocessor menu for the component type designated in Column 2. *As in previous spreadsheets, the program interprets the first row with a blank first column as the end of the spreadsheet data.*

8.6.3.1.1 Information on the Input of Component End points and Corner Points

The coordinates x1, x2, y1, and y2 in Table 12 are used to determine: 1) the span length of the component; 2) the component center point where the blast load will be calculated; and 3) how the components is supported by and supports other components. The coordinates x1, x2, y1, and y2 should define two end points of one-way components, beam components, and exterior column components. The span length for these components is calculated as the distance between the input end point coordinates. The coordinates x1, x2, y1, and y2 should define two opposite corners of two-way components. One span of the two-way component is calculated as the distance between x1 and x2 by the code, and the other span is calculated as the distance between y1 and y2. The blast load on each component is calculated at the midpoint between the input component end points or corner points. The blast load on interior columns is calculated at the top end point of the column, where it intersects with the roof. This is the only end point which should be input for wood, steel and concrete columns. The column length is a component property input. The dependencies, which are explained in Section 8.6.4, describe how the building components support each other. They are calculated based on rules described in Section 8.6.4.1 that check for components with end points which match (occupy the same point in space) or which lie along the line between the end points of another component. The FACEDAP program internally converts the component end or corner points, which are input in terms of the local wall/roof coordinate systems, into the global coordinate system.

Table 12. Component Geometry Input Parameters

COLUMN HEADER	UNITS	VARIABLE TYPE	FIELD TYPE	HELP MESSAGE
Component Material Type	N/A	char	option	Press Space Bar for Selection of Component Material Type for Current Wall
Component Type	N/A	char	option	Press Space Bar for Selection of Component Type for Current Wall
Component Property Name	N/A	char	option	Press Space Bar for Selection of Component Property Name for Current Wall
Component Id. Number	N/A	char	read-only	Component Identification Number - Assigned by BDAMPREP
x1	ft	real	user entry	Enter End or Opposite Corner x1 Coordinate in LOCAL Wall Coord. System
y1	ft	real	user entry	Enter End or Opposite Corner y1 Coordinate in LOCAL Wall Coord. System
x2	ft	real	user entry	Enter End or Opposite Corner x2 Coordinate in LOCAL Wall Coord. System
y2	ft	real	user entry	Enter End or Opposite Corner y2 Coordinate in LOCAL Wall Coord. System
Master Component for Comp. Generation	N/A	char	option	Press Space Bar to Indicate Master (YES) or Unique (NO) Component
Center to Center Spacing	ft	real	user entry	Center to Center Spacing in +x or +y Direction from Master to Gen. Comp.
Local Direction of C/C Spacing	N/A	char	option	Press Space Bar to Indicate Generation in +x or +y from MASTER Component
Number of Components to be Generated	integer	integer	user entry	Enter the Number of Additional Components in Current Wall/Roof area to be Generated

These factors require that the user considers the following guidance when inputting component end points and corner points. For **one-way components**, which include all one-way slab, one-way masonry, beam, joist, metal stud wall, exterior column, wood wall, and wood roof components, **the x1, x2, y1, and y2 coordinates should be two end points of the component which define the span length of the component.** In the case of one-way slabs, one-way walls, wood walls and wood roofs, **the end points which define the span of the component and which are at (or at least near) the center of the component width should be input.** Center points along the width should be input because this will cause the midpoint between the end points, which is where the blast load on the component is calculated, to be in the geometrical center of the component. Since the blast load on a component varies with its distance from the blast, a long wall should be broken into a limited number of separate components which have an uniform blast load over their width. Widths of ten to twenty feet are usually very adequate for this purpose as long as the scaled charge standoff (the distance from charge to component in ft divided by the cube root of the charge weight in lb) is at least 3.0 $ft/lb^{1/3}$. The section width which is input as a component property for these components is NOT related to the width between one-way component end points referred to in this paragraph. The section width is a separate dimension and is defined as shown in Section 8.5.2.

A final consideration when assigning end points to one-way slabs, one-way walls, wood walls and wood roofs, is the need to choose end points for these components which will match with the end points of any component which is supported by the one-way component. For example, if a wood wall component supports a wood roof component, the input end points of the two components should be such so that after the Preprocessor internally converts all input end and corner points to global coordinates, the top end point of the wood wall component will be very near (within 1 ft) of the end point of the wood roof component which bears onto the wall component. The end points must match. Failure to do this will prevent the Preprocessor from calculating the fact that the wood roof component is dependent on (supported by) the wood wall component. The user can define this dependency directly as explained in Section 8.6.4, but it is much easier to let the Preprocessor calculate dependencies. The case where a wood beam is supported by a wood wall is analogous. When a beam or column supports one of the one-way components referred to above, a dependency will be calculated by the Preprocessor anytime an end point of the one-way component is anywhere within the length of the column or beam. The manner in which dependencies are calculated by the Preprocessor is summarized in Tables 16 and 17 in Section 8.6.4.1.1.

For **two-way components**, which include two-way slab and two-way masonry components, the (x1, y1) and (x2, y2) coordinates should define the coordinates of two OPPOSITE corner points of the components which do not both lie along the same side of the component. In other words, the two input corner points should lie along a diagonal through the component. These coordinates define the span of the component in both directions. As mentioned above, the distance between x1 and x2 defines one span the component and the distance between y1 and y2 defines the other span. The aspect ratio, or the ratio of the short span to long span, must be greater than 0.09 for components with simple support conditions and must be greater than 0.3 for components with fixed support conditions. If the aspect ratio is less than these limits, the component should be input as a one-way component of the same material type and type of reinforcement (reinforced or unreinforced). For all **interior column components**, only the x1 and y1 coordinates of the top end point of the column which lies in the plane of the roof, should be input and the x2 and y2 data fields should remain BLANKED. Interior columns must be entered into the roof area where the column intersects the roof and the top column end point should be defined in the local coordinate system of that roof area.

8.6.3.1.2 Component ID Numbers

The user should NOT enter any information into the column entitled COMPONENT ID NUMBER. Component numbering is controlled by the Preprocessor, since the component ID numbering system is structured to allow the code to quickly access and keep track of component information. Each time control is returned to the Preprocessor, the program determines the component ID number for any new components and displays this number for the user. The first 2 digits of the Component ID Number (ahead of the first decimal point) indicate the internal program ID number assigned to the wall/roof area containing the component. If no rows have been deleted from the *Wall/Roof Area Definitions* spreadsheet, this wall/roof area ID number will correspond to the row number of the wall/roof area in the *Wall/Roof Area Definition* spreadsheet.

The next 3 digits in the Component ID Number are assigned sequentially to master and unique components in each Component Geometry spreadsheet. Thus, the component defined in row 1 of the *Component Geometry Definitions* spreadsheet for the wall/roof area in the fourth row of the *Wall/Roof Area Definitions* spreadsheet (assuming no row deletions in either spreadsheet) has a Component ID Number equal to 04.001.00.

The last two digits of the Component ID Number are used to sequentially number components generated from a Master component. If 10 components are generated from a Master component with the Component ID Number 04.001.00, the Component ID Number of the first generated component (closest to the master component) will be 04.001.01, the Component ID Number of the second generated component will be 04.001.02, etc. These 10 components all have the same first two digits (they are on the same wall or roof area) and the same middle three digits (they have the same three digits as the "Master" from which they were generated). Since only Master or Unique components are on the *Component Geometry Definitions* spreadsheet of a wall/roof area, the last two digits of these components are always "00". Generated components can be viewed by the user as explained in the next subsection. Deleted components are not replaced in the numbering system.

8.6.3.1.3 Generating Components from a Master Component

The procedure used to generate components from a Master component is explained in a step-by-step procedure here for convenient reference. If there is a group of geometrically similar components <u>within</u> a wall/roof area spaced relative to each other at a uniform spacing, then the geometry of all the components in this group can be input in a single row of a *Component Geometry Definitions* spreadsheet in the following manner.

1) Locate the component in the group CLOSEST to (0,0) in the wall/roof local coordinate system, this becomes the "Master" component.

2) Enter all information required for the Master component up to the ninth column of the spreadsheet, headed "*Master Component for Comp. Generation*", in the normal manner.

3) While on the field "*Master Component for Comp. Generation*", use the [Space Bar] to "pop-up" a Yes/No menu. Choose the "Yes" option to generate components.

4) Enter the additional information into the last three columns of Table 12. These three columns contain the uniform center to center spacing between the similar components, the direction of spacing between components (local x or local y direction), and the number of additional components (NOT including the "Master" component which is being entered) that should be generated.

These steps provide the Preprocessor with the information necessary for component generation. However, no component generation occurs until the user presses F9 while the cursor is in the row with the Master component. This keystroke causes the Preprocessor to generate the additional components. After the components have been generated, the following message is displayed.

COMPONENTS GENERATED Press Any Key Followed by F10 to View/Edit

A keystroke (any keystroke) must then be used to return control to the spreadsheet. Press F10, as stated in the message, in order to inspect the generated components. This invokes a new spreadsheet, the *Master Component* spreadsheet, to the screen showing only the generated components from the current "master" component. This spreadsheet, which is described in Table 13, may be edited by the user in the same manner as the previous *Component Definition* spreadsheet.

The F2 key returns the user to the *Component Geometry* spreadsheet. The F10 key can be used at any time to view or edit the components generated from a "master" component. As with the F9 key, the F10 key will access the components generated from the master component in the row where the cursor is located when the key is pressed.

The Preprocessor generates the input number of additional components at the input spacing with the same Component Type and Component Property Name as the Master component. Usually no editing is required. If editing is required, the user should note that all edits to the *Master Component* spreadsheet will be overwritten, and therefore lost, if the F9 key is used to regenerate components from the given Master component. Since the user will usually use the F9 key to propagate any change/corrections to a Master component through to the generated components, it is best to hold off on any edits to the *Master Component* spreadsheet until all other input seems correct. If necessary, the user can use the F10 key to access generated components and propagate a change/correction in a Master component to the generated components by editing each component. Any time the user selects the F9 key to *re*generate components from a master component, the following message is displayed which warns that the regenerated components will overwrite the existing generated components.

ALL PREVIOUS EDITS FOR GENERATED COMPONENTS WILL BE LOST! ARE YOU SURE?? (N)o NOT SURE - Do Not Generate (Y)es - Generate

Regeneration will occur if the user responds affirmatively to the message. If the F10 key is pressed prior to generation of any components from a Master component, the components will automatically be generated and then the "Master" spreadsheet will be displayed. Also, if the user leaves the *Component Geometry* spreadsheet for a given wall/roof area and no components have been generated from a Master component with the F9 key, the components will be generated automatically and stored by the Preprocessor. However, it is safest to always generate components with the F9 key and then view them with the F10 key. **It is very important for the user to either regenerate components with the F9 key after any change/correction is made in a Master component or use the F10 key to access the generated components and propagate the change/correction through to the generated components by editing all these components.**

Table 13. Parameters in Spreadsheet Showing Components Generated from a Master Component

COLUMN HEADER	UNITS	VARIABLE TYPE	FIELD TYPE	HELP MESSAGE
Component Material Type	N/A	char	option	Press Space Bar for Selection of Component Material Type for Current Wall
Component Type	N/A	char	option	Press Space Bar for Selection of Component Type for Current Wall
Component Property Name	N/A	char	option	Press Space Bar for Selection of Component Property Name for Current Wall
Component Id. Number	N/A	char	read-only	Component Identification Number - Assigned by BDAMPREP
x1	ft	real	user entry	Enter End or Opposite Corner Coordinate in LOCAL Wall Coord. System
y1	ft	real	user entry	Enter End or Opposite Corner Coordinate in LOCAL Wall Coord. System
x2	ft	real	user entry	Enter End or Opposite Corner Coordinate in LOCAL Wall Coord. System
y2	ft	real	user entry	Enter End or Opposite Corner Coordinate in LOCAL Wall Coord. System

After the component geometry of all components in a given wall/roof area have been input, press [F2] to exit from the component geometry spreadsheet and return to the list of wall/roof area names. When the component geometry for all wall/roof areas and frames in the building is complete, select *RETURN***** to return to the main *Building Geometry Definitions* menu. Input of frame component geometry is discussed next.

8.6.3.1.4 Error Checks

After the user exits the component geometry spreadsheet, the following error checking is performed.

1) Check if any Master components were changed to Unique components while on the spreadsheet. The user is prompted whether to restore the component to Master status or to leave it as a Unique component and delete all components which had been generated from it.

2) Check if all required inputs are defined.

3) Check if a mismatch exists between the input Component Material Type and the input Component Type for a component, where the material type (i.e., concrete) is not consistent with the component type (i.e., steel beam).

4) Check if a mismatch exists between the input Component Property Name and the input Component Type for a component, where the assigned component property name is not the name of a set of component properties previously defined for the given component type.

5) Check that the end points of all one-way components lie along a line parallel to the local x or y coordinate axis.

6) Check that two-way components have input corner points which are diagonal with respect to each other in the local coordinate system and have an aspect ratio greater than 0.09 if simply supported or greater than 0.3 if the supports are fixed (no support rotation).

7) Check that Metal Stud Wall components, all Masonry components, all Exterior Column components, and Wood Wall components are located only in wall (vertical) wall/roof areas.

8) Check that Steel Open Web Joists, all Interior Column components, and all Wood Roof components are located only in a roof (non-vertical) wall/roof area.

9) Check that the data fields for the x2 and y2 coordinates of all Interior Column components are blank. The Preprocessor blanks these fields if they are not blank.

10) Check that no input component end point or corner point coordinates are outside the boundaries of the wall/roof area where they are input.

Messages 5 through 8 are only warnings (except for the aspect ratio limitation) since these conditions are unexpected but are not precluded. If such unusual situations are input intentionally, the user must inspect the calculated component Dependencies (see Section 8.6.4) very carefully and edit them where necessary since the rules used to calculate dependencies are based on typical conditions. See the alphabetical listing (listed alphabetically by the first letter of the error description) of Preprocessor Error Messages in Section 12.2 if a warning message is not self-explanatory.

8.6.3.2 Component Geometry Definition for Frame Components

The geometry of reinforced concrete and steel frame components is input into the *Frame Definition* spreadsheet by selecting the FRAME option from the menu of wall/roof areas under the *Component Geometry* option. At least one component property set for a reinforced concrete frame or a steel frame must have been input using the *Component Properties* option in the main Preprocessor menu before the *Frame Definition* spreadsheet will be displayed. This restriction exists because a previously defined Component Property Name must be assigned to each frame component input in the spreadsheet. **The required input into the Frame Definition spreadsheet includes;** 1) the name of the component property set which contains the cross sectional and material properties of the frame, 2) the name of the wall area where lateral blast load is applied to the frame (this is the wall area closest to the charge which contains an exterior column of the frame - it is called the "blastward" wall area), 3) the coordinates of the base of the exterior column of the frame which is in the blastward wall area (in the local coordinate system of the blastward wall area), and 4) the frame height (the height of the frame roof beam off the ground). The Preprocessor cannot generate frame components and therefore each frame in the building must be input into a separate row of the spreadsheet. A portion of the *Frame Definition* spreadsheet follows.

Input all Frames
Line by Line

FRAME DEFINITION

Row No.	Component Type	Component Property Name	Component Id. Number	GROUND LEVEL Endpoint of Column in Wall Nearest Charge Local Wall Area Coordinates Wall Name	x1 (ft)	y1 (ft)
1	RCMRFI	FRAME 2	00.001.00	WALL 1	10.	0.
2	RCMRFI	FRAME 1	00.002.00	WALL 1	25.	0.
3	RCMRFI	FRAME 1	00.003.00	WALL 1	40.	0.
4	RCMRFI	FRAME 1	00.004.00	WALL 1	55.	0.
5	RCMRFI	FRAME 2	00.005.00	WALL 1	70.	0.
6	RCMRFI	FRAME 3	00.006.00	WALL 1	0.	0.
7	RCMRFI	FRAME 3	00.007.00	WALL 1	80.	0.
8						
9						
10						

F1 HELP F2 Menu

Press Space Bar for Selection of Component Type for Current Frame

Table 14 summarizes all the input parameters on the spreadsheet. As in previous spreadsheets, the first row with a blank first column is assumed to mark the end of the spreadsheet data. Frame Component ID Numbers all begin with the two digits "00", the next three digits are sequentially assigned by the Preprocessor, and the last two digits are always "00" because the Preprocessor cannot generate frame components.

Table 14. Frame Definition Input Parameters

COLUMN HEADER	UNITS	VARIABLE TYPE	FIELD TYPE	HELP MESSAGE
Component Type	N/A	char	option	Press Space Bar for Selection of Component Type for Current Frame
Component Property Name	N/A	char	option	Press Space Bar for Selection of Component Property Name for Current Frame
Component Id. Number	N/A	char	read-only	Component Identification Number - Assigned by BDAMPREP
Wall Name	N/A	char	option	Press Space Bar to Select Blastward Wall with a Frame Ext. Column
x1	ft	real	user entry	Enter Local x Coordinate of Column End point at Ground Level
y1	ft	real	user entry	Enter Local y Coordinate of Column End point at Ground Level
Total Height of Frame	ft	real	user entry	Enter Height of Frame from Floor to Roof

Frames consist of exterior columns, roof beams, and interior columns which are exposed to blast load. A two story frame is illustrated in Section 8.5.2.6. The frame damage calculation methodology in the FACEDAP program assumes that these components do not respond to the blast except as part of the overall frame. It is assumed that the blast load applied to the wall which is laterally supported by the frame column(s) closest to the explosive charge causes the entire frame to sway, or move laterally at the top relative to the ground. The maximum deflection of all frame components is the lateral deflection at the top of the columns. In some buildings, this assumed response is quite accurate. In other buildings, blast loading does not cause significant frame sway and frame components respond primarily as independent columns and beams in flexure with maximum deflection at midspan. The FACEDAP program cannot determine which of these two modes, frame response or individual component bending response, will control damage in frame components. Generally, blast damage to moment resisting steel frames in light metal buildings is controlled by frame sway. Blast damage to buildings with heavier wall and roof panels (concrete or masonry), and to buildings with shear walls or lateral bracing which resist lateral forces, is generally controlled by flexural response of individual frame columns and beams.

It is generally recommended that the user analyze buildings with moment resisting frames twice. The building damage should be taken as the larger of the building damage factors calculated with the two analyses. In the first analysis all the frame components are input as separate components (i.e., as exterior column, beam, and interior column components), the input file is run through the Analysis module, and the percentage of building damage is determined. No frame components are defined for this run. Then, the building is reanalyzed in a second analysis where the frames are defined as frame components and all the individual column and beam components which are contained within the frames are deleted. The individual frame components should be deleted from the *Component Geometry* spreadsheets of all wall/roof areas where these components were defined using the Ctrl-D keystroke. **It is very important that the user does not doubly define components in the same input file by defining them as individual exterior/interior columns or roof beams and also defining the frame which includes these columns and beams.**

It is also important to assign consistent weighting factors to frame components in both runs. The weighting factor for each frame component input during the second run should be equal to the sum of the weighting factors assigned to the individual exterior/interior columns and beams within that frame input during the first run. This will ensure that damage to the frame, whether it is flexural damage to individual frame components during the first run, or damage to the frame as a whole during the second run, is weighted equally in the building damage summation which determines the building damage factors. If this approach is not used, the two analyses will not calculate comparable building damage factors and the objective of determining which frame response mode, frame sway or flexural response or individual frame components, controls building damage will not be achieved.

After component geometry input for all frames is complete, press F2 to exit from the spreadsheet and return to the list of wall/roof area names. When the component geometry for all components in the building is complete, select *RETURN***** from the list of wall/roof areas to return to the main *Building Geometry Definitions* menu.

8.6.3.2.1 Error Checks

When the user exits the *Frame Definition* spreadsheet the following error checking is performed.

1) Check if all required input fields are defined

3) Check if a mismatch occurs between the frame Component Type and the Component Property Name.

4) Check if the Blastward Wall Area Name has been defined in the *Wall/Roof Area Definition* spreadsheet.

5) Check that the selected Blastward Wall is a wall area (should not be a roof area).

6) Check if the local coordinates of the base of the exterior column of the frame are within the boundaries of the selected Blastward Wall area.

Warning messages will always be displayed if one of the checks above is not satisfied. See the alphabetical listing (by the first letter of the error description) of Preprocessor Error Messages in Section 12.2 if a warning message is not self-explanatory.

8.6.4 Dependencies

After the user finishes defining the geometry of all components, the final step required for building geometry definition is the definition of the component dependencies. The "dependencies" of each component is the list of all components which directly support the given component. Some components are only supporting components and therefore have no dependencies. The user does not usually have to individually define the component dependencies since logic has been programmed into the Preprocessor which performs this task. However, the user must invoke the part of the Preprocessor which generates the dependencies and check the generated dependencies for any missing or incorrect dependencies. The logic in the Preprocessor is considered adequate for calculating the dependencies in most standard buildings and it has been debugged for a number of different types of buildings. However, the user is responsible for checking all dependencies generated with the Preprocessor.

The dependencies are used by the FACEDAP program ONLY if a supporting component sustains 100% blast damage and the calculated damage of the supported component is less than 100% damage. In this case, the program increases the damage level of the supported component to 100% damage based on the reasoning that the 100% damage sustained by the supporting component precludes it from actually providing support and this causes the supported component to also sustain 100% damage. This type of secondary failure of supported components is also referred to as "cascading" failure. Since the supporting components are typically designed to be at least as strong (considering their loaded area and span) as the supported components, the dependencies are not used often by the program. A notable exception to this general rule is the case of corrugated metal roof decking spanning between closely spaced open web steel joists. For this case, the decking is typically strong for its short span compared to the open web steel joists and therefore cascading failure of the decking is often calculated when the joists sustain 100% damage.

8.6.4.1 Non-Frame Dependencies

To begin generation/editing of the component dependencies the building, the user should select the *Dependencies* option from the main *Building Geometry Definitions* menu. This causes a screen to appear which lists the names of all the wall/roof areas defined in the *Wall/Roof Area Definitions* input, as well as the option *FRAME*, to be displayed. The user should sequentially select each of the wall/roof areas. When each wall/roof area name is selected, the *Dependencies* spreadsheet for that area is displayed. A portion of this spreadsheet follows. **If an area is not selected the dependencies will NOT be generated** for that area. After generated and/or editing the dependencies for the specified area, press F2 to return to the list of all wall/roof areas. Selecting *RETURN*****, transfers control back to the main *Building Geometry* menu.

Press F9 to Gen. or Regen. Depend.

WALL/ROOF "SOUTH WALL" DEPENDENCIES

Press F7 for Info. on Indepndt. Comp.

Row No.	Dependent Component Type	End or Opposite Corner Points in Local Wall Area Coordinates x1 (ft)	y1 (ft)	x2 (ft)	y2 (ft)	Dependent Component Id. Number
1	MAR2WI	0.	0.	10.	12.	01.001.00
2	MAR2WI	70.00	0.	80.00	12.	01.001.01
3	MAR2WI	10.	0.	25.	12.	01.003.00
4	MAR2WI	25.00	0.	40.00	12.	01.003.01
5	MAR2WI	40.00	0.	55.00	12.	01.003.02
6	MAR2WI	55.00	0.	70.00	12.	01.003.03

F1 HELP F2 Menu

Abbreviated Component Type Name (Press Space Bar for Full Name)

Table 15 gives a detailed explanation of each of the fields on the *Dependencies* spreadsheet.

After a wall/roof area has been selected for component dependency input, one of two possibilities will occur. If the input is new (that is, an existing input file is not being edited) or if the selected wall/roof area is in an existing file but the components in the wall/roof area had no dependencies prior to editing, then the Preprocessor will generate the dependencies of each component in the selected wall/roof area automatically. *Component generation for wall/roof areas in large buildings (over 100 total components) may take five to ten minutes depending on the computational speed of the user's PC.* Some advice for reducing the time required for dependency calculations follows later in this section. After the components are generated, a spreadsheet will automatically appear on the screen showing the name, location, and Component ID Number of each component in the selected wall/roof area and the list of the ID numbers of components which support the given component. The supported component is called the Dependent Component in the spreadsheet and the supporting components are called the Independent Components.

The second possibility is that the dependencies had previously been generated or stored for the selected wall/roof area. In this case, the *Dependencies* spreadsheet will appear showing ONLY the "old" dependencies which were previously generated. No automatic dependency generation will occur because preexisting dependencies for a selected wall/roof area are never overwritten by the Preprocessor unless the user selects F9 while in the dependencies spreadsheet for that wall/roof area. This allows the user to view previously generated and edited dependencies without overwriting previous edits. Usually, the user should use F9 to generate a new set of dependencies each time any new components or component locations have been input anywhere in the building. It is usually best to save any editing of calculated dependencies until the user is confident that all component geometry input is correct.

The user should inspect the generated dependencies and edit them as required, although usually no editing will be necessary. The user can only edit the list of Independent Components in the last four columns of the spreadsheet. Rules for generating dependencies used by the Preprocessor are described later in this section. **If the user is interested in information which helps identify a calculated Independent Component, they should locate their cursor over the ID number of**

Table 15. Component Dependency Input Parameters

COLUMN HEADER	UNITS	VARIABLE TYPE	FIELD TYPE	HELP MESSAGE
Dependent Component Type	NA	char	read-only & option	Abbreviated Component Type Name (Press Space Bar for Unabbreviated Name)
x1	ft	real	read-only	Local x Coordinate of First Component End Point
y1	ft	real	read-only	Local y Coordinate of First Component End Point
x2	ft	real	read-only	Local x Coordinate of Second Component End Point
y2	ft	real	read-only	Local y Coordinate of Second Component End Point
Dependent Component Id. Number	N/A	char	read-only	Dependent Component Identification Number - Assigned by BDAMPREP
(Independent) Component 1	N/A	char	calculated & user entry	Modify Supporting Component ID Number if Necessary
(Independent) Component 2	N/A	char	calculated & user entry	Modify Supporting Component ID Number if Necessary
(Independent) Component 3	N/A	char	calculated & user entry	Modify Supporting Component ID Number if Necessary
(Independent) Component 4	N/A	char	calculated & user entry	Modify Supporting Component ID Number if Necessary

the Independent Component of interest and **press** [F7]. This will cause the component type, wall/roof area, and local end or corner point coordinates of the Independent Component to be displayed. The local coordinates shown in this report are those in the local coordinate system of the wall/roof area shown in the report, which is where the Independent Component was defined in the *Component Geometry Definition* phase. Also, if the cursor is in the first column of the *Dependencies* spreadsheet and the [Space Bar] is pressed, the unabbreviated name of the Component Type for the given Dependent component is displayed.

If more than 200 dependencies (more than 200 total independent components) are generated in the building, the Preprocessor will warn the user that only a total of 200 dependencies can be accepted in the *Analysis* module. Also, the generation of more than 100 dependencies can be quite time consuming depending on the speed of the user's PC. **The number of dependencies in a large building can be reduced without reducing the accuracy of the calculated building damage as follows.** The user can enter only a portion of the secondary components (i.e., cladding components, girts, etc.) in a large structure (every third such component, for example) AND increase the weighting factors (see Section 8.5.2) for these components accordingly (by a factor of three for the preceding example). This will ensure that overall building damage is representatively calculated since; 1) the secondary components which are input are representatively spaced around the structure, and 2) the increase in weighting factors causes the damage to input components to also represent damage to the neighboring components, which were not input, in the building damage summation algorithm. The building damage summation algorithm is discussed in Section 11.1.1. The input of the roof components in Example Building No. 1 in Section 14.0 illustrates this approach. This input is in the file EXAMPL1.BLG on the program disk. See Section 3.0 for instructions on how to copy the example building files from the program floppy disk.

After all the generated dependencies in the wall/roof area have been inspected and edited where necessary, press [F2] to exit from the spreadsheet and return to the list of wall/roof area names. If there are any frame components, the FRAME option in this menu should also be selected as discussed in Section 8.6.4.2. When all the dependencies for all wall/roof areas have been generated and inspected, select *RETURN***** from the list of wall/roof area names to return to the main *Building Geometry* menu.

8.6.4.1.1 Rules Used to Generate Non-Frame Dependencies

Tables 16 and 17 describe the rules used by the Preprocessor to generate dependencies. These tables show how the twenty-four components are divided into five groups, called "dependency types". A component dependency is calculated if one out of eight different dependency rules called out in the two tables is satisfied. These rules allow dependencies based on; 1) the dependency type of the dependent component, 2) the dependency type of the independent component, 3) the location of the dependent and independent components (whether they are in a wall area or a roof area), and 4) matching end points, or midpoints of the independent and dependent components. "Matching" means that the points are within 1 ft of each other or that an end point of the dependent component lies along the length of a beam or column within a 1 ft tolerance. This tolerance distance was chosen since it would be very unusual for the end points of two components in a typical building which do not connect to be within 1 ft of each other. In some cases there are two candidate independent components which satisfy one of the eight rules and support the same side or end point of a dependent

Table 16. Rules Used in FACEDAP Program to Calculate Component Dependencies*

Dependent Component Type	Maximum Possible Number of Independent Components	Independent Component Type	Location of Dependent and Independent Components				
			Location Case 1	Location Case 2	Location Case 3	Location Case 4	Location Case 5
1	2	1	n/a	n/a	n/a	**1**	n/a
		2	n/a	n/a	n/a	**4**	n/a
		3	**9**	n/a	**9**	**9**	**9**
		4	**9**	**9**	n/a	n/a	n/a
		5	n/a	n/a	n/a	n/a	n/a
2	4	1	n/a	n/a	n/a	**4**	n/a
		2	n/a	n/a	n/a	**3**	n/a
		3	**5**	n/a	**5**	**5**	**5**
		4	**5**	**5**	n/a	n/a	n/a
		5	**7**	n/a	n/a	n/a	n/a
3	2	1	n/a	n/a	n/a	**1**	n/a
		2	n/a	n/a	n/a	**2**	n/a
		3	**8**	n/a	n/a	**8**	**8**
		4	**9**	**9**	n/a	**1**	n/a
		5	**6**	n/a	n/a	n/a	**6**

*Bolded numbers in table represent different rules explained on next page.

Location Case	Description
1	Both dependent and independent components in same wall or roof area.
2	Dependent and independent components in adjacent wall areas.
3	Dependent component in a wall area and independent component in an adjacent roof area.
4	Dependent component in a roof area and independent component in an adjacent wall area.
5	Dependent and independent components in adjacent roof areas.

Table 17. List Showing the "Dependency Type" of Each Component

Component	Dependency Type
Concrete Reinforced Beam	3
One-Way Reinforced Concrete Slab	1
Two-Way Reinforced Concrete Slab	2
Exterior Reinforced Concrete Column	4
Interior Reinforced Concrete Column	5
Prestressed Concrete Beam	3
Steel Beam	3
Metal Stud Wall	1
Open Web Steel Joists	3
Steel Corrugated Decking	1
Exterior Steel Column	4
Interior Steel Column	5
One-Way Unreinforced Masonry Wall	1
Two-Way Unreinforced Masonry Wall	2
One-Way Reinforced Masonry Wall	1
Two-Way Reinforced Masonry Wall	2
Masonry Pilasters	4
Wood Wall	1
Wood Roof	1
Wood Beam	3
Exterior Wood Column	4
Interior Wood Column	5

Rule Number	Description
1	An endpoint of dependent component must match an endpoint of independent component.
2	An endpoint of dependent component must lie along one side of a Type 2 independent component, but not within the tolerance distance of a corner.
3	A midpoint along a side of dependent component must match a midpoint along a side of independent component.
4	An endpoint of a Type 1 component must match a midpoint along a side of a Type 2 component.
5	A midpoint along a side of dependent Type 2 component must match a midpoint along a Type 3 or 4 independent component.
6	An endpoint of the dependent Type 3 component must match the (roof) point of the independent Type 5 component.
7	A corner point of the dependent Type 2 component must match the (roof) point of the independent Type 5 component. This applies only to the case of a flat slab. The FACEDAP code first looks for two way roof slabs to be dependent on beams or walls and, if the maximum four dependencies are not satisfied, goes back to consider the case discussed here.
8	An endpoint of Type 3 dependent component must lie along a Type 3 independent component, but not within a tolerance distance of either endpoint.
9	An endpoint of Type 1 or 3 dependent component must lie along a Type 3 or Type 4 independent component anywhere between the endpoints of independent component.

component. In this case, priority is given to the Dependency Type 3 component. The supported component is called the Dependent Component in Tables 16 and 17 and the supporting components are called the Independent Components as discussed in Section 8.6.4.1.

The type of wall/roof area (wall or roof area) containing the dependent and independent components is important because the *Analysis* module of the FACEDAP program calculates "secondary" dependencies between components and some precautions in the "primary" dependency calculations made by the Preprocessor must be taken to avoid the calculation of unreasonable secondary dependencies. An example of a secondary dependency is a case where Component A is dependent, or supported by, Component B and Component B is dependent on Component C. A secondary dependency exists between Component A and Component C, which is calculated in the *Analysis* module. A potential problem with this approach is that a wall component (Component A) can be laterally supported against blast loading by a roof component (Component B) which is supported vertically on a component in the opposite wall (Component C). The secondary dependency for this case is Component A in one wall dependent on, or supported by Component C in an opposite wall, which makes no sense. The logic in the Preprocessor precludes the calculation of "primary" dependencies which will lead to this situation. As a consequence, roof components providing lateral support to the top of wall components are not included as independent components except for some special cases where it is known that the potential problem which was just discussed will not occur.

8.6.4.2 Frame Dependencies

Selection of the *FRAME* option from the menu of wall/roof area names that appears when the *Dependencies* option is invoked results in a screen which allows the user to either turn off all frame dependencies or allow the FACEDAP program to calculate frame dependencies.

```
  The FACEDAP code can automatically fail all
components which have at least one endpoint
within the front wall, backwall, or roof area
supported by a frame with 100% damage in the
Postprocessor.
  Alternatively, there will be no such calcu-
lations and the user must input the appropriate
frame component ID number as the independent
component into the dependency input screen for
any (dependent) components affected by frame
failure.
  ENTER YES(Y) FOR THE FACEDAP CODE TO CALCULATE
DEPENDENCIES BETWEEN COMPONENTS AND FRAMES OR
NO(N) FOR NO AUTOMATIC FRAME DEPENDENCY
CALCULATIONS.
    (Y)es - Generate Frame Dependencies
    (N)o  - Do NOT Generate Frame Dependencies
```

component. In this case, priority is given to the Dependency Type 3 component. The supported component is called the Dependent Component in Tables 16 and 17 and the supporting components are called the Independent Components as discussed in Section 8.6.4.1.

The type of wall/roof area (wall or roof area) containing the dependent and independent components is important because the *Analysis* module of the FACEDAP program calculates "secondary" dependencies between components and some precautions in the "primary" dependency calculations made by the Preprocessor must be taken to avoid the calculation of unreasonable secondary dependencies. An example of a secondary dependency is a case where Component A is dependent, or supported by, Component B and Component B is dependent on Component C. A secondary dependency exists between Component A and Component C, which is calculated in the *Analysis* module. A potential problem with this approach is that a wall component (Component A) can be laterally supported against blast loading by a roof component (Component B) which is supported vertically on a component in the opposite wall (Component C). The secondary dependency for this case is Component A in one wall dependent on, or supported by Component C in an opposite wall, which makes no sense. The logic in the Preprocessor precludes the calculation of "primary" dependencies which will lead to this situation. As a consequence, roof components providing lateral support to the top of wall components are not included as independent components except for some special cases where it is known that the potential problem which was just discussed will not occur.

8.6.4.2 Frame Dependencies

Selection of the *FRAME* option from the menu of wall/roof area names that appears when the *Dependencies* option is invoked results in a screen which allows the user to either turn off all frame dependencies or allow the FACEDAP program to calculate frame dependencies.

```
  The FACEDAP code can automatically fail all
components which have at least one endpoint
within the front wall, backwall, or roof area
supported by a frame with 100% damage in the
Postprocessor.
  Alternatively, there will be no such calcu-
lations and the user must input the appropriate
frame component ID number as the independent
component into the dependency input screen for
any (dependent) components affected by frame
failure.
  ENTER YES(Y) FOR THE FACEDAP CODE TO CALCULATE
DEPENDENCIES BETWEEN COMPONENTS AND FRAMES OR
NO(N) FOR NO AUTOMATIC FRAME DEPENDENCY
CALCULATIONS.
    (Y)es - Generate Frame Dependencies
    (N)o  - Do NOT Generate Frame Dependencies
```

Selecting "Y", either by pressing Ⓨ or clicking the left mouse button on (Y) will cause the FACEDAP program to generate the Frame Dependencies. Selection of "N" causes no Frame Dependencies to be considered. After this input has been entered, the user will be returned to the menu of wall/roof area names.

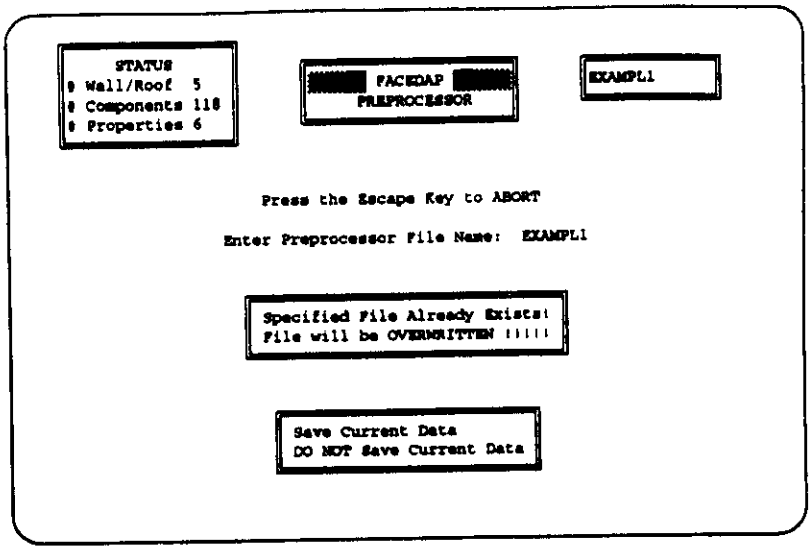
For typical situations, the FACEDAP program should be allowed to calculate frame dependencies. Frame dependencies are calculated based on the assumption that the column and beam members of the frame provide either primary or secondary support to all components within the loaded width of the frame. The loaded width is an input component property of the frame and typically it is equal to the frame spacing. For frames with 100% damage, the FACEDAP program changes the damage of ALL components within the loaded width of the frame to 100% damage if a lesser damage level was calculated due to direct blast loading. Frame dependencies are calculated immediately after the Postprocessor option is selected off the main FACEDAP program menu if the user allows the calculation of these dependencies. The damage levels of any supported components which fail as a result of cascading failure are modified by the Postprocessor prior to the display of any component or building damage results.

8.7 Save Input File

After all *Building Geometry* input is completed, the user should return to the main Preprocessor menu shown in Section 8.2 and use the *Save Input File* option on this menu to save the input into a *.BLG file. It also a good idea to save the input file intermittently during the input. All data input and/or edited prior to saving the file will be stored. As previously mentioned, all saved files have the extension .BLG and these files are saved to the default program data directory defined with the FACECFG executable or with the *Configuration* option under the *Utilities* option on the main program menu. Any modifications to an input file which are not saved will be lost when the user exits the Preprocessor and returns to the main FACEDAP program menu. Therefore, the user should always save an input file before leaving the Preprocessor unless no edits have been made.

Invoking the *Save Input File* option produces a box on the screen with the save file name. If the building was created from cleared input screens, the box is blank. In this case, the user should type in the name of the saved input file, **without any extension**, and follow with a ⏎. The Preprocessor automatically appends the .BLG extension to the file. If an existing input file has been read into the Preprocessor and edited, the box contains the name of this file. The user can save the input with this file name by pressing ⏎ or they can type in a new file name which will overwrite the old file name assumed by the Preprocessor, and then press ⏎. Only files storing input compatible with the type of analysis that was selected (i.e. single component analysis or building analysis) are displayed. **The names of files storing input for single component analysis begin with the four letter string "COMP" and names of files storing input for building analysis can begin with any other four letter string.** The program will prompt the user for a file naming beginning with "COMP" if the Single Component Analysis option has been selected in the Preprocessor. This format for naming input files allows the program to check whether a retrieved input file contains input which is compatible with the type of analysis selected by the user in the preprocessor.

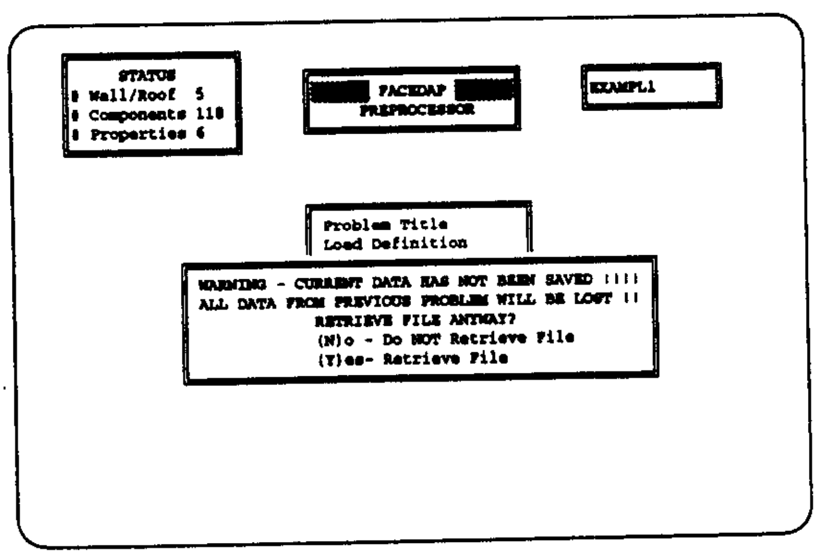
The Preprocessor checks the entered save file name to determine if the file already exists. If the file does exist, the following screen is displayed.



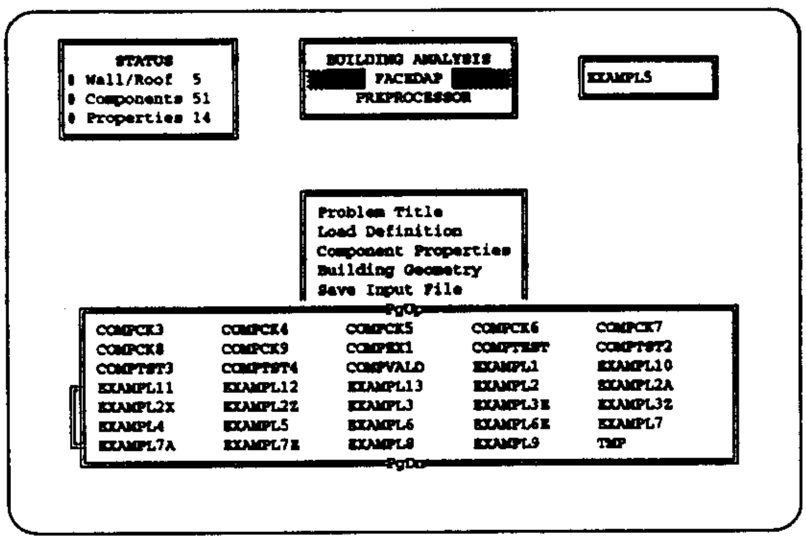
To overwrite the existing file, press S followed by a ↵ or click the left mouse button somewhere on the *Save Current Data* line. The file is then saved. To avoid overwriting the existing file, press "D" followed by a ↵ or click the left mouse button on the *DO NOT Save Current Data line*. In this case, the file will not be saved and control will be returned to the Preprocessor Main Menu. If the user still wishes to save the file under a different name, the *Save Input File* option should then be selected again and another save file name should be input. To quickly abort the save file option, press the Esc key.

8.8 Retrieve Input File

The *Retrieve Input File* option on the main Preprocessor menu allows the user to retrieve a preexisting building file. This option has the same capability as the *Retrieve* option in the main program menu. If this option is selected in the Preprocessor, the following screen is displayed. This screen prevents an unsaved file which has just been edited from being unintentionally overwritten.



Since extensive edit check capabilities are not contained in the Preprocessor, this message may sometimes appear even if no physical changes have been made to the current input file. If (N) is selected or clicked on with the left mouse button, control is returned to the Preprocess Main Menu. If (Y) is pressed or clicked on, a box is displayed which allows the user to select the data file directory path for the file which will be retrieved. *NOTE: this box is not for typing the filename.* The user should accept the displayed data directory path by pressing (↵) or type in any necessary edits to the path and then press (↵). After the (↵) key is selected, a menu is provided for user selection with the names of all files in the input data directory path which have the extension .BLG. The following screen illustrates this menu. **The user must select a file which is compatible with the type of analysis that was selected (i.e., single component analysis or building analysis).** The names of files storing input for single component analysis begin with the four letter string "COMP" and names of files storing input for building analysis can begin with any other four letter string. If the user selects an incompatible file, the program will issue a explanatory message and the incompatible file will not be read into the Preprocessor. Instead, the Preprocessor will read the name of the last compatible input file that was processed from a session file stored on the hard drive by the FACEDAP program and then read this input file into the Preprocessor. If no filename is stored in the session file, the Preprocessor will have blank input screens.



The user should click the left mouse button on the file of interest or use the arrow keys to highlight the file of interest followed by the (↵) key to select the file. Refer to Section 6.3 for more information on using this type of menu.

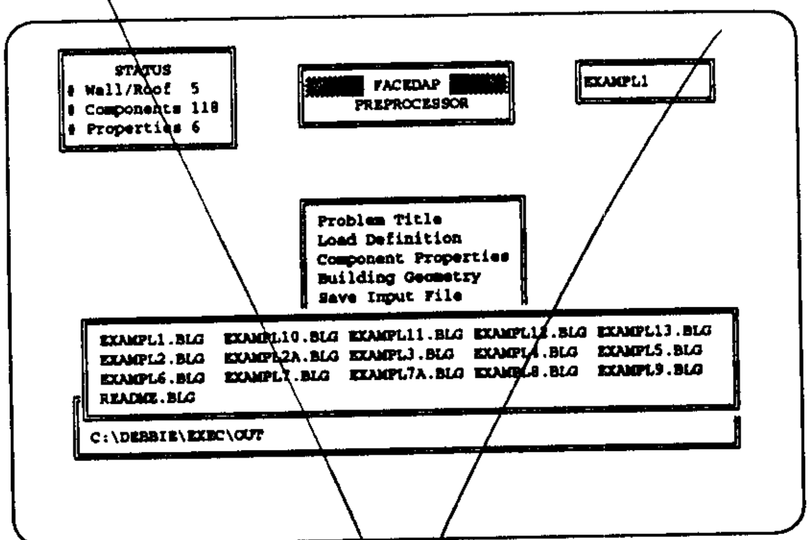
After an input file is selected, the Status box on the screen is updated with the new totals for the selected file and the new filename is displayed in a box on the upper right-hand portion of the screen. Control is also returned to the main Preprocessor menu.

8.9 Clear Forms

The *Clear Forms* option on the main Preprocessor menu provides the capability of clearing all forms and spreadsheets and resetting all default values so that a new building can be entered from "scratch." As with the *Retrieve Input File* option, a safeguard is built in to prevent overwriting data that has not been saved. If the current data has not been saved, a message appears on the screen stating "CURRENT DATA HAS NOT BEEN SAVED." As mentioned above, the Preprocessor does not have extensive edit check capabilities and therefore this message may sometimes appear even if no physical changes have been made to the current input. Any keystroke causes the warning message to disappear and the following screen to be displayed.

THIS PAGE INTENTIONALLY LEFT BLANK

Since extensive edit check capabilities are not contained in the Preprocessor, this message may sometimes appear even if no physical changes have been made to the current input file. If [N] is selected or clicked on with the left mouse button, control is returned to the Preprocess Main Menu. If [Y] is pressed or clicked on, a box is displayed which allows the user to select the data file directory path for the file which will be retrieved. *NOTE: this box is not for typing the filename.* The user should accept the displayed data directory path by pressing [↵] or type in any necessary edits to the path and then press [↵]. After the [↵] key is selected, a menu is provided for user selection with the names of all files in the input data directory path which have the extension .BLG. The following screen illustrates this menu.

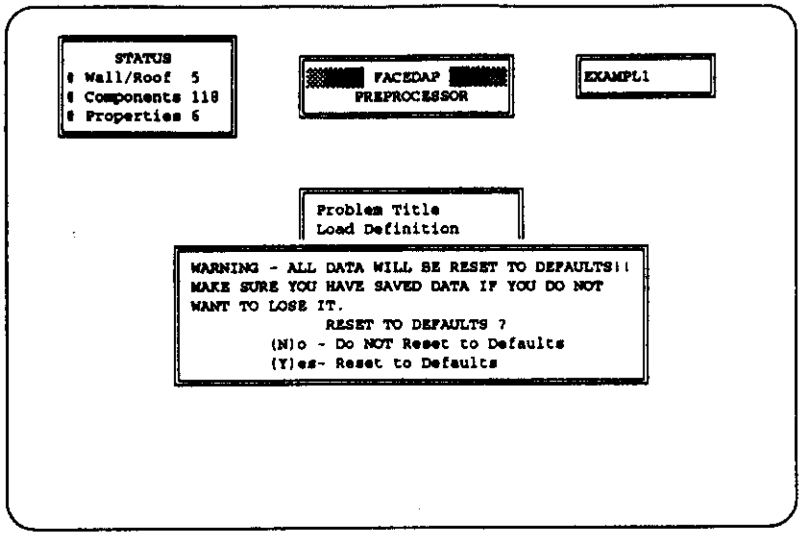


The user should click the left mouse button on the file of interest or use the arrow keys to highlight the file of interest followed by the [↵] key to select the file. Refer to Section 6.3 for more information on using this type of menu.

After an input file is selected, the Status box on the screen is updated with the new totals for the selected file and the new filename is displayed in a box on the upper right-hand portion of the screen. Control is also returned to the main Preprocessor menu.

8.9 Clear Forms

The *Clear Forms* option on the main Preprocessor menu provides the capability of clearing all forms and spreadsheets and resetting all default values so that a new building can be entered from "scratch". As with the *Retrieve Input File* option, a safeguard is built in to prevent overwriting data that has not been saved. If the current data has not been saved, a message appears on the screen stating "CURRENT DATA HAS NOT BEEN SAVED". As mentioned above, the Preprocessor does not have extensive edit check capabilities and therefore this message may sometimes appear even if no physical changes have been made to the current input. Any keystroke causes the warning message to disappear and the following screen to be displayed.

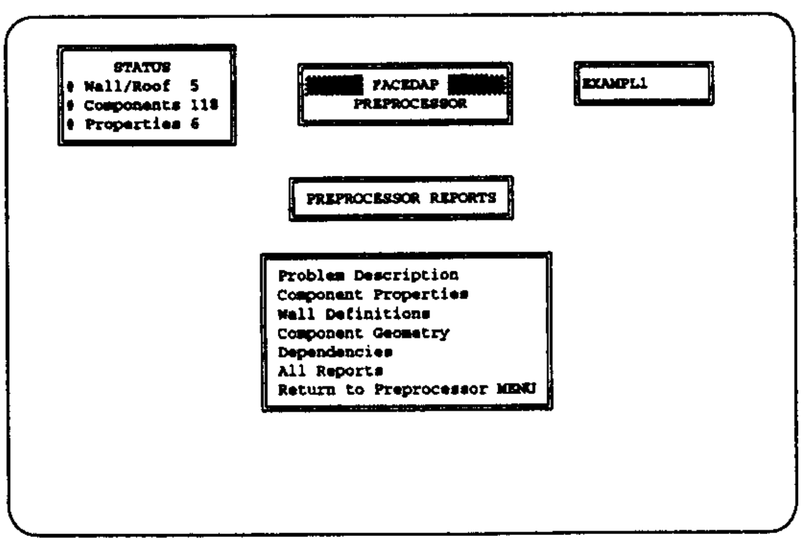


Press Ⓝ or click on the Ⓝ to revert control to the Preprocessor Main Menu without resetting the data. However, if the user presses Ⓨ or clicks on the Ⓨ, all previous data will be lost and all forms and spreadsheets will be reset to blank values or pre-programmed default values.

8.10 Print Reports

The *Print Reports* option on the main Preprocessor menu provides a means for obtaining a "hard copy" of all the information input into the Preprocessor. The printed information is initially written to a file, which has the root name of the most recently saved input file or the input file most recently read into the Preprocessor, and the extension .REP except as noted below. This file is sent to the printer when the user *exits* the Preprocessor and returns to the main FACEDAP program menu. Section 13.0 contains a sample Preprocessor print report for the worked example problem in this section.

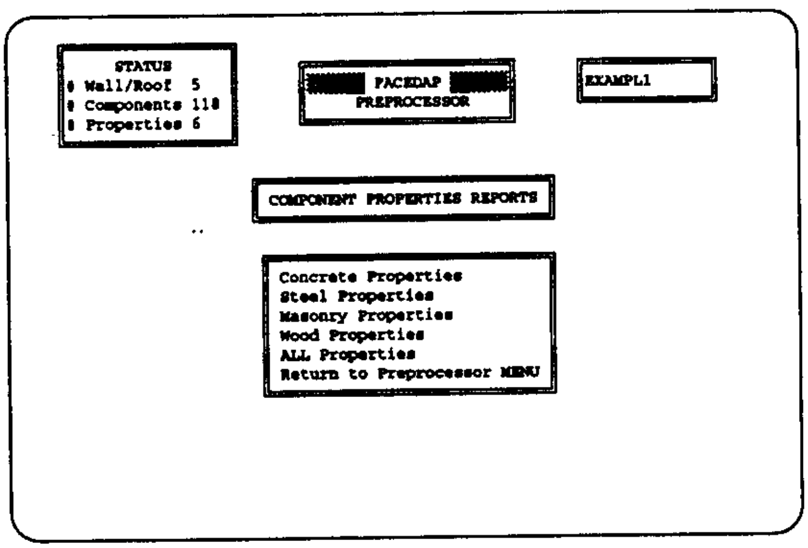
When *Print Reports* option is selected, the following menu is displayed.



This menu gives the user the option of printing selected groups of input information, which are called reports, or printing all input information at once. The user can generate reports showing all Preprocessor input by selecting the *All Reports* option on the Print Menu. Portions of the Preprocessor input can be printed using the other options on the Print Menu as described in the following paragraphs. Multiple reports can be generated by choosing the desired options sequentially without leaving the Print Reports section of the Preprocessor. After the user selects a print option, a message appears indicating the report is being generated. After the report has been generated, a beep sounds and the message "Press any key to continue" is displayed at the bottom of the screen. This indicates the selected report has been written to the *.REP file and the user can select another report, which will also be written to this file. The user can select print options from the menu in this manner until all the desired prints have been generated. **As mentioned, above the file is not sent to the printer until the user returns to the main FACEDAP program menu.**

The *Problem Description* option on the Print Reports menu prints the title and problem description input into the *Problem Title* and *Load Definition* options in the main Preprocessor menu. This information includes the problem title and description and the charge weight and location.

The *Component Properties* Report option provides the following submenu with a list of the four material types and a *ALL Reports* option.



If one of the four materials is selected, another submenu to be displayed which lists all the component types constructed from the selected material type. These are the same two submenus which are displayed by the Component Properties option in the main Preprocessor menu. The user should select each Component Type which will be printed one-by-one. A report is generated which contains the input for each unique property set which was defined for each selected Component Type. If the *ALL Report* option is chosen from the first submenu, reports will be generated for all Component Types which have one or more property sets defined. Most of these reports require 2 pages. The pages of each report are numbered separately as Report 1 of 1 or Report 1 of 2, etc., depending on how many pages are in the report.

The *Wall Definitions* option prints all the wall and roof areas input into the Wall/Roof Area Definitions spreadsheet. This report requires two pages. The pages are numbered as discussed above for the *Component Properties* option.

The *Component Geometry* and *Dependencies* options display a menu with the names of all wall/roof areas and the "Frame" option. This menu is the same menu displayed by the *Component Geometry* and *Dependencies* options within the *Building Geometry* option in the main Preprocessor menu. The geometry or dependencies for all the building components in each selected wall/roof area are printed.

As mentioned above, each generated print report is written to a *.REP file. Usually, the root name of this report file is the name of the most recently saved file or the file most recently read into the Preprocessor. If the report is generated before a saved filename has been established, the file is assigned the default name of REPORT.REP. The following message is displayed indicating that this has occurred. It is recommended the user save a file prior to printing it so that the report name will be consistent with the root name which stores the input information.

THE CURRENT BUILDING FILE HAS NOT BEEN NAMED.
REPORT FILE ASSIGNED DEFAULT NAME "REPORT".

A list all files which store print reports (*.REP files) generated during a Preprocessor session is maintained in the file REPORT.STS. For example, if the user retrieved EXAMPL1.BLG and printed all reports, REPORT.STS would contain the name EXAMPL1.REP. If the user then retrieved EXAMPL2.BLG and printed the Component Properties Report, the file name EXAMPL2.REP would be added to REPORT.STS. When control is then returned to the FACEDAP driver, (when the user returns to the main FACEDAP program menu), the driver program prints all files found in the REPORT.STS file.

The printer routes the file to the printer specified as "Txt Printer 1" in the *Configuration* option off *Utilities* in the main FACEDAP program menu. *The file can only be printed if a printer has been selected with the Configuration program and the corresponding port has been selected.* If the printer had not been previously selected a message appears at the bottom of the screen stating the No Printer has been selected. If this occurs, the user should proceed to the *Utilities* Module, select *Configuration Program*, and then define the printer and port for *Txt Printer 1*. If either the printer and the corresponding communication port are not correctly chosen, the FACEDAP program may become hung. In this case, control can be regained by pressing the "CTRL" key and "C" simultaneously or by rebooting the PC.

8.11 Return to Main Menu

The *Return to Main Menu* option on the main Preprocessor menu transfers control back to the FACEDAP program main menu. When this option is selected, a warning is issued if the current building has not been saved. The format of the warning message is the same as in the *Retrieve Input File* option in Section 8.8. [N] should be selected if the user wants to return to the FACEDAP Main Menu without saving the file. [Y] causes the *Save Input File* procedure described in Section 8.7 to be invoked. After the file has been saved, the user is returned to the Preprocessor Main Menu with the *Save Input File* option highlighted. The user should then choose the *Return to Main Menu* option again and control will be transferred to the FACEDAP Main Menu.

A warning message reminding the user to validate the input file is displayed prior to transfer of control to the FACEDAP Main Menu. This display is cleared by pressing a key. Upon returning to the FACEDAP Main Menu, the user should validate the input file by selecting the *Input Validation* option under the *Preprocess* option.

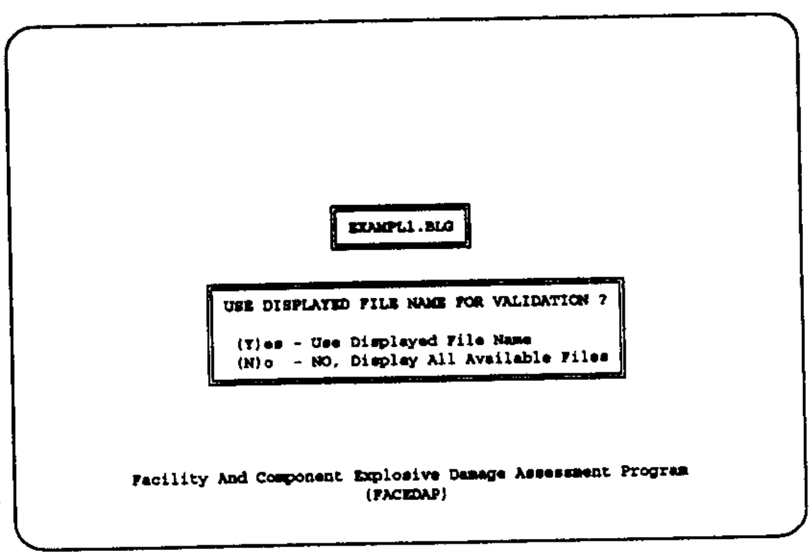
9.0 INPUT VALIDATION

The Validation program, VALIDFIL, checks a Preprocessor input file for errors, which are classified as either warnings or fatal errors. If no fatal errors occur, a file is "stamped" as being "validated" and can be run by the *Analysis* module. If VALIDFIL finds any fatal errors the input file is stamped as being "not validated" and cannot be run by the *Analysis* program until the errors are corrected in the Preprocessor and the file has been validated.

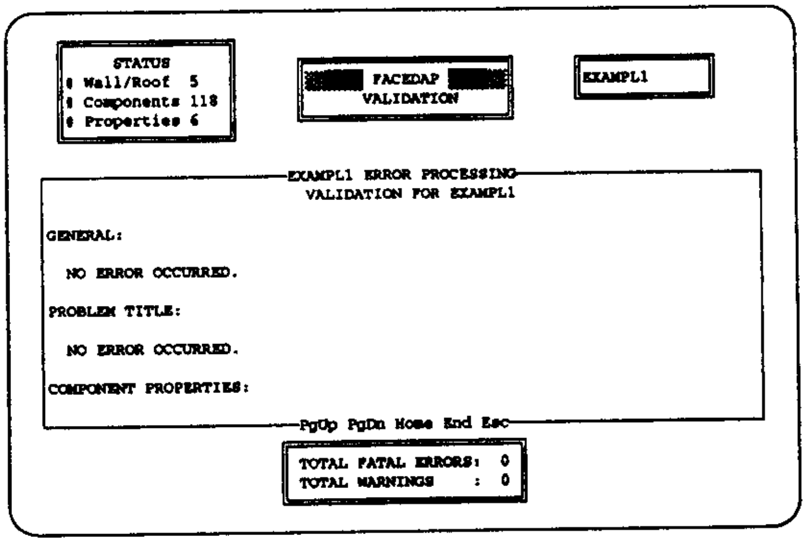
The error checking is divided into several categories. These categories are designed to aid the user in diagnosing the cause of the error. For the most part, the divisions correspond to the options in the main Preprocessor menu and in the Building Geometry menu. Errors are output in terms of the following categories: General, Problem Title, Component Properties, Wall Definitions, Load Definition, Wall Component Definitions, and Wall Dependency Definitions. Refer to Section 12.3 for a listing and explanation of the error messages output by the *Validation* module.

9.1 Entering the Validation Module

The user validates an input file by selecting the *Validate Input* option under the *Preprocess* option in the main FACEDAP program menu. If no input file has been retrieved or saved during the current session, a menu of all the input files (*.BLG files) in the default data directory appears for user selection. If an input file was retrieved or the Input Definition option under the *Preprocess* option has been used to read in or save an input file, a dialog message displaying the save file name of the input file appears on the screen. This message is shown below.

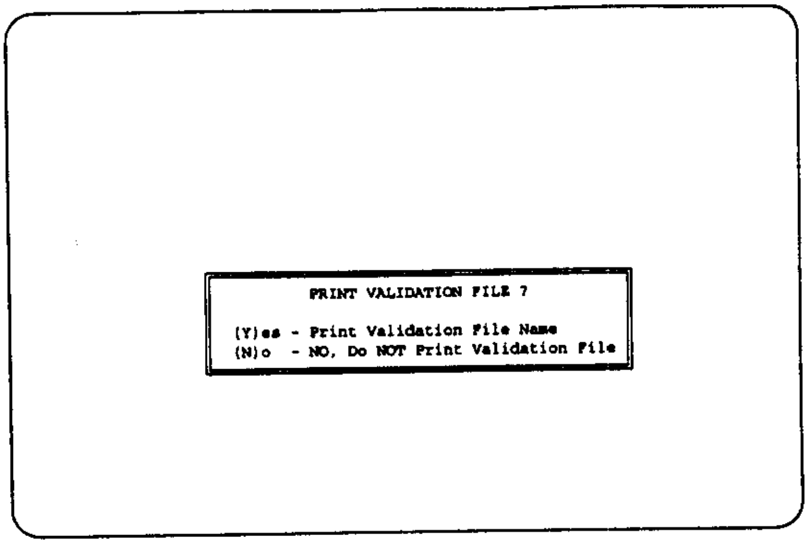


If the user selects "Y", the Validation program is invoked. If the user selects "N", a menu is displayed with all the available saved input files in the default data directory. After the correct input file is selected, the VALIDFIL module is executed. While the file is being checked for errors, a message is shown on the screen stating, "VALIDATING DATA FILE". After the program has checked the selected file for errors, a review window appears on the screen which allows the user to review any errors which were found. A sample of this window follows.



The user can page through the review window using the (PgUp) and (PgDn) keys or by clicking on the words "PgUp" or "PgDn" below the window with a mouse. The errors are grouped in the categories discussed above. A summary of the total warnings and fatal errors appears at the bottom of the window. After the error messages have been reviewed, press (Esc) or click on the word "Esc" to exit this screen.

The *Validation* module automatically writes the error report shown in the review window to a file in the default data directory having the same name as the input file and the extension .ERR. After the user exits the review window, a dialog message automatically appears on the screen which asks the user if the file which contains all the error messages displayed in the review window should be printed. The following screen shows this message.



If the user selected "Y", the *.ERR file is sent to the printer specified as *Txt Printer 1* in the *Configuration* option of the *Utilities* Module and in the FACECFG program. The user must make sure that a printer and port have been selected. If the printer had not been previously selected a message appears at the bottom of the screen stating the No Printer has been selected. If this occurs, the user should proceed to the *Utilities* Module, select *Configuration Program*, and then define the printer and port for *Txt Printer 1*. If either the printer and the corresponding communication port are not correctly chosen, the FACEDAP program may become hung. In this case, control can be regained by pressing the "CTRL" key and "C" simultaneously or by rebooting the PC. If the user selects "N" for the above screen, the *.ERR file will not be sent to the printer. In either case, the user will be returned to the main FACEDAP program menu.

10.0 ANALYSIS

The user initiates the Analysis module by selecting the *Begin Analysis* option from the main FACEDAP program menu. After this option is selected, the FACEDAP program checks for the most recent input file was retrieved or saved during the current session and reads this input file. If no such file exists, a menu of all input files (*.BLG files) is displayed on the screen for user selection. The selected file is checked for a "validation stamp". If the file is not validated, a message appears stating the file has not been validated and the *Analysis* module is aborted. If the selected input file has been validated, an intermediate executable, MAKEBDMA.EXE, is automatically run by the FACEDAP driver. This executable converts the input data into the format required by the BDAMA.EXE executable and writes the reformatted data to the BDAMA.IN file in the default data directory. Next, the FACEDAP driver program automatically initiates the BDAMA.EXE executable which calculates the building blast damage. Before this executable terminates, it writes an output file with the root name of the input file and the extension .PST. After the *Analysis* module has run, the user is returned to the main FACEDAP program menu he/she should select the *Postprocess* option to view the calculated blast damage.

The *Analysis* module of the program calculates the building blast damage using a multi-step procedure. The blast load on each component in the building is calculated in the first step. The blast load on the component is characterized in terms of the peak blast pressure and the blast impulse. The blast impulse is the integral of the blast pressure history, or the area under the blast pressure vs. time curve. Next, the component blast damage is determined by: 1) calculating special non-dimensional terms which relate the calculated blast load to the input component geometry and material property parameters and then; 2) comparing these non-dimensional terms to limit values which define the four component damage levels considered by FACEDAP program. The four component damage levels, which are discussed in more detail in Section 11.1.1, are 0% damage, 30% damage, 60% damage, and 100% damage. Two non-dimensional terms are calculated for each component which separately compare the peak applied blast pressure and the impulse on the component to input component geometry and material property parameters. These two non-dimensional values are called "Pbar" and "Ibar" respectively. Damage levels are determined based on comparisons of both the Pbar and Ibar terms to limit values that define the different damage levels. The limit values for Pbar and Ibar can be plotted as curves on a graph which has Pbar values plotted on one axis and Ibar plotted on the other axis (a P-i diagram), and these "damage curves" will divide the graph into separate regions where the component damage is equal to one of the four damage levels. The damage curves for each component type are programmed into the *Analysis* module. These limit values vary with the component type. After the damage level has been calculated for all components, the component dependencies are considered and component damage is modified to account for cascading failure where necessary.

After all component damage levels have been calculated, the damage level of each component is converted to decimal form (i.e., the 30% damage level is considered as 0.3) and multiplied by the user defined component weighting factor. This product is the weighted component damage level. The weighting factor is explained in more detail in Section 8.5.2. In the final step of the building damage calculation procedure, the weighted damage levels of all components in the building are summed and this sum is divided by the sum of all the weighting factors. This ratio is multiplied by 100 to get the percentage of building blast damage. A building replacement factor and a building reusability factor, which are explained in the next section, are calculated in a similar manner. Reference 2 contains a more thorough explanation of the calculations performed in the *Analysis* module.

11.0 POSTPROCESSOR

The FACEDAP Postprocessor provides the user the capability to view and/or print the calculated building blast damage and the calculated damage and blast load on each building component. Building damage is expressed in terms of four parameters; 1) the percentage of building damage, 2) the amount of required component replacement, 3) the percentage of reusable building floor space prior to repair/replacement, and 4) the level of protection provided to building inhabitants from the input explosive threat. These values are calculated according to criteria which is defined in the next section. The calculated blast load and blast damage to building components is organized for display in terms of the wall/roof areas defined in the Preprocessor (see Section 8.6.2 and 8.6.3 for an explanation of these areas). The blast loads and component damage levels for all building components in each wall/roof area are displayed together. The Pbar and Ibar terms calculated for each component, which are discussed in Section 10.0 and in Reference 2, are also displayed.

The *Analysis* module of the program calculates the single component or building blast damage. The blast loads are calculated in the first step in terms of the peak blast pressure and the blast impulse. The blast impulse is the integral of the blast pressure history, or the area under the blast pressure vs. time curve. Next, the component blast damage is determined by: 1) calculating special non-dimensional terms which relate the calculated blast load to the input component geometry and material property parameters and then; 2) comparing these non-dimensional terms to limit values which define the four component damage levels considered by FACEDAP program. The four component damage levels, which are discussed in more detail in Section 11.1.1, are 0% damage, 30% damage, 60% damage, and 100% damage. Two non-dimensional terms are calculated for each component which separately compare the peak applied blast pressure and the impulse on the component to input component geometry and material property parameters. These two non-dimensional values are called "Pbar" and "Ibar" respectively. Damage levels are determined based on comparisons of both the Pbar and Ibar terms to limit values that define the different damage levels. The limit values for Pbar and Ibar can be plotted as curves on a graph which has Pbar values plotted on one axis and Ibar plotted on the other axis (a P-i diagram), and these "damage curves" will divide the graph into separate regions where the component damage is equal to one of the four damage levels. The damage curves for each component type are programmed into the *Analysis* module. These limit values vary with the component type.

For the case of building analyses, the blast load and damage are calculated for each component in the building and then secondary, cascading failure is calculated considering the component damage caused directly by blast loads and the input component dependencies. Then, the building damage level is calculated with a two-step process. First, the calculated damage level of each building component is converted to decimal form (i.e., the 30% damage level is considered as 0.3) and multiplied by the user defined component weighting factor to get weighted component damage levels. The weighting factor is explained in more detail in Section 8.5.2. Then, the weighted damage levels of all building components are summed and this sum is divided by the weighted damage representing 100% damage to all the components. This ratio is multiplied by 100 to get the percentage of building blast damage. A building replacement factor and a building reusability factor, which are explained in the next section, are calculated in a similar manner. Reference 2 contains a more thorough explanation of the calculations performed in the *Analysis* module.

11.0 POSTPROCESSOR

The FACEDAP Postprocessor provides the user the capability to view and/or print the calculated single component or building blast damage. For the case of building analyses, the overall building damage is expressed in terms of four parameters; 1) the percentage of building damage, 2) the amount of required component replacement, 3) the percentage of reusable building floor space prior to repair/replacement, and 4) the level of protection provided to building inhabitants and assets from the input explosive threat. These values are calculated according to criteria which is defined in the next section. The blast loads, damage levels, and Pbar and Ibar values calculated for each component in buildings are also displayed. The Pbar, Ibar, and blast loads calculated by the Analysis Module during the blast damage analysis procedure are discussed in Section 10.0 and in Reference 2. This output is organized for display in terms of the wall/roof areas defined in the

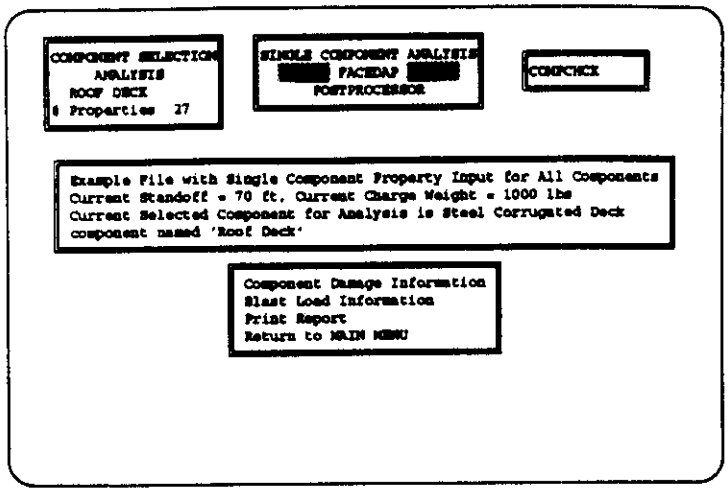
Preprocessor (see Section 8.6.2 and 8.6.3 for an explanation of these areas). The blast loads, Pbar and Ibar values, and component level of protection calculated in a single component analysis are also displayed by the Postprocessor.

The user initiates the Postprocessor module by selecting the *Postprocess* option from the FACEDAP Main Menu. After this option is selected, the FACEDAP program reads the input file which was most recently retrieved or analyzed in the *Analysis* module. If no such file exists, a menu of all input files (*.BLG files) is displayed on the screen for user selection. **The file selected for viewing by the Postprocessor should have been validated and subsequently processed by the Analysis module.** Files beginning with the string "COMP" store information related to single component analyses and files beginning with any other string store information related to building analyses. Regardless of the type of file selected, the Postprocessor will check that: 1) the file selected for postprocessing has been validated by the *Input Validation* option of the *Preprocess* module; 2) an output file for the selected file (with the same root name and the extension .PST) has been created by the *Analysis* module, and 3) the number of components in the input file for the selected filename matches that in the *Analysis* module output file with the selected root filename. If these checks indicate that the selected file is not current or has not been run through the analysis module, an explanatory message will be displayed. There are some circumstances where these checks alone will not ensure that a file read into in the *Postprocessor* module is consistent with the most recent information input into the *.BLG file. In these cases a series of programmer error messages may be written to the screen. If this occurs, the user should exit the Postprocessor and reanalyze the selected file in the *Analysis* module. It is good practice to always analyze a file immediately before viewing the results.

Depending on the type of analysis that has been performed, or that has been stored in the selected input file, the Postprocessor will display single component output or building output. These two types of output are discussed separately in the following sections. In some cases, the description of the Single Component output refers to sections of the manual describing Building Analysis output because the Single Component option was not originally part of the FACEDAP program. As the user will probably notice, the flow of the manual favors the Building Analysis option for this reason.

11.0.1 Single Component Analysis Postprocessor

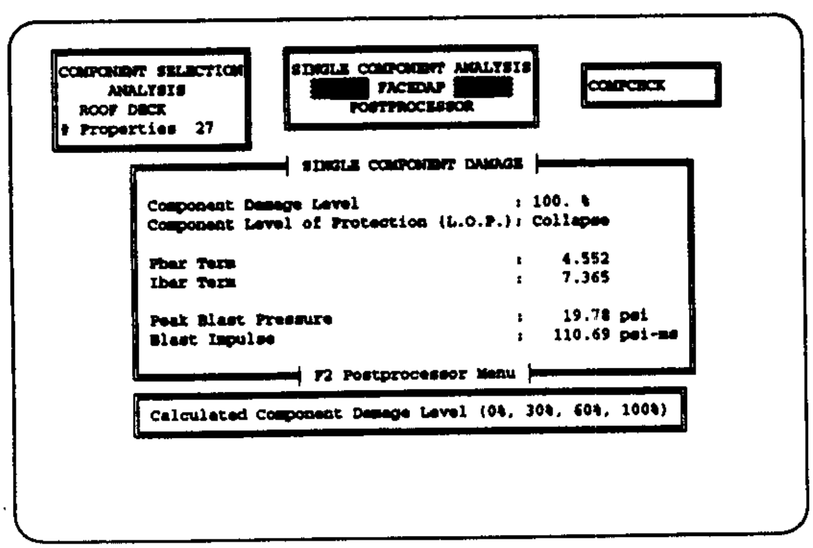
When the *Postprocessor* option is used to view output from a single component analysis, the following Single Component Postprocessor Main Menu is displayed:



The user selects one of the four available options using one of the following methods; 1) pressing the first letter of the desired menu item to highlight the item and then ⏎ to select, 2) using the ↑ and ↓ keys to highlight the desired item and then ⏎ to select, 3) clicking the left mouse button on the desired row, and then ⏎ to select, or 4) clicking the right mouse button to highlight the desired item and then ⏎ to select.

11.0.1.1 Component Damage Information

The *Component Damage Information* option on the main Postprocessor menu invokes the following, read-only, form which displays the following six output parameters: 1) the percentage of component damage; 2) the component level of protection; 3) pbar; 4) ibar; 5) peak blast pressure and 6) the blast impulse. These parameters are explained in the following sections. The Component Damage form appears as follows:



The percentage of component damage is calculated in the *Analysis* module as either 0% damage, 30% damage, 60% damage, and 100% damage. These levels are qualitatively defined as follows.

0% damage	-	Slight damage
30% Damage	-	Moderate damage
60% Damage	-	Severe damage
100% Damage	-	Very severe damage or collapse

These four damage levels are defined in terms of ductility ratios, μ, and ratios of maximum deflection to span length (w/l) for each component type in Table 18 in Section 11.1.1. These approximate definitions, which are only presented to give the user a general idea of the amount of component response associated with each damage level, are discussed in more detail in Section 11.1.1.

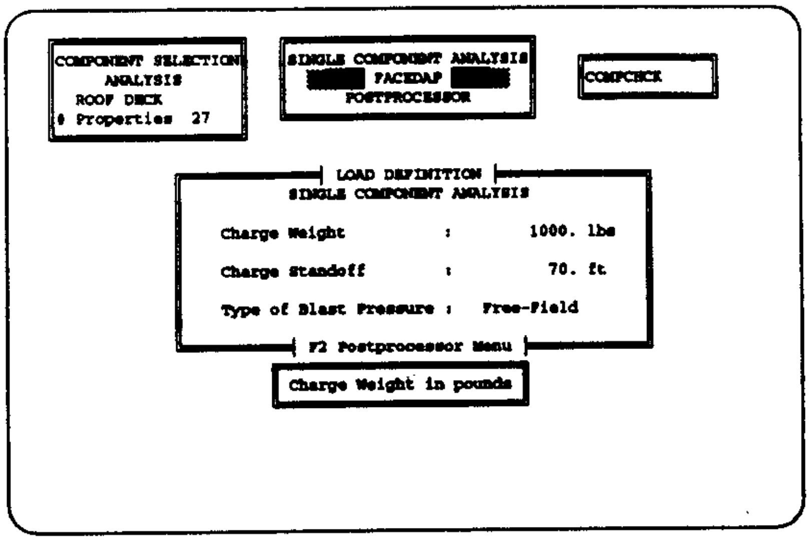
The component level of protection is calculated based on the percentage of building damage and the three levels of protection used by the U.S. Army Corp of Engineers to characterize the protection provided to personnel and equipment by a structural component during response to blast loading. The levels of protection are associated with component damage levels as follows: 1) the 0% damage level is similar to the High Level of Protection; 2) the 30% damage level is similar to the Medium Level of Protection; 3) the 60% damage level is similar to the Low Level of Protection; and 4) the 100% damage level is assumed equivalent to Collapse. However, components with 100% damage that are near the threshold between 60% and 100% damage will most probably not be collapsed in terms of the general usage of the word. Section 4.1 of the Theory Manual[2] discusses component levels of protection in more detail.

Ibar and pbar terms are parameters calculated in the *Analysis Module* in order to determine the component damage level. They are output so that the user can plot component damage on the appropriate P-i diagram (depending on the component type) in the Theory Manual and estimate the component damage more accurately. These terms are discussed briefly in Sections 1.0 and 10.0 of this manual and they are thoroughly discussed in Section 4.0 of the Theory Manual.

The blast load calculated at the center of each component is displayed in terms of the peak blast pressure and the blast impulse. The blast impulse is the integral of the blast pressure history, or the area under the blast pressure vs. time curve. A right triangular shape is assumed for the pressure history, where the applied blast pressure immediately rises to its peak value and then linearly decays to zero. The peak blast overpressure and impulse are calculated using the procedure in Reference 6 for a surface burst explosion assuming the component is exposed to the input type of blast pressure (free-field or fully reflected blast pressure). Blast pressures are described in more detail in Section 3.0 of the Theory Manual[2].

11.0.1.2 Blast Load Information

If the user selects *Blast Load Information* from the main *Single Component Postprocessor Main Menu,* the following read-only screen is displayed showing the charge weight, standoff and blast pressure type input in the Preprocessor. This option is available so that the user can check the input blast load information.

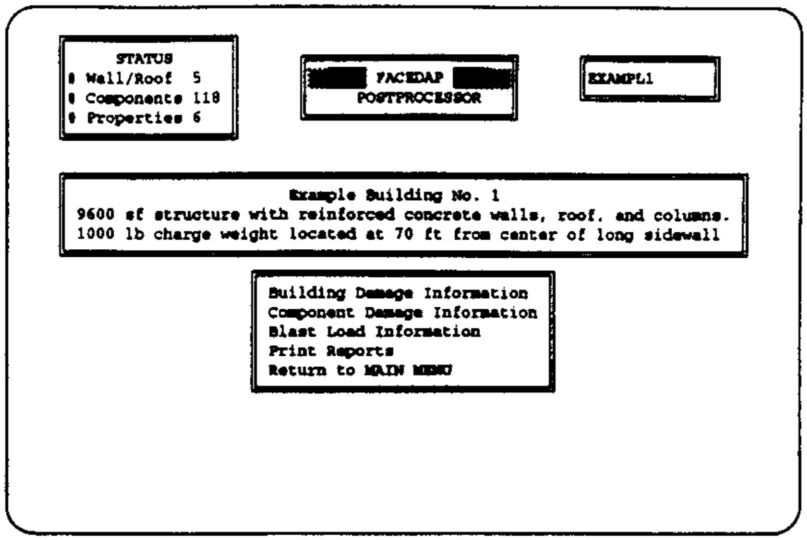


11.0.1.3 Print Report

The *Print Report* option on the Single Component Postprocessor *Main Menu* provides a means for obtaining a hard copy of the component blast damage information and the input blast load information discussed in the two preceding subsections. This option is similar enough to the *Print Report* option in the Building Analysis portion of the Postprocessor so that the user is referred to Section 11.4 for a description.

11.0.2 Building Analysis Postprocessor

When the *Postprocessor* option is used to view output from a building analysis, the following menu is displayed:

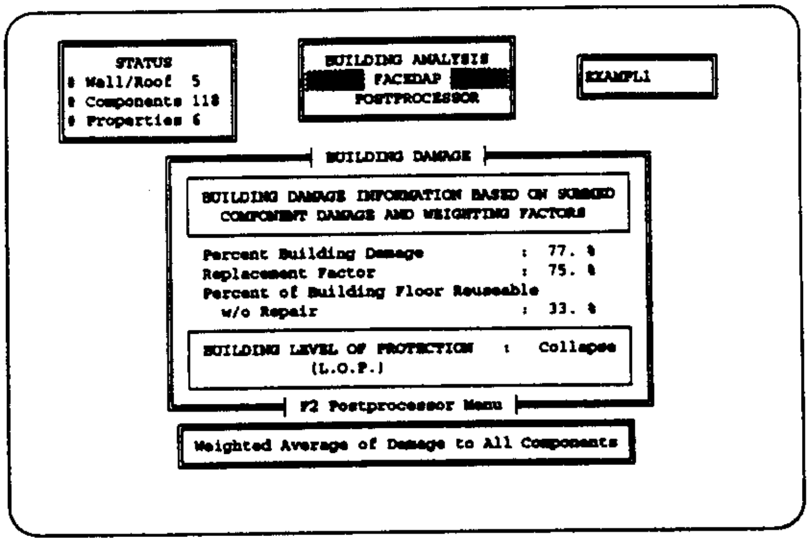


The user selects one of the five available options using one of the following methods; 1) pressing the first letter of the desired menu item to highlight the item and then ↵ to select, 2) using the ↑ and ↓ keys to highlight the desired item and then ↵ to select, 3) clicking the left mouse button on the desired row, or 4) clicking the right mouse button to highlight the desired item and then ↵ to select.

In some cases, the user may see a message prior to the screen shown above which states that Frame Dependencies are being considered. If a building contains frames and the user has allowed the FACEDAP program to calculate frame dependencies, these dependencies are considered immediately by the Postprocessor. These dependencies are discussed in Section 8.6.4.2. If any cascading failure of components is calculated in the Postprocessor due to Frame Dependencies, all building damage parameters are updated to include the increased damage.

11.1 Building Damage Information

The *Building Damage Information* option on the main Postprocessor menu invokes the following, read-only, form which displays the following four output parameters which define the amount of building blast damage: 1) the percentage of building damage; 2) the replacement factor; 3) the percentage of building floor space reusable without repair, and 4) the building level of protection. These parameters are explained below.



Return to the Postprocessor is accomplished by pressing F2 or clicking the mouse within the *F2 Postprocessor Menu* area of the form.

11.1.1 Percentage of Building Damage

The percentage of building damage is a weighted average of the damage to all components in the building. The percentage of building damage is calculated in the *Analysis* module as follows after the blast damage has been calculated for each component. First, the damage level of each component in decimal form (i.e., the 30% damage level is considered as 0.3) is multiplied by the user defined component weighting factor. This product is the weighted component damage level. The weighting factor is explained in more detail in Section 8.5.2. Secondly, the weighted damage levels of all components in the building are summed. Then, finally, the percentage of building damage is calculated as the ratio of the summed weighted damage of all components divided by the sum of all the weighting factors. This ratio is multiplied by 100 to get the percentage of building damage.

As discussed in the paragraph above, the calculated percentage of building damage is controlled by the calculated component damage. Four component damage levels are calculated by the FACEDAP program, which are designated 0% damage, 30% damage, 60% damage, and 100% damage. These levels are qualitatively defined as follows.

0% damage - Slight damage

30% Damage - Moderate damage

60% Damage - Severe damage

100% Damage - Very severe damage or collapse

The four damage levels listed above have been correlated with the levels of protection used by the U.S. Army Corp of Engineers as follows[9]: 1) the 0% damage level is similar to the High Level of Protection; 2) the 30% damage level is similar to the Medium Level of Protection; 3) the 60% damage level is similar to the Low Level of Protection; and 4) the 100% damage level is similar to Collapse. These four damage categories have also been defined quantitatively for each component type in terms of the amount of component deflection caused by the blast loading. The four damage levels for each component type are defined in Table 18 in terms of both ductility ratio, μ, and the ratio of maximum deflection to span length (w/l). These definitions are approximate and they are only presented to give the user a general idea of the amount of component response which is associated with each damage level for each component type. Component response which is less than that shown for the lower limit of 30% damage in the table is considered 0% damage. The ductility ratio is the ratio of the maximum component deflection divided by the yield deflection. The yield deflection is the deflection causing the outer fiber (with maximum strain) of the component to begin yielding. If the ductility ratio and w/l ratio correspond to different damage levels, the more severe of the damage level controls. The ductility ratios and relative deflections (w/l) which define a given damage level vary from component to component because some components are tougher (less brittle) than others, and therefore they can respond more to blast load without sustaining as much "damage". The development of Table 18 is discussed in Reference 2.

11.1.2 Building Replacement Factor

A limit damage level has been established for each component type which defines the maximum damage that the component can sustain and remain "repairable". A "repair/replace" factor is assigned to each building component in the *Analysis* module based on its repairability. If the calculated component damage level is greater than the limit level for repairability, the component is considered unrepairable and a repair/replace factor of 1 is assigned to the component. Otherwise, the component is considered repairable and a repair/replace factor of 0 is assigned. The building replacement factor is the weighted average of the repair/replace factors of all the components in the building. This factor is determined in an analogous manner as the percentage of building damage except that the repair/replace factor of each component (equal to 0 or 1) is considered in the weighted averaging scheme rather than the damage level of the component. This factor is also expressed as a percentage. A high building replacement factor (near 100%) indicates that almost all building components require replacement.

Table 19 shows the correlation between component repairability and damage level which is used by the FACEDAP program to determine the repair/replace factor for the 24 different component types. Note that the R's indicate a repairable component (with a repair/replace factor of 0), while the U's indicate a component requiring replacement (with a repair/replace factor of 1). The correlations shown in Table 19 are subjective. Higher damage levels are generally chosen as the break point between repair and replace for component types which are primary load bearing components because these components are more expensive to replace and therefore they must sustain more damage before it becomes uneconomical to replace them.

Table 18. Quantitative Criteria Defining Damage Levels for Each Component Type

Component Type	Damage Criteria: Lower Limit of 30% Damage		Lower Limit of 60% Damage		Lower Limit of 100% Damage		Notes (See text in Section 4.1 of Theory Manual for more discussion and see general notes)
	μ	w/l	μ	w/l	μ	w/l	
Reinforced Concrete (R/C) Beam	1	.005	5	.022	20	.09	Ductility values assumed same as one-way R/C slab
R/C One-Way Slabs	1	.007	5	.034	20	.135	Ductility values validated w/data
R/C Two-Way Slabs without Arching	1	.015	5	.08	20	.31	Ductility values assumed same as one-way R/C slab
R/C Two-Way Slabs with Arching	1	.005	5	.013	20	.20	Ductility criteria determined from data using approx. theoretical approach, w/l values based directly on data
R/C Exterior Columns (bending)	1	.003	5	.014	20	.054	Ductility values assumed same as one-way R/C slab
R/C Interior Columns (buckling)	--	--	--	--	1	.002	Criteria apply only to impulsive response and are assumed
R/C Frames	1.3	.014	6	.066	12	.133	Ductility values validated w/some data, w/l values are ratio of max. frame sway to column height
Prestressed Beams	.5	.005	1	.01	2	.02	Ductility values are assumed
Steel Beams	2	.012	7	.04	15	.009	Ductility values are based on some data
Metal Stud Walls	2	.02	7	.07	15	.15	Ductility values are assumed same as steel beams
Open Web Steel Joists (based on flexural tensile stress in bottom chord)	1	.01	3.5	.035	6	.06	Ductility values are assumed
Corrugated Metal Deck	2	.012	7	.042	15	.09	Ductility values validated w/some data
Steel Exterior Columns (bending)	2	.009	7	.032	15	.068	Ductility values are assumed same as steel beams
Steel Interior Columns (buckling)	--	--	--	--	1	.0045	Ductility values apply only to impulsive response and are assumed
Steel Frames	1.3	.021	6	.10	12	.20	Ductility values validated w/some data, w/l values are ratio of max. frame sway to column height
One-Way Unreinforced Masonry (unarched)	--	--	--	--	1	.0005	Ductility values are assumed
One-Way Unreinforced Masonry (arched)	.25	.005	.5	.02	1.0	.04	Ductility values determined from data using approx. theoretical approach, w/l values based directly on data

Table 18. Quantitative Criteria Defining Damage Levels for Each Component Type (Continued)

Component Type	Damage Criteria						Notes
	Lower Limit of 30% Damage		Lower Limit of 60% Damage		Lower Limit of 100% Damage		(See text in Section 4.1 of Theory Manual for more discussion and see general notes)
	μ	w/l	μ	w/l	μ	w/l	
Two-Way Unreinforced Masonry (fully arched)	.1	.005	.15	.02	.25	.04	Ductility values determined from data using approx. theoretical approach, w/l values based directly on data
One-Way Reinforced Masonry	1	.0016	5	.008	20	.032	Ductility values assumed same as one-way R/C slab
Two-Way Reinforced Masonry	1	.0016	5	.008	20	.032	Ductility values assumed same as two-way R/C slab
Masonry Pilasters	1	.0006	5	.003	20	.012	Ductility values assumed same as R/C beam
Wood Stud Walls	.5	.01	1	.021	2	.043	Ductility values based on data
Wood Roofs	.5	.01	1	.021	2	.043	Ductility values based on data
Wood Beams	.5	.008	1	.016	2	.032	Ductility values assumed same as wood walls/roof
Wood Exterior Columns (bending)	.5	.01	1	.021	2	.043	Ductility values assumed same as wood walls/roof
Wood Interior Columns (buckling)	--	--	--	--	1	.021	Ductility values apply only to impulsive response and are assumed

General Notes:

1. All w/l values are derived from ductility values using an assumed ratio of deflection to span length for a "typical" component except where indicated otherwise.
2. All values in this table are intended to correlate as well as possible to damage observed in test data and therefore will not always correlate with design criteria
3. The lower limits of 30%, 60%, and 100% damage referred to in this table correspond directly to the upper bounds of High, Medium, and Low Levels of Protection, respectively, as discussed in Section 4.1 of Theory Manual.

Table 19. Correlation Between Component Damage and Assumed Replacement

Component Type	Component Damage Level*			
	0%	30%	60%	100%
R/C Beams	R	R	R	U
R/C One-Way Slabs	R	R	U	U
R/C Two-Way Slabs	R	R	U	U
R/C Exterior Columns (bending)	R	R	R	U
R/C Interior Columns (buckling)	R	N/A	N/A	U
R/C Frames	R	R	R	U
Prestressed Beams	R	U	U	U
Steel Beams	R	U	U	U
Metal Stud Walls	R	U	U	U
Open Web Steel Joists (chord bending failure)	R	U	U	U
Corrugated Metal Deck	R	U	U	U
Steel Exterior Columns (bending)	R	R	U	U
Steel Interior Columns (buckling)	R	N/A	N/A	U
Steel Frames	R	R	R	U
One-Way Unreinforced Masonry	R	R	U	U
Two-Way Unreinforced Masonry	R	R	U	U
One-Way Reinforced Masonry	R	R	U	U
Two-Way Reinforced Masonry	R	R	U	U
Masonry Pilasters	R	R	U	U
Wood Stud Walls	R	R	U	U
Wood Roofs	R	R	U	U
Wood Beams	R	R	U	U
Wood Exterior Columns (bending)	R	R	U	U
Wood Interior Columns (buckling)	R	N/A	N/A	U

**Note:* R = repairable, U = replace

11.1.3 Percent of Building Floor Area Reusable without Repair

In the original development of the methodology used in the *Analysis* module to calculate component and building blast damage, the percentage of building floor space which is reusable without repair was calculated with a graphical procedure. This procedure summed the floor space which was not below or immediately beside a component with 100% damage and then divided by the total floor area to determine the percentage of reusable floor space. A simplified approximation of this procedure is programmed in the FACEDAP *Analysis* module. In the first step, the total number of components with 100% damage is counted and divided by the total number of building components to estimate the percentage of nonreusable floor space. Then, this is subtracted from 100% to estimate the percentage of reusable floor space. This building damage parameter is meant to apply to wartime conditions when only very severe component damage is assumed to affect reusability.

11.1.4 Building Level of Protection

The building level of protection is defined in terms of categories used by the U.S. Army Corps of Engineers to characterize the protection provided to personnel and equipment by building components subjected to blast loading. In the FACEDAP program, the building level of protection is equal to the level of protection provided by the most damaged building component. This is a conservative approach which assumes that the personnel or assets requiring protection are located near the building component with the largest amount of blast damage. The component levels of protection are determined directly from the calculated component damage levels as discussed in Section 11.1.1. The levels of protection are defined in Reference 7 as follows.

Low Level of Protection - unrepairable structural components with a large amount of damage that are not collapsed

Medium Level of Protection - repairable structural components with a significant degree of damage

High Level of Protection - superficially damaged components

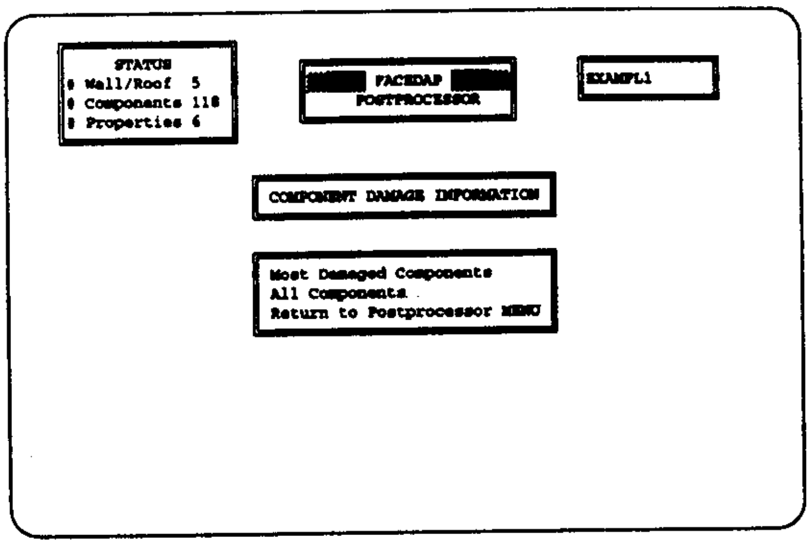
A fourth level is also considered as follows.

Collapse - a collapsed component, or a component near collapse

The calculated building level of protection should only be considered as one indicator of the amount of protection from the input explosive threat which is provided to personnel and equipment in the building. Among other factors, it does not consider the injury/damage caused by failed doors and windows.

11.2 Component Damage Information

When the *Component Damage Information* option is selected from the main Postprocessor menu, the following menu is displayed. The options on this menu allow the user to view information related to the blast damage of each building component.



11.2.1 Most Damaged Components

If the user chooses the *Most Damaged Components* option, a spreadsheet is displayed which contains all components having the maximum damage level. The Postprocessor searches the calculated damage levels of all components and displays only those which have the highest level of damage. A maximum of 150 components are displayed. ***The user must page down to see all of the most damaged components if more than 10 such components are displayed.*** The four component damage levels calculated by the FACEDAP program are discussed in Section 11.1.1. If the calculated damage level for all components in the building is 0%, a message is displayed indicating that there is no significant blast damage. A portion of the *Most Damaged Components* spreadsheet follows.

MOST DAMAGED BUILDING COMPONENTS

Scroll Right to View P-i Diagram Terms

Row	Wall/Roof Area Name	Component Type	Percent Damage (%) Without Cascading	With Cascading	Component Loca x1 (ft)	y1 (ft)
1	ROOF	RC1WI	0	100	10.000	50.000
2	ROOF	RC1WI	0	100	150.000	50.000
3	ROOF	RC1WI	0	100	130.000	50.000
4	ROOF	RC1WI	0	100	110.000	50.000
5	ROOF	RC1WI	0	100	90.000	50.000
6	ROOF	RC1WI	0	100	70.000	50.000
7	ROOF	RC1WI	0	100	50.000	50.000
8	ROOF	RC1WI	0	100	30.000	50.000
9	ROOF	RC1WI	0	100	150.000	30.000
10	ROOF	RC1WI	0	100	130.000	30.000

F1 HELP F2 Menu

Wall Area Where Component Located

Table 20 summarizes the information displayed in the *Most Damaged Components* spreadsheet. All data fields on the spreadsheet are for display purposes only. A help menu which summarized all available keys for traversing the spreadsheet is displayed when F1 is pressed. Only a few of the more useful keys will be mentioned here. The Home key highlights the 1st cell in the current row. The End key highlights the last column of the current row. PgUp and PgDn page up or down a screen at a time. The F2 key is used to exit the spreadsheet and return to the *Component Damage Information* menu. See Section 6.5 for a more complete explanation of keystrokes used in spreadsheet-type screens.

The Wall/Roof Area Name in the first column of the spreadsheet contains the name of the wall/roof area where the component geometry was defined in the Preprocessor unless the component type is a concrete or steel frame. If the component type is a frame component, the name of the blastward wall is displayed in this column. This wall area contains the exterior frame column nearest the explosive charge. An "*" sign precedes the Wall/Roof Area Name when it is a blastward wall name. If the blastward wall is 10 characters long, the last character is truncated so that the "*" can be displayed.

The Percent Damage, Ibar and Pbar values which are displayed in the *Most Damaged Components* spreadsheet were calculated in the *Analysis* module as discussed in Section 10.0 and Reference 2. The percent damage is shown both before and after the effects of any cascading damage are considered. The percent damage without cascading is the component damage level calculated considering only the direct effects of the applied blast pressure. The percent damage with cascading considers the damage caused directly by the blast damage and any secondary damage caused by loss of a support resulting from 100% blast damage to a supporting component. If the component damage level caused directly by the applied blast pressures is equal to 100%, cascading is irrelevant. An explanation of cascading is provided in Section 8.6.4. The Pbar and Ibar terms are output so that the user can plot component damage on the appropriate P-i diagram (depending on the component type) in Reference 2 and estimate the component damage more accurately.

The Repair/Replace factor which is calculated for each component in the *Analysis* module is also displayed. This factor is calculated based on the component type and the calculated percent damage (or damage level) as shown in Table 19. The discussion in Section 11.1.2. explains the Repair/Replace factor.

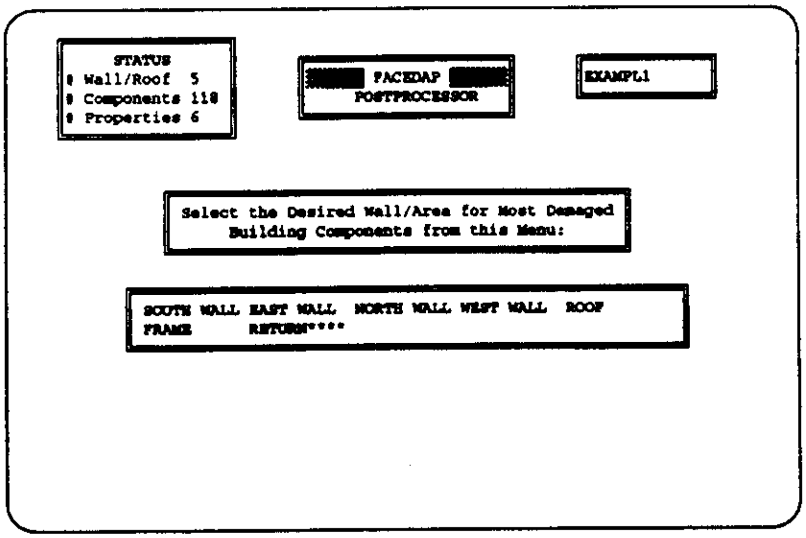
The local coordinates of the component end points or corner points which are displayed in the *Most Damaged Components* spreadsheet are coordinates in the local coordinate system of the Wall/Roof Area shown in the first column, which is the Wall/Roof Area where the component was defined in the Preprocessor (see Section 8.6.2 and 8.6.3). If the component is a frame or interior column, only the coordinates of the first end point are displayed because only one end point is required to define these components in the Preprocessor.

Table 20. Parameters Displayed in the Most Damaged Components Spreadsheet

Column Header	Units	Variable Type	Field Type	Help Message
Wall/Roof Area Name	NA	char	read-only	Wall Area Where Component Located
Component Type	N/A	char	read-only & option	Abbreviated Component Type Name (Press Space Bar for Unabbreviated Name)
Percent Damage	%	int	read-only	Calculated Damage Levels (0%, 30%, 60%, or 100%) with and without any calculated cascading damage
x1	ft	real	read-only	Local x Coordinate of First Component End Point
y1	ft	real	read-only	Local y Coordinate of First Component End Point
x2	ft	real	read-only	Local x Coordinate of Second Component End Point
y2	ft	real	read-only	Local y Coordinate of Second Component End Point
Repair or Replace	N/A	integer	read-only	Repair/Replace Factor for Component
Pbar	N/A	real	read-only	Calculated P-i Diagram Term Along Horizontal Axis
Ibar	N/A	real	read-only	Calculated P-i Diagram Term Along Vertical Axis

11.2.2 Damage to All Components

If the user chooses the *All Components* option in the *Component Damage Information* menu in Section 11.2, a screen is displayed showing all the wall/roof areas defined by the user in the Preprocessor and the FRAME option. This screen is displayed below.



The user selects each wall/roof area which contains the building components of interest. After each wall/roof area is selected, the *Damage Components* spreadsheet is displayed which shows blast damage information for each component in the selected wall/roof area. A portion of this spreadsheet follows. The definition of components in wall/roof areas is explained in Sections 8.6.2 and 8.6.3. When the *FRAME* option is selected, and at least one frame component was defined for the building, a spreadsheet is displayed which shows blast damage for all frame components in the building. When the *RETURN***** option is selected, the user is returned to the *Component Damage Information* menu.

Scroll Right for Additional Parameters

DAMAGED COMPONENTS FOR WALL/ROOF 'SOUTH WALL'

Row	Component Type	Percent Damage (%) Without Cascading	Percent Damage (%) With Cascading	Component Local Coordinate x1 (ft)	y1 (ft)	x2 (ft)
1	RC2WI	0	100	.000	.000	20.000
2	RCBCI	100	100	20.000	.000	20.000
3	RC2WI	30	100	20.000	.000	40.000
4	RCBCI	100	100	40.000	.000	40.000
5	RC2WI	60	100	40.000	.000	60.000
6	RCBCI	100	100	60.000	.000	60.000
7	RC2WI	60	100	60.000	.000	80.000
8	RCBCI	100	100	80.000	.000	80.000
9	RC2WI	60	100	80.000	.000	100.000
10	RCBCI	100	100	100.000	.000	100.000

F1 HELP F2 Menu

Abbreviated Component Type Name (Press Space Bar for Full Name)

The information which is displayed on the preceding spreadsheet is summarized in Table 21. The F2 key is used to exit the spreadsheet and return to the menu of all wall/roof areas. Other keystrokes which can be used in the spreadsheet are explained in Section 6.5.

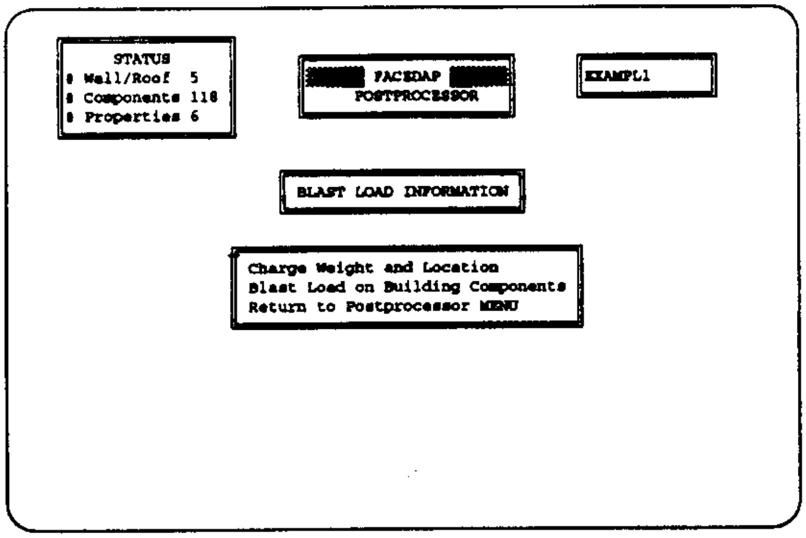
Table 21. Parameters Displayed in the Damaged Components Spreadsheet

Column Header	Units	Variable Type	Field Type	Help Message
Component Type	N/A	char	read-only & option	Abbreviated Component Type Name (Press Space Bar for Unabbreviated Name)
Percent Damage	%	int	read-only	Calculated Damage Levels (0%, 30%, 60%, or 100%) with and without any calculated cascading damage
x1	ft	real	read-only	Local x Coordinate of First Component End Point
y1	ft	real	read-only	Local y Coordinate of First Component End Point
x2	ft	real	read-only	Local x Coordinate of Second Component End Point
y2	ft	real	read-only	Local y Coordinate of Second Component End Point
Repair or Replace	N/A	integer	read-only	Repair/Replace Factor for Component
Pbar	N/A	real	read-only	Calculated P-i Diagram Term Along Horizontal Axis
Ibar	N/A	real	read-only	Calculated P-i Diagram Term Along Vertical Axis

The data fields on this spreadsheet are very similar to those summarize in Table 20. Therefore, they will not be discussed again here.

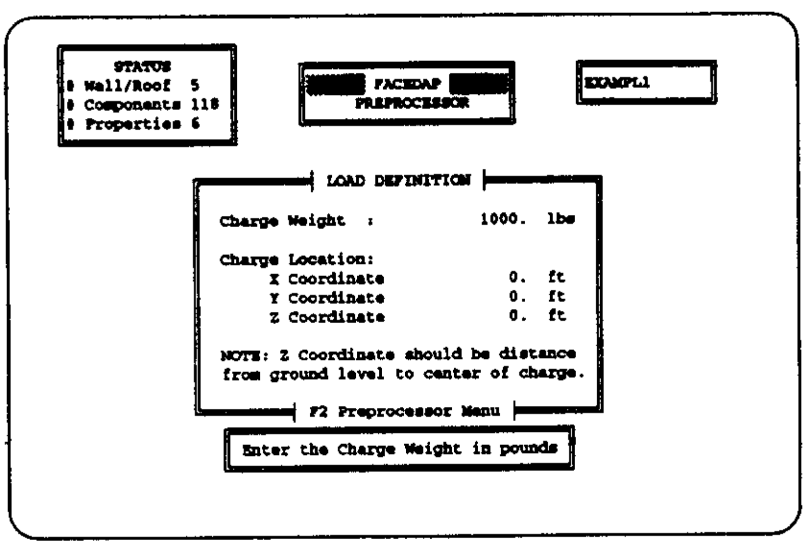
11.3 Blast Loads Information

The selection of the *Blast Loads Information* option from the main Postprocessor menu causes the menu shown below to be displayed. The options on this menu allow the user to view information related to the blast load calculated on each building component.



11.3.1 Charge Information

If the user selects the *Charge Information* option on the preceding menu, the following screen is displayed showing the input charge weight and the global coordinates of the charge location. This is the same charge weight and location information input into the Preprocessor.



11.3.2 Blast Load on All Components

If the user selects *Blast Load on Building Components* option on the *Blast Loads Information* menu, a screen is displayed showing all the wall/roof areas defined by the user in the Preprocessor and the FRAME option. This is similar to the corresponding screen shown in Section 11.2.2. The user should sequentially select that wall/roof component to view the blast loads calculated on components defined in each wall/roof area in the Preprocessor. When each wall/roof area or the FRAME option is selected, a spreadsheet screen is displayed showing blast load information calculated on each component defined in the selected wall/roof area. A portion of this screen follows. Blast load information on frames is shown if the FRAME option is selected and one or more frames were defined in the building input.

BLAST LOAD FOR WALL/ROOF "SOUTH WALL"

Scroll Right for Local Coordinates

Row No.	Component Type	Blast Load Peak Pressure (psi)	Impulse (psi-ms)	End or Opposite Corner Poi in Local Wall Area Coordin x1 (ft)	y1 (ft)	x2 (ft)
1	MAR2WI	43.48	229.3	0.	0.	10.
2	MAR2WI	51.26	246.9	10.	0.	25.
3	MAR2WI	57.63	259.9	25.00	0.	40.00
4	MAR2WI	57.63	259.9	40.00	0.	55.00
5	MAR2WI	51.26	246.9	55.00	0.	70.00
6	MAR2WI	43.48	229.3	70.00	0.	80.00

F1 HELP F2 Menu

Abbreviated Component Type Name (Press Space Bar for Full Name)

The information which is displayed on this spreadsheet is summarized in Table 22. The F2 key is used to exit the spreadsheet and return to the menu of all wall/roof areas. Other keystrokes which can be used in the spreadsheet are explained in Section 6.5.

Table 22. Parameters Displayed in the Blast Load Spreadsheet

Column Header	Units	Variable Type	Field Type	Help Message
Component Type	N/A	char	read-only & option	Abbreviated Component Type Name (Press Space Bar for Unabbreviated Name)
Peak Pressure	psi	real	read-only	Calculated Peak Overpressure at Center of Component
Impulse	psi - ms	real	read-only	Calculated Impulse at Center of Component
Wall Name*	N/A	char	read-only	Blastward Wall Containing Frame Exterior Column Nearest Charge
x1	ft	real	read-only	Local x Coordinate of First Component End Point
y1	ft	real	read-only	Local y Coordinate of First Component End Point
x2**	ft	real	read-only	Local x Coordinate of Second Component End Point
y2**	ft	real	read-only	Local y Coordinate of Second Component End Point

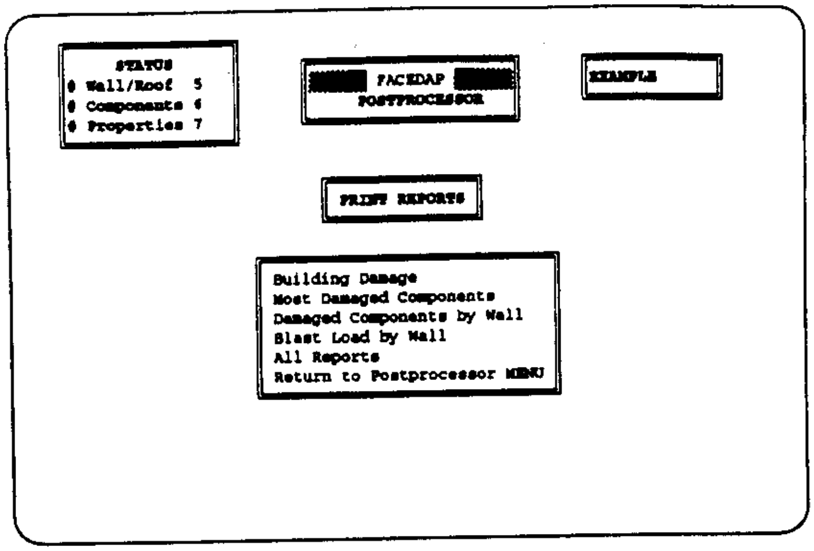
* Used on the Blast Load Information spreadsheet for FRAME components

** Not used on the Blast Load Information spreadsheet for FRAME and INTERIOR COLUMN components

The blast load calculated at the center of each component is defined by the peak blast pressure and the blast impulse. The blast impulse is the integral of the blast pressure history, or the area under the blast pressure vs. time curve. A right triangular shape is assumed for the pressure history, where the applied blast pressure immediately rises to its peak value and then linearly decays to zero. The peak blast overpressure and impulse are calculated using the procedure in Reference 6 for a surface burst explosion. A side-on, or free-field blast pressure is calculated on components if the angle of incidence (the angle between the outward normal from the component and the straight line from the charge to the component) is greater than 45 degrees. Otherwise, a reflected blast pressure is calculated. For the same charge weight and distance between the explosive charge and component, the reflected blast pressure is two to eight times greater than side-on blast pressure. Reference 2 contains a detailed discussion of the procedure used in the FACEDAP program to calculate the blast load on building components.

11.4 Print Reports

The *Print Reports* option on the main Postprocessor menu provides a means for obtaining a "hard copy" of the information displayed in the Postprocessor. The selected output is written in form of reports to a file which has the same filename as the file which is read into the Postprocessor and an extension of .REP. Each report contains a given type of output, such as blast loads or blast damage to the most damage components. The user has the choice of printing selected reports or generating all reports at once. A screen showing the Print Reports menu follows. A sample Postprocessor print report for the worked example problem is included in Section 13.0.



Each report of interest is highlighted with the arrow keys or by pressing the first letter of the desired report and selected with ⏎ key, or is selected by clicking the left mouse button on the desired report. A message then appears indicating the report is being generated. A beep follows with another message at the bottom of the screen stating "Press any key to continue". This indicates the current report has been written to the *.REP file and the user should press any key to return to the *Print Reports* menu. After generating all desired reports, the user should select the *Return to Postprocessor Menu* to return to the main Postprocessor menu.

The *Building Damage* option on the *Print Reports* menu writes a report with the problem title and description, the charge weight and location, and the four calculated building damage parameters. The *Most Damaged Components* option writes a report with all the information displayed on the Most Damaged Spreadsheet. *Damaged Components by Wall* option and *Blast Load by Wall* option on the *Print Reports* menu both invoke a secondary menu with the names of all the wall/roof areas and "Frame". When the name of each wall/roof area (or the "Frame" option) is selected in this submenu, a report is generated with the calculated component damage information or the calculated blast load information for all the building components in the selected area. All the information displayed in the Postprocessor can be printed at once by selecting the *All Reports* option in the *Print Reports* menu.

The file with all the selected reports is not printed immediately. It is printed when the user exits from the Postprocessor and returns to the main FACEDAP menu. The printer routes the file to the printer specified in the *Configuration* option off *Utilities* in the main FACEDAP menu as Txt Printer 1. The file can only be printed if a printer has been selected with the Configuration program and the corresponding port has been selected. If the printer had not been previously selected a message appears at the bottom of the screen stating the No Printer has been selected. If this occurs, the user should proceed to the *Utilities* Module, select *Configuration Program*, and then define the printer and port for *Txt Printer 1*. If either the printer and the corresponding communication port are not correctly chosen, the FACEDAP program may become hung. In this case, control can be regained by pressing the "CTRL" key and "C" simultaneously or by rebooting the PC.

12.0 ERROR MESSAGES

All error and warning messages output by the FACEDAP program are listed and explained in this section. In some cases the message is just listed because it is considered self-explanatory. The error messages for each module of the program are shown in the separate subsections 12.1 through 12.5. Section 12.1 contains error messages generated by the *FACEDAP driver program*, Section 12.2 contains error messages generated by the *Preprocessor*, Section 12.3 contains error messages generated by the *Validation* module, Section 12.4 contains error messages generated by the *Analysis* module, Section 12.5 contains error messages generated by the *Postprocessor*. Sections 12.2, 12.3 and 12.5 are split into two subsections so that error messages related to single component analyses are shown separately from those related to building component analyses. The listings within each section are arranged alphabetically according to the first letter in the first word of the error/warning message excluding the word WARNING or FATAL, which often begin error/warning messages. When the first word in the error message is a variable such a wall/roof area name, a column or row number, etc., the error message is listed at the end of the section. In some cases, the user may need to refer to other sections of the User's Manual for a more complete discussion of input variables which are in error.

12.1 Error Messages in the FACEDAP Driver

ERROR MESSAGE	ERROR TYPE
FACE CONFIGURATION FILE NOT FOUND. RUN FACECFG.	**FATAL**
The FACE.CFG file is not in the program directory path. Exit to main menu and use configuration option to check path and check that the FACE.CFG file is in intended area on hard disk. The FACECFG program may not have been run to create this file as explained in Section 3.0.	
MAINTITL.DAT FILE NOT FOUND SHOULD BE IN PROGRAM DIRECTORY	**WARNING**
MAINTITL.DAT was not found in the program directory path. Exit to Main Menu and use Configuration option under Utilities to check path and check that the MAINTITL.DAT file is in the intended area on the hard disk.	
SELECTED FILE HAS NOT BEEN VALIDATED OR FAILED TO PASS VALIDATION DUE TO ERRORS. RETURN TO PREPROCESSOR "INPUT VALIDATION" AND VALIDATE FILE OR TO "INPUT DEFINITION" AND CORRECT ERRORS.	**WARNING**

12.2 Error Messages During Preprocessor Input Definition

Explanations of errors which can occur during Preprocessor input for Single Component Analyses are shown in Section 12.2.1. Explanations for errors which can occur during Preprocessor input for Building Analyses are shown in Section 12.2.2.

12.2.1 Error Messages During Preprocessor Input for Single Component Analyses

<u>ERROR MESSAGE</u>	<u>ERROR TYPE</u>
FACE.CFG file not found!	**FATAL**
The FACE.CFG file is not in the program directory path. Exit to main menu and use configuration option to check path and check the FACE.CFG file exists in the intended directory.	
INPUT FILE SELECTED IN MAIN PROGRAM NOT FOUND ! SESSION FILE WILL BE USED IF IT EXISTS -OTHERWISE DEFAULT SETTINGS WILL BE MADE.	**WARNING**
Exit to FACEDAP main menu and use configuration option to check path for the data file directory and check the data files stored in this directory.	
OPTION2.DAT FILE NOT FOUND IN PROGRAM DIRECTORY UNABLE TO PROCEED WITHOUT THIS FILE !	**FATAL**
The OPTION2.DAT file is not in the program directory path. Exit to main menu and use configuration option to check path and check the OPTION2.DAT file is in intended area on hard disk.	
EXTENSION OF FILE NAME NOT BLG. DEFAULT SETTINGS WILL BE USED.	**WARNING**
THE BUILDING ANALYSIS INPUT FILE YOU HAVE RETRIEVED. CANNOT BE READ IN A SINGLE COMPONENT ANALYSIS. DEFAULT SETTINGS WILL BE USED UNLESS YOU RETRIEVE A SINGLE COMPONENT INPUT FILE BEGINNING WITH 'COMP'.	**WARNING**

ERROR MESSAGE	ERROR TYPE
ERROR OCCURRED READING SESSION FILE NAME DEFAULT INITIALIZATIONS WILL BE USED !!!	**WARNING**

Normally, when a default file must be read into the Preprocessor, the proper file name is read from a session file written by the FACEDAP program to the hard disk. However, this is not possible. Either exit to FACEDAP main menu and use File Option to retrieve an existing input file or input all data into the mostly empty input screens which will be shown in the Preprocessor.

LAST EDITED FILE NOT FOUND. DEFAULT INITIALIZATIONS WILL BE USED !!! — **WARNING**

Normally, when a default file must be read into the Preprocessor, the proper file name is read from a session file written by the FACEDAP program to the hard disk. However, this is not possible. Either exit to FACEDAP main menu and use File Option to retrieve an existing input file or input all data into the mostly empty input screens which will be shown in the Preprocessor.

PROBLEM TITLE AND/OR DESCRIPTION NOT DEFINED. — **WARNING**

Return to the *Problem Title* section of the main *Preprocessor* menu to define title and problem description.

LOAD DEFINITION NOT DEFINED FOR 1 OR MORE FIELDS. PLEASE DEFINE. — **WARNING**

Return to the *Load Definition* section of the main *Preprocessor* menu and define all input fields.

SCALED STANDOFF(Z) IS N — **WARNING**
Recommended Minimum: Z = 3.0 ft/lb^(1/3)
This Limit Helps to Ensure that Assumptions of Uniform Load over Entire Component and No Localized Failure Apply

The FACEDAP program will not allow input of a problem where the minimum scaled standoff between the charge and the selected component is less than 1.0 ft/lb$^{1/3}$. The scaled standoff is the minimum distance between the charge and the component (in the units of ft) divided by the cube root of the charge weight (in the units of lb). The minimum distance is measured

ERROR MESSAGE	ERROR TYPE
as a straight line distance from the charge to the component. The recommended minimum scaled standoff is 3.0 ft/lb$^{1/3}$. Refer to Section 8.0.2.3 for more discussion on this restriction.	
SCALED STANDOFF(Z) IS N **Minimum Allowable : Z = 1.0 ft/lb^(1/3)** **REDO INPUT and Increase Standoff or Reduce Charge Weight**	**WARNING**
The FACEDAP program will not allow input of a problem where the minimum scaled standoff between the charge and the selected component is less than 1.0 ft/lb$^{1/3}$. The scaled standoff is the minimum distance between the charge and the component (in the units of ft) divided by the cube root of the charge weight (in the units of lb). The recommended minimum scaled standoff is 3.0 ft/lb$^{1/3}$. Refer to Section 8.0.2.3 for more discussion on this restriction.	
NO COMPONENT PROPERTIES HAVE BEEN DEFINED. RETURN TO COMPONENT PROPERTIES AND DEFINE.	**WARNING**
Use the *Component Properties* option in the main *Preprocessor* menu to define at least one component. This input must precede Component Selection.	
ONLY ONE COMPONENT PROPERTY SET HAS BEEN DEFINED. NO COMPONENT SELECTION REQUIRED.	**WARNING**
COMPONENT SELECTION NOT MADE. PLEASE SELECT A COMPONENT.	**WARNING**
Use the *Component Selection* section of the main *Preprocessor* menu to select the component to be analyzed.	
ERROR OCCURRED BUILDING INPUT LIST. Reason for Error is One or More of the Following: **1. No Files Found with .BLG Extension.** **2. Data Directory Contains More than 200 Files.**	**WARNING**
The data path may be incorrectly set. Check that the data directory path is set to the directory containing the desired .BLG files. The other problem may be that more than 200 *.blg files are in the current data directory.	
FILENAME ENTERED CONTAINED A PERIOD. NO EXTENSION SHOULD BE INPUT. THE EXTENSION WILL BE APPENDED FOR YOU.	**WARNING**

ERROR MESSAGE	ERROR TYPE
4th COORDINATE HAS GREATER LOCAL X VALUE THAN 3rd COORD. RE-ENTER GLOBAL COORDINATES FOR WALL/ROOF AREA ""**	**WARNING**

The four corner points of the wall area (**) should be input such that the third point is closer to the second point than the fourth point.

A MASTER COMPONENT IS BEING DELETED. ALL COMPONENTS GENERATED FROM THIS MASTER WILL ALSO BE DELETED! ARE YOU SURE YOU WANT TO DELETE?	**WARNING**

(Y)es - Delete Master & Generated Components
(N)o - Do Not Delete Master

A MAXIMUM OF 200 PAIRS IS ALLOWED BY THE ANALYSIS PROGRAM. A TOTAL OF "x" PAIRS HAS BEEN CREATED. THE USER SHOULD DELETE ENOUGH INDEPENDENT IDs TO REMAIN WITHIN THE 200 ALLOWABLE PAIRS.	**WARNING**

More dependencies have been generated than the *Analysis* module can process. Each independent component which is calculated counts as a "dependency". The *Analysis* module cannot consider more than 200 dependencies. This problem can be remedied as discussed in Section 8.6.4.1.

ALL AVAILABLE SPACE HAS BEEN USED FOR STORING THE CURRENT DATA. DECREASE YOUR NUMBER OF ITEMS OR CONSULT PROGRAMMER FOR INCREASING ROW DIMENSION OF THE STORAGE ARRAY.	**WARNING**

If coming off a Component Geometry Definitions spreadsheet, too many total components have been entered (more than 370). If leaving a Component Property spreadsheet more than 150 components have been entered. Break building up so that two separate runs are used to analyze entire building. See Section 8.1.6 for more guidance.

ALL COORDINATES ARE ZERO FOR WALL/ROOF ""**	**WARNING**

THIS PAGE INTENTIONALLY LEFT BLANK

ERROR MESSAGE	ERROR TYPE

ASPECT RATIO IS BELOW MINIMUM VALUE FOR COMPONENT TYPE "**" FOR ID # "***"** — **WARNING**

A minimum aspect ratio of 0.09 for two-way components with simple end conditions and 0.3 for two-way components with fixed end conditions is allowed. The specified component (****) should be redefined as a one-way component.

BLASTWARD WALL "" , COMPONENT ID# "***" x COORDINATE NOT WITHIN WALL BOUNDS** — **WARNING**

The local coordinates input for the endpoint of the frame with the ID# (***) have been converted into global coordinates. The global x coordinate does not lie within the wall/roof area (**) input as the blastward wall. Refine local coordinates of the given frame endpoint in the *Frame Definition* spreadsheet or redefine the four corners of specified blastward wall area in *Wall/Roof Area Definition* spreadsheet.

BLASTWARD WALL "" , COMPONENT ID# "***" y COORDINATE NOT WITHIN WALL BOUNDS** — **WARNING**

The local coordinates input for the endpoint of the frame with the given ID# have been converted into global coordinates. The global y coordinate does not lie within the area defined by the four corner points of the given blastward wall area. Refine local coordinates of the given frame endpoint in the *Frame Definition* spreadsheet or redefine the four corners of given blastward wall area in *Wall/Roof Area Definition* spreadsheet.

CENTER TO CENTER SPACING NOT DEFINED FOR CURRENT ROW. PLEASE DEFINE. — **WARNING**

The component designated in the current row as a "master" component does not have center-to-center spacing defined which is necessary to generate additional components. Return to the *Component Geometry Definitions* spreadsheet for the specified wall/roof area and enter a value.

CHARGE WEIGHT AND/OR CHARGE COORDINATES HAVE NOT BEEN DEFINED. — **WARNING**

COMPONENT ID "*" NOT FOUND - CHECK DEPENDENCIES FOR POSSIBLE INCORRECT ENTRY OF AN INDEPENDENT ID#** — **WARNING**

The component ID number shown above has been entered as an Independent Component in one of the *Dependencies* spreadsheets and no such component ID number is located in the component list. Find this component ID number in the *Dependencies* spreadsheets and correct it or delete it.

ERROR MESSAGE	ERROR TYPE
ASPECT RATIO IS BELOW MINIMUM VALUE FOR COMPONENT TYPE "**" FOR ID # "***"**	**WARNING**

A minimum aspect ratio of 0.09 for two-way components with simple end conditions and 0.3 for two-way components with fixed end conditions is allowed. The specified component (****) should be redefined as a one-way component.

BLASTWARD WALL "" , COMPONENT ID# "***" x COORDINATE NOT WITHIN WALL BOUNDS**	**WARNING**

The local coordinates input for the endpoint of the frame with the ID# (***) have been converted into global coordinates. The global x coordinate does not lie within the wall/roof area (**) input as the blastward wall. Refine local coordinates of the given frame endpoint in the *Frame Definition* spreadsheet or redefine the four corners of specified blastward wall area in *Wall/Roof Area Definition* spreadsheet.

BLASTWARD WALL "" , COMPONENT ID# "***" y COORDINATE NOT WITHIN WALL BOUNDS**	**WARNING**

The local coordinates input for the endpoint of the frame with the given ID# have been converted into global coordinates. The global y coordinate does not lie within the area defined by the four corner points of the given blastward wall area. Refine local coordinates of the given frame endpoint in the *Frame Definition* spreadsheet or redefine the four corners of given blastward wall area in *Wall/Roof Area Definition* spreadsheet.

CENTER TO CENTER SPACING NOT DEFINED FOR CURRENT ROW. PLEASE DEFINE.	**WARNING**

The component designated in the current row as a "master" component does not have center-to-center spacing defined which is necessary to generate additional components. Return to the *Component Geometry Definitions* spreadsheet for the specified wall/roof area and enter a value.

CHARGE WEIGHT AND/OR CHARGE COORDINATES HAVE NOT BEEN DEFINED.	**WARNING**
COMPONENT ID "**" NOT FOUND - CHECK DEPENDENCIES FOR POSSIBLE INCORRECT ENTRY OF AN INDEPENDENT ID#**	**WARNING**

The component ID number shown above has been entered as an Independent Component in one of the *Dependencies* spreadsheets and no such component ID number is located in the component list. Find this component ID number in the *Dependencies* spreadsheets and correct it or delete it.

ERROR MESSAGE	ERROR TYPE
COMPONENT IS NOT A MASTER. CANNOT GENERATE FROM A UNIQUE.	**WARNING**

The F9 key has been used to generate components from the component in the current row of a *Component Geometry Definitions* spreadsheet but this component has not been designated as a "master" component.

COMPONENT MATERIAL TYPE NOT SELECTED. UNABLE TO PROCESS COMPONENT TYPE UNTIL COMPONENT MATERIAL TYPE SELECTED.	**WARNING**

The space bar has been pressed on the Component Type on the Component Geometry Definitions or Master Component spreadsheet, but the Material Type column is blank. It has not yet been selected. Until the Material Type is known, the Component Type list cannot be displayed.

COMPONENT MATERIAL TYPE OR COMPONENT TYPE NOT SELECTED. UNABLE TO PROCESS COMPONENT PROPERTY NAME UNTIL THESE SELECTIONS ARE MADE.	**WARNING**

The space bar has been pressed on the Component Property Name on the Component Geometry Definitions or Master Component spreadsheet, but the Material Type column and/or the Component Type fields are blank. Until the Material Type and Component Type are known, the Component Property Name list cannot be displayed.

COMPONENT PROPERTY NAME NOT DEFINED FOR ID # "*"**	**WARNING**

Return to the Component Geometry Definitions spreadsheet containing the given component ID number (***) and press the space bar to select a Component Property name for the component.

COMPONENT TYPE NOT DEFINED FOR ID # "*"**	**WARNING**

Return to the Component Geometry Definitions spreadsheet containing the given component ID number (***) and press the space bar to define a Component Type for the component.

COMPONENT TYPE NOT SELECTED. UNABLE TO PROCESS FRAME COMPONENT NAME UNTIL THESE SELECTIONS ARE MADE.	**WARNING**

The space bar has been pressed on the Frame Component Name of the Frame Definitions spreadsheet, but the Component Type field in column 1 is blank. Until the Component Type is known, the Frame Property Name list cannot be displayed.

ERROR MESSAGE	ERROR TYPE
COMPONENT TYPE "*****", COMPONENT ID# ***: BOTH ENDPOINTS ARE IDENTICAL	WARNING
COMPONENT TYPE "*****", COMPONENT ID# ***: NOT ALLOWED IN A ROOF AREA. Component types which include the word "exterior column" cannot be placed in a non-vertical wall/roof area. Change the given component to a different type or relocate it in a vertical wall area.	WARNING
COMPONENT TYPE "*****", COMPONENT ID# ***: NOT USUALLY ON ROOF An unusual Component Type has been located in a non-vertical (roof) area. Check that this component was not placed in a roof area by error.	WARNING
COMPONENT TYPE "*****", COMPONENT ID# ***: NOT USUALLY ALLOWED ON A WALL AREA. Components types with the word "roof" or "frame" (such as wood roofs) cannot be placed in a vertical wall/roof area. Also, open web steel joists are usually not in wall areas. Change the given component to a different type or relocate it in a roof area if there is an error.	WARNING
COMPONENT TYPE "*****", COMPONENT ID# ***: INPUT "BLASTWARD WALL" OF FRAME IS A ROOF. MUST BE A WALL. The wall area input as the "blastward wall" in the Frame Definitions spreadsheet under *Component Geometry Definitions* must be vertical.	WARNING
COMPONENT TYPE "*****", COMPONENT ID# ***: INTERIOR COLUMN MUST BE RELOCATED IN A ROOF Components with the words "interior column" in their component type (such as wood interior columns) cannot be placed in a vertical wall/roof area. Change the given component to a different component type or relocate it in a roof.	WARNING
COMPONENT TYPE "*****", COMPONENT ID# ***: NOT HORIZONTAL OR VERTICAL, INPUT AT AN ANGLE The given component type is usually parallel to either the local x or y coordinate system for the wall/roof area where it is located. Check that this component is not input at an angle with respect to the local coordinate system by mistake.	WARNING

ERROR MESSAGE	ERROR TYPE
COMPONENT TYPE "***", COMPONENT ID# ***: SHOULD NOT HAVE 2 SETS OF INPUT COORDINATES. 2nd SET OF COORDINATES WILL BE BLANKED.** Interior columns should only have one endpoint defined in the Component Geometry Definitions spreadsheets. A total of two were input for this component and the second set has been deleted by the program. Redefine the given component using only one endpoint if there is any possibility of error.	WARNING
COMPONENT TYPE "***", COMPONENT ID# ***: USUALLY NOT ON A WALL AREA** An unusual Component type has been located in a vertical (wall) area. Go to the Component Geometry Definitions spreadsheets and check that this component was not placed in wall area by error.	WARNING
COMPONENT TYPE - COMPONENT PROPERTY MISMATCH! COMPONENT PROPERTY "*" NOT VALID FOR COMPONENT TYPE "***" FOR ID # "***".** The given Component Property Name has not been defined for the given Component Type. Use the space bar option to redefine the Component Property Name or Component Type for the component with the given ID#.	WARNING
ENDPOINTS ARE NOT DIAGONAL FOR 2-WAY COMPONENT TYPE "***" FOR ID # "***"** A two-way component has been located with identical local x or y coordinates for its endpoints. Therefore, the endpoints do not define corner points of a flat planar area as required. Redefine component endpoints or redefine component as a one-way component.	WARNING
ERROR - SPACING MUST BE ≥ 0. The component designated in the current row as a "master" component has zero or negative center-to-center spacing defined. Redefine spacing as positive non-zero value or change component to a unique (non-master) component.	WARNING
ERROR - NUMBER GENERATED MUST BE ≥ 0. The component designated in the current row as a "master" component has a zero or negative number of additional components to be generated defined. Redefine this number as a positive non-zero value or change component to a unique (non-master) component.	WARNING

ERROR MESSAGE	ERROR TYPE
ERROR - NUMBER OF COMPONENTS TO BE GENERATED IS GREATER THAN THE MAXIMUM ALLOWABLE NUMBER OF GENERATED COMPONENTS. ONLY "50" COMPONENTS WILL BE GENERATED. CHECK NUMBER GENERATED COLUMN FOR POSSIBLE ERROR. SETTING FIELD TO MAX.	**WARNING**

Only 50 components may be generated from one "master" component. Create another "master" component and create half the components with one "master" and half with the other "master".

ERROR ALL COORDINATES OF LOCAL Y VECTOR = ZERO, REENTER GLOBAL COORD. FOR WALL AREA ** — **WARNING**

The four corner points input to define the given wall/roof area are colinear. These points must define a four-sided planar surface.

ERROR OCCURRED BUILDING INPUT LIST. REASON FOR ERROR IS ONE OR MORE OF THE FOLLOWING: — **WARNING**

1. **No Files Found with .BLG Extension.**
2. **Data Directory Contains More than 200 Files.**

The data path may be incorrectly set. Check that the data directory path is set to the directory containing the desired .BLG files. The other problem may be that more than 200 *.blg files are in the current data directory.

ERROR OCCURRED OPENING SPRDHEAD.DAT. CHECK SYSTEM PATH IN CONFIGURATION PROGRAM. — **WARNING**

The SPRDHEAD.DAT file is not in the program directory path. Exit to main menu and use configuration option to check path and check the SPRDHEAD.DAT file is in intended area on hard disk.

ERRORS OCCURRED IN WALL DEFINITIONS. UNTIL THESE ERRORS ARE CORRECTED, NO DEPENDENCY GENERATION OR MODIFICATIONS ARE ALLOWED. — **WARNING**

ERRORS OCCURRED IN WALL DEFINITIONS. UNTIL THESE ERRORS ARE CORRECTED, NO ENTERING OF COMPONENT DEFINITIONS IS ALLOWED. — **WARNING**

FACE.CFG FILE NOT FOUND! — **FATAL**

The FACE.CFG file is not in the program directory path. Exit to main menu and use configuration option to check path and check the FACE.CFG file exists in the intended directory.

ERROR MESSAGE	ERROR TYPE
FILENAME ENTERED CONTAINED A PERIOD. NO EXTENSION SHOULD BE INPUT. THE EXTENSION WILL BE APPENDED FOR YOU.	**WARNING**
FOUR WALL NODES ARE NOT COPLANAR. RE-ENTER WALL/ROOF NODES FOR WALL/ROOF AREA ""**	**WARNING**
**** WARNING ** INPUT CORNER POINT "x" IN WALL/ROOF AREA "**" DOES NOT HAVE COORDINATES WHICH MATCH ANY OTHER WALL/ROOF AREA CORNER** A set of wall/roof areas must be defined such that they all connect together at their corners to form an enclosed structure. This will not occur for the specified wall /roof area because it has one corner which does not match with at least one corner from another wall/roof area. This must be corrected.	**WARNING**
LOCAL COORDINATES FOR THE END POINTS OR CORNER POINTS HAVE NOT BEEN ENTERED. PLEASE ENTER. All the required endpoints for a component have not been entered into the current Component Geometry Definitions spreadsheet.	**WARNING**
MATERIAL TYPE - COMPONENT TYPE MISMATCH! COMPONENT TYPE "***" NOT VALID FOR MATERIAL TYPE "******" FOR ID # "***".** The Component Type of the specified component is associated with a Material Type different than that the input type. Check that the Material Type is valid and then use the space bar option to redefine the Component Type for the component with the given ID#.	**WARNING**
MAXIMUM NUMBER OF COMPONENTS HAS BEEN REACHED FOR THE CURRENT WALL/ROOF AREA NO MORE COMPONENTS WILL BE SAVED. Only 150 components can be input into a Component Geometry Definitions spreadsheet. Break the current wall/roof area into two separate wall areas in Wall/Roof Area Definitions input spreadsheet and redefine Component Geometry Definitions into the two wall areas.	**WARNING**

ERROR MESSAGE	ERROR TYPE
MAXIMUM NUMBER OF ROWS ALLOWED FOR DEPENDENCIES IS EXCEEDED! DECREASE NUMBER OF COMPONENTS FOR SPECIFIED WALL OR MAXIMUM ALLOWABLE IN PROGRAM.	**WARNING**
The dependency spreadsheet can only display dependencies for a maximum of 150 components per wall/roof area. The user should break up the given wall/roof area into two areas with half the total number of components per area and redefine component geometries within the new areas.	
NO COMPONENTS ARE DEFINED FOR THE CURRENT MATERIAL TYPE AND COMPONENT TYPE. RETURN TO COMPONENT PROPERTIES DEFINITION AND DEFINE.	**WARNING**
A set of component properties with a Component Property Name must be input in the *Component Properties Definition* before any component of the given Component Type can be defined in a *Component Geometry Definitions* spreadsheet.	
NO COMPONENT DEFINITIONS FOR WALL **	**WARNING**
No components were defined within the given wall/roof area.	
NO COMPONENT PROPERTIES ARE DEFINED FOR COMPONENT TYPE "**" FOR ID # "***".**	**WARNING**
A component with the specified component type (****) was input in the current Component Geometry Definitions spreadsheet but no set of component properties has been defined for this component type in the Component Properties Definition section. Such a set of component properties must be defined.	
WALL AREA "" IS NOT CONNECTED TO OTHER WALLS AT ALL FOUR CORNERS**	**WARNING**
A corner of the given wall/roof area is not connected to a corner from any other wall/roof area. Return to the Wall/Roof Area Definition spreadsheet and correct.	
NO FILE NAME ENTERED. PRESS RETURN IN RESPONSE TO THIS HELP MESSAGE, FOLLOWED BY PRESSING THE ESCAPE, IF YOU WISH TO ABORT !!!!!!	**WARNING**
NO FRAME COMPONENTS DEFINED IN CURRENT BUILDING	**WARNING**

ERROR MESSAGE	ERROR TYPE
NO FRAME COMPONENT PROPERTIES HAVE BEEN DEFINED. DEFINE FRAME PROPERTIES PRIOR TO ENTERING GEOMETRY INFORMATION.	**WARNING**

At least one set of properties must be input into the Component Property Definition spreadsheet for reinforced concrete or steel frames prior to entering the Frame Definitions spreadsheet used to define the location of frames.

NO GENERATED COMPONENTS FOUND FOR MASTER COMPONENT ID # * **	**WARNING**

The F9 key was not enacted by the user to generate the additional components of a "master" component in the previous Component Geometry Definitions spreadsheet. Return to the spreadsheet and make sure that additional generated components exist for all "master" components using the F10 key.

**** WARNING **** **NO INPUT WALL/ROOF AREAS ARE HORIZONTAL. USER MAY HAVE FORGOTTEN TO INPUT ROOF.**	**WARNING**
NO WALL/ROOF AREAS HAVE BEEN DEFINED. PLEASE DEFINE.	**WARNING**

Wall/roof areas must be defined before Wall Components can be defined and before component dependencies can be defined.

NUMBER OF ADDITIONAL REPEATED GROUPS TO BE GENERATED HAS NOT BEEN DEFINED. PLEASE DEFINE.	**WARNING**

The number of additional components to be generated has not been defined for a given component that has been identified as a "master" component. Define this number in the Component Geometry Definitions spreadsheet or change component to a unique component.

On the previous Component Geometry Spreadsheet, Component "*" was changed from a MASTER to a UNIQUE component. This component has been used to generate components. This change will result in the LOSS of the GENERATED components. Do you wish these generated components to be DELETED?**	WARNING

ERROR MESSAGE	ERROR TYPE
ONE OF GLOBAL COORDINATES HAD NEGATIVE LOCAL Y VALUE. RE-ENTER GLOBAL COORDINATES FOR WALL/ROOF AREA ""** The four corner points of the given wall/roof area are not sequential around the perimeter of the area in a clockwise or counterclockwise manner as required.	WARNING
ONE OR MORE VALUE(S) REQUIRED FOR THE CURRENT DEFAULT CALCULATION ARE ZERO (0). REQUIRED VALUES ARE CONTAINED IN COLUMNS TO THE LEFT OF THE CURRENT COLUMN. PLEASE DEFINE. The F7 key has been enacted in a Component Property Definition spreadsheet but the input values in columns to the left of the current column, which are required in the formula used to calculate the default input into the current column, have been defined equal to zero. The default input value cannot be calculated using zero values.	WARNING
ONE OR MORE VALUE(S) REQUIRED FOR THE CURRENT DEFAULT CALCULATION HAVE NOT BEEN DEFINED. REQUIRED VALUES ARE CONTAINED IN COLUMNS TO THE LEFT OF THE CURRENT COLUMN. PLEASE DEFINE. The F7 key has been enacted in a Component Property Definition spreadsheet but the input values in columns to the left of the current column, which are required in the formula used to calculate the default input into the current column, have not been defined.	WARNING
OPTIONTB.DAT FILE NOT FOUND IN PROGRAM DIRECTORY. UNABLE TO PROCEED WITHOUT THIS FILE ! The OPTIONB.DAT file is not in the program directory path. Exit to main menu and use configuration option to check path and check the OPTIONB.DAT file is in intended area on hard disk.	FATAL
PROBLEM TITLE AND/OR DESCRIPTION NOT DEFINED.	WARNING

ERROR MESSAGE	ERROR TYPE
RE-ENTER GLOBAL X AND Y COORDINATES IN EITHER CLOCKWISE OR COUNTERCLOCKWISE ORDER FOR WALL AREA/ROOF ""** The four corner points of the given wall/roof area are not sequential around the perimeter of the area in a clockwise or counterclockwise manner as required.	**WARNING**
RE-ENTER GLOBAL X AND Y COORDINATES FOR THIS WALL AREA It is possible that a non-numeric character or something which forms an invalid number has been entered. Return to Wall/Roof Area Definitions spreadsheet and check coordinates on specified wall/roof area.	**WARNING**
ROW "X" AND ROW "Y" HAVE DUPLICATE NAMES. PLEASE CORRECT. Duplicate names have been used to define component property sets in the previous input spreadsheet. This is not allowed since it can result in numerous errors if uncorrected.	**WARNING**
ROW "x" AND ROW "y" HAVE IDENTICAL WALL NAMES. PLEASE CORRECT. Duplicate names have been used to define wall/roof areas in the previous input spreadsheet. This is not allowed since it can result in numerous errors if uncorrected.	**WARNING**
ROW "x" USED THE RESERVED NAME "FRAME". PLEASE RENAME THIS WALL/ROOF AREA. Go to the Wall/Roof Area Definition spreadsheet and rename the area named "FRAME" in the xth row.	**WARNING**
UNDEFINED NODE(s) FOR WALL/ROOF AREA ""** Blank or unacceptable input exists for corner points in the given wall area. Go to the Wall/Roof Area Definition spreadsheet and correct this.	**WARNING**

ERROR MESSAGE	ERROR TYPE

WALL/ROOF "" COMPONENT ID #*** x COORDINATE NOT WITHIN WALL BOUNDS.** — WARNING

The local x coordinate input for one of the endpoints of the component with the given ID# is outside the range of values which fall within the four corner points of the given wall/roof area input into the Wall/Roof Area Definition spreadsheet. Check the local coordinate endpoints of the component. If these seem correct, check the global coordinates of the wall/roof area corner points, and the order with which corner points are defined. Use the F3 key in the Wall/Roof Area Definition spreadsheet or the User's Manual for guidance on requirements for definition of wall/roof area corner points.

WALL/ROOF "" COMPONENT ID #*** y COORDINATE NOT WITHIN WALL BOUNDS.** — WARNING

The local y coordinate input for one of the endpoints of the component with the given ID# is outside the range of values which fall within the four corner points of the given wall/roof area input into the Wall/Roof Area Definition spreadsheet. Check the local coordinate endpoints of the component. If these seem correct, check the global coordinates of the wall/roof area corner points, and the order with which corner points are defined. Use the F3 key in the Wall/Roof Area Definition spreadsheet or the User's Manual for guidance on requirements for definition of wall/roof area corner points.

WALL/ROOF AREA "" AND "**" HAVE IDENTICAL COORD.** — WARNING

The same wall/roof area has been input twice. One area must be deleted in the Wall/Roof Area Definition spreadsheet or one area must have its coordinates changed.

WALL/ROOF AREA "" HAS TWO IDENTICAL COORDINATES** — WARNING

The given wall/roof area does not have four unique corner points. This will probably cause the endpoints of some components input into this wall area to lie outside the area. Change one of the identical corner points in the Wall/Roof Area Definition spreadsheet.

ERROR MESSAGE	ERROR TYPE
WARNING - IT IS THE RESPONSIBILITY OF THE USER TO REGENERATE DEPENDENCIES IF ANY CHANGES ARE MADE TO THE WALL COORDINATES OR COMPONENT COORDINATES AFTER INITIAL GENERATION. THE PROGRAM DOES NOT PERFORM AUTOMATIC RE-GENERATION.	**WARNING**

This warning is issued whenever the user leaves the Building Geometry input section without viewing the dependencies spreadsheet for each wall/roof area since it is possible that the user created, deleted or edited components which affect the component dependencies without incorporating this effect into the dependencies. The program does not automatically generate dependencies so that it will not overwrite any dependencies input by the user. Dependencies are calculated by the program in the Dependencies spreadsheets when a Dependency spreadsheet is entered that has never had its dependencies generated or if the F9 generate/regenerate option is selected by the user.

ZERO COMPONENT LENGTH CALCULATED IN GENDPND SUBROUTINE FOR WALL/ROOF AREA "". UNDEFINED LOCAL COORDINATES FOR COMPONENT GEOMETRY DEFINITION DEFINITIONS IS MOST LIKELY CAUSE. DEPENDENCIES GENERATION HAS BEEN ABORTED.**	**WARNING**

If the user selects the generate option in one of the dependencies input spreadsheets, this option will only work if the endpoints of all non-interior column and non-frame components defined in the Component Geometry Definitions spreadsheets define a non-zero component length.

Note:
- * - Indicates Component Property Name
- ** - Indicates Wall/Roof Area Name
- *** - Indicates ID #
- **** - Indicates Component Type

ERROR MESSAGE	ERROR TYPE
WARNING - IT IS THE RESPONSIBILITY OF THE USER TO REGENERATE DEPENDENCIES IF ANY CHANGES ARE MADE TO THE WALL COORDINATES OR COMPONENT COORDINATES AFTER INITIAL GENERATION. THE PROGRAM DOES NOT PERFORM AUTOMATIC RE-GENERATION.	**WARNING**

This warning is issued whenever the user leaves the Building Geometry input section without viewing the dependencies spreadsheet for each wall/roof area since it is possible that the user created, deleted or edited components which affect the component dependencies without incorporating this effect into the dependencies. The program does not automatically generate dependencies so that it will not overwrite any dependencies input by the user. Dependencies are calculated by the program in the Dependencies spreadsheets when a Dependency spreadsheet is entered that has never had its dependencies generated or if the F9 generate/regenerate option is selected by the user.

ZERO COMPONENT LENGTH CALCULATED IN GENDPND SUBROUTINE FOR WALL/ROOF AREA "". UNDEFINED LOCAL COORDINATES FOR COMPONENT GEOMETRY DEFINITION DEFINITIONS IS MOST LIKELY CAUSE. DEPENDENCIES GENERATION HAS BEEN ABORTED.**	**WARNING**

If the user selects the generate option in one of the dependencies input spreadsheets, this option will only work if the endpoints of all non-interior column and non-frame components defined in the Component Geometry Definitions spreadsheets define a non-zero component length.

Note:
- * - Indicates Component Property Name
- ** - Indicates Wall/Roof Area Name
- *** - Indicates ID #
- **** - Indicates Component Type

12.3 Error Messages in Input Validation Module

Explanations of errors in which can occur during validation of input for Single Component Analyses are shown in Section 12.3.1. Explanations for errors which can occur during validation of input for Building Analyses are shown in Section 12.3.2.

12.3.1 Error Messages During Validation of Input for Single Component Analyses

ERROR MESSAGE	ERROR TYPE
FACE.CFG file not found!	**FATAL**
The FACE.CFG file is not in the program directory path. Exit to main menu and use configuration option to check path and check the FACE.CFG file exists in the intended directory.	
NO COMPONENTS HAVE BEEN DEFINED. NO FURTHER ERROR CHECKING WILL BE DONE.	**FATAL**
No components have been entered in the *Component Properties* section of the main *Single Component Preprocessor* menu.	
Problem Title Not Defined.	**WARNING**
Return to the *Problem Title* section of the main *Preprocessor* menu and input a problem title.	
Problem Description Not Defined.	**WARNING**
Return to the *Problem Title* section of the main *Preprocessor* menu and input a problem description.	
Charge Weight ≤ 0.	**FATAL**
Return to the *Load Definition* section of the main *Preprocessor* menu and input a positive valued charge weight.	
Standoff Not Defined.	**FATAL**
Return to the *Load Definition* section of the main *Preprocessor* menu and input a standoff.	

ERROR MESSAGE	ERROR TYPE

Standoff ≤ 0. — **FATAL**

Return to the *Load Definition* section of the main *Preprocessor* menu and input a positive valued standoff.

Scaled Standoff (Z) IS N
Minimum Allowable Z = 1.0 ft/lb^(1/3)
REDO INPUT and Increase Standoff or Reduce Charge Weight — **FATAL**

The FACEDAP program will not allow input of a problem where the minimum scaled standoff between the charge and the selected component is less than 1.0 ft/lb$^{1/3}$. The scaled standoff is the minimum distance between the charge and the component (in the units of ft) divided by the cube root of the charge weight (in the units of lb). The minimum distance is measured as a straight line distance from the charge to the component. The recommended minimum scaled standoff is 3.0 ft/lb$^{1/3}$. Refer to Section 8.0.2.3 for more discussion on this restriction.

Scaled Standoff (Z) IS N Recommended Minimum Z = 3.0 ft/lb^(1/3)
This Limit Helps to Ensure that Assumptions of Uniform Blast
Load over Entire Component and No Localized Failure Apply. — **WARNING**

The FACEDAP program will not allow input of a problem where the minimum scaled standoff between the charge and the selected component is less than 1.0 ft/lb$^{1/3}$. The scaled standoff is the minimum distance between the charge and the component (in the units of ft) divided by the cube root of the charge weight (in the units of lb). The recommended minimum scaled standoff is 3.0 ft/lb$^{1/3}$. Refer to Section 8.0.2.3 for more discussion on this restriction.

No Material Type Selected — **FATAL**

Return to the *Component Selection* section of the main *Preprocessor* menu and select at material type for the component type to be analyzed.

No Component Type Selected — **FATAL**

Return to the *Component Selection* section of the main *Preprocessor* menu and select at component type for the component to be analyzed.

No Component Name Selected — **FATAL**

Return to the *Component Selection* section of the main *Preprocessor* menu and select at the name of component to be analyzed.

ERROR MESSAGE	ERROR TYPE

Mismatch Between Material, Component Type and Component Name — FATAL

Return to the *Component Selection* section of the main *Preprocessor* menu and reselect the material, component type, and component name of the component to be analyzed in this order, using the pop-up menus for each selection.

Selected Component, "*" Not Found in Component Properties based on Selected Material & Component Type. — FATAL

Return to the *Component Selection* section of the main *Preprocessor* menu and reselect the material, component type, and component name of the component to be analyzed in this order, using the pop-up menus for each selection. If the name of the desired component is not shown on the pop-up menu, return to the *Component Properties* section of the main *Preprocessor* menu and define the component.

12.3.2 Error Messages During Validation of Input for Building Analyses

ERROR MESSAGE	ERROR TYPE

CHARGE COORDINATES NOT DEFINED. — WARNING

One or more of the explosive charge locations coordinates are blank. Return to the *Load Definition* section of the main *Preprocessor* menu and enter coordinates of charge.

CHARGE IS DIRECTLY OVER ROOF AREA "". INACCURATE BLAST LOADS WILL BE CALCULATED, REDO INPUT** — WARNING

Blast pressures in the *Analysis* module are calculated assuming a surface burst explosion. Therefore the explosive charge cannot be located over the building. Return to the *Load Definition* section of the main *Preprocessor* menu and reenter coordinates of charge or use another procedure to determine building damage.

CHARGE IS INSIDE BUILDING, REDO INPUT — FATAL

Only building damage and blast pressures from external explosive loading is considered in the FACEDAP program. Return to the *Load Definition* section of the main *Preprocessor* menu and reenter coordinates of charge or use another procedure to determine building damage.

CHARGE WEIGHT ≤ 0. — WARNING

12.3 Error Messages in Input Validation Module

ERROR MESSAGE	ERROR TYPE
CHARGE COORDINATES NOT DEFINED. One or more of the explosive charge locations coordinates are blank. Return to the *Load Definition* section of the main *Preprocessor* menu and enter coordinates of charge.	WARNING
CHARGE IS DIRECTLY OVER ROOF AREA "". INACCURATE BLAST LOADS WILL BE CALCULATED, REDO INPUT** Blast pressures in the *Analysis* module are calculated assuming a surface burst explosion. Therefore the explosive charge cannot be located over the building. Return to the *Load Definition* section of the main *Preprocessor* menu and reenter coordinates of charge or use another procedure to determine building damage.	WARNING
CHARGE IS INSIDE BUILDING, REDO INPUT Only building damage and blast pressures from external explosive loading is considered in the FACEDAP program. Return to the *Load Definition* section of the main *Preprocessor* menu and reenter coordinates of charge or use another procedure to determine building damage.	FATAL
CHARGE WEIGHT ≤ 0.	WARNING

ERROR MESSAGE	ERROR TYPE
CONNECTIVITY INDEX NOT FOUND. MAY RESULT FROM NOT ENTERING ANY WALL COORDINATES. NO FURTHER TEST DONE ON CHARGE UNTIL THIS IS CORRECTED.	**FATAL**

Return to *Wall/Roof Area Definition* in the *Building Geometry* section of the main *Preprocessor* menu and make sure all corner point coordinates of all wall/roof areas are defined. Each corner point must match with at least one corner point from another input wall/roof area. This check is made to prevent gaps and overlapping between wall/roof areas.

ERROR ALL COORDINATES OF LOCAL Y VECTOR = ZERO, REENTER GLOBAL COORD. FOR WALL AREA **	**WARNING**

The four corner points input to define the given wall/roof area are colinear. Therefore the y axis in the wall/roof local coordinate system cannot be defined. The four corner points must define a four-sided planar surface. Return to *Wall/Roof Area Definition* in the *Building Geometry* section of the main *Preprocessor* menu and redefine the corners of the specified wall/roof area (**).

FACE.CFG FILE NOT FOUND!	**FATAL**

The FACE.CFG file is not in the program directory path. Exit to main menu and use *Configuration* option under *Utilities* to check path and check the FACE.CFG file is in intended area on hard disk. Run FACECFG.EXE as explained in Section 3.0 if this file does not exist.

INPUT CORNER POINT * IN WALL/ROOF AREA "" DOES NOT HAVE COORDINATES WHICH MATCH ANY OTHER WALL/ROOF AREA CORNER**	**WARNING**

Each corner point of a wall/roof area must match with at least one corner point from another input wall/roof area. This check is made to prevent gaps and overlapping between wall/roof areas. A set of wall/roof areas must be defined such that they all connect together at their corners to form an enclosed structure. Return to *Wall/Roof Area Definition* in the *Building Geometry* section of the main *Preprocessor* menu and check all corner point coordinates of the specified wall/roof area (**) and connecting wall/roof areas.

ERROR MESSAGE	ERROR TYPE
INPUT FILE SELECTED IN MAIN PROGRAM NOT FOUND!	**FATAL**
NO COMPONENTS HAVE BEEN DEFINED. NO FURTHER ERROR CHECKING WILL BE DONE.	**FATAL**

No building components have been entered in the *Component Geometry Definitions* section under *Building Geometry* in the main *Preprocessor* menu. No error checking is done.

NO DEPENDENCIES HAVE BEEN GENERATED	**WARNING**

No Dependencies have been generated for any wall/roof areas. Return to the *Dependencies* under *Building Geometry* in the main *Preprocessor* menu and generate dependencies for each wall/roof area. See Section 8.6.4 for discussion on generating dependencies.

NO INPUT WALL/ROOF AREAS ARE HORIZONTAL. USER MAY HAVE FORGOTTEN TO INPUT ROOF.	**WARNING**

There must be at least one wall/roof area per building which is non-vertical so that it can be identified as a "roof", which is assumed to cover the interior of the building. One side of a "roof" area must overlay one side of the each vertical (i.e., wall) wall/roof area in the global X-Y plane so this wall area can be associated with the roof area and therefore the "inside" surface of the wall area can be distinguished from the "outside" surface. Return to *Wall/Roof Area Definition* in the *Building Geometry* section of the main *Preprocessor* menu and input a roof area.

NO ROOF CONNECTIVITY FOR WALL AREA ""**	**FATAL**

The specified wall area (**) does not connect to a non-vertical (i.e., roof) wall/roof area. One side of a roof area must overlay one side of the given "wall" area in the global X-Y plane so this wall area can be associated with the roof area and therefore the "inside" surface of the wall area can be distinguished from the "outside" surface. Return to

ERROR MESSAGE	ERROR TYPE
Wall/Roof Area Definition in the *Building Geometry* section of the main *Preprocessor* menu and check the corner point coordinates of the given wall area and the input roof areas.	
NUMBER WALL/ROOF AREA ≤ 4. ARE YOU SURE ALL WALL AREAS ARE DEFINED?	**WARNING**
Four or less wall areas have been defined. Typically a building has four walls and a roof. Return to *Wall/Roof Area Definition* in the *Building Geometry* section of the main *Preprocessor* menu make sure all wall/roof areas have been defined.	
PROBLEM DESCRIPTION NOT DEFINED.	**WARNING**
Return to the *Problem Title* section of the main *Preprocessor* menu and input a problem description.	
PROBLEM TITLE NOT DEFINED.	**WARNING**
Return to the *Problem Title* section of the main *Preprocessor* menu and input a problem title.	
ROW * AND ROW * COORDINATES ARE IDENTICAL.	**FATAL**
The same wall/roof area has been input twice into the *Wall/Roof Area Definition* spreadsheet in the *Building Geometry* section of the main *Preprocessor* menu. The wall/roof area in one of the specified rows (*) in this spreadsheet must be deleted or have its corner point coordinates changed.	
ROW * AND ROW * WALL NAMES ARE IDENTICAL.	**FATAL**
Duplicate names have been used to define wall/roof areas in the *Wall/Roof Area Definition* spreadsheet in the *Building Geometry* section of the main *Preprocessor* menu. This is not allowed since it can result in numerous errors if uncorrected. The name of the wall/roof areas in one of the specified rows in the spreadsheet must be changed.	

ERROR MESSAGE	ERROR TYPE
Same Global Coordinates	FATAL

Area "**", ****, ID# *** &
Area "**", ****, ID# ***
End 1 X11 Y11 Z11 ID# ***
End 1 X21 Y21 Z21 ID# ***
End 2 X12 Y12 Z12 ID# ***
End 2 X22 Y22 Z22 ID# ***

During a check of the global coordinates of all building components, two components were found which have identical endpoints. Each component should only be defined once. Return to the *Component Geometry Definitions* within the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof areas (**), and delete one of the components with the specified ID numbers (***). Typically, this occurs when a components lies along a shared boundary between adjacent wall/roof areas and it is input into both areas. Also, a component may have been defined from as spanning from Endpoint 1 to Endpoint 2 and then redefined as spanning from Endpoint 2 to Endpoint 1.

SCALED STANDOFF (Z) TO WALL "**" IS N FT/LB^(1/3)	FATAL

Minimum Allowable: Z = 1.0 ft/lb^(1/3)
Recommended Minimum: Z = 3.0 ft/lb^(1/3)

TO ENSURE THAT ASSUMPTIONS OF UNIFORM BLAST OVER ENTIRE COMPONENT AND NO LOCAL RESPONSE APPLY

The FACEDAP program will not allow input of a problem where the minimum scaled standoff between the charge and the building is less than 1.0 ft/lb$^{1/3}$. The scaled standoff is the minimum distance between the charge and the building (in the units of ft) divided by the cube root of the charge weight (in the units of lb). The minimum distance is measured as a straight line distance from the charge to the building and then up to the midheight of the building. The recommended minimum scaled standoff is 3.0 ft/lb$^{1/3}$. Refer to Section 8.4 for more discussion on this restriction.

ERROR MESSAGE	ERROR TYPE
SCALED STANDOFF (Z) TO WALL "" IS N FT/LB^(1/3) RECOMMENDED MINIMUM Z = 3.0 FT/LB^(1/3) TO ENSURE THAT ASSUMPTIONS OF UNIFORM BLAST LOAD OVER ENTIRE COMPONENT AND NO LOCAL RESPONSE APPLY**	**WARNING**

The recommended minimum scaled standoff between the charge and the building is 3.0 $ft/lb^{1/3}$. The scaled standoff is the minimum distance between the charge and the building (in the units of ft) divided by the cube root of the charge weight (in the units of lb). The minimum **distance is measured as a straight line distance** from the charge to the building and then up to the midheight of the building. Refer to Section 8.4 for more discussion on this restriction.

STANDOFF TO WALL "" NEAREST CHARGE = * FT THIS IS LESS THAN 5 TIMES THE CHARGE HEIGHT FROM GROUND THIS MAY INVALIDATE SURFACE BURST ASSUMPTION USED IN BLAST LOAD CALCULATIONS**	**WARNING**

The blast damage calculation procedure in the *Analysis* module assumes that the blast load is applied from a surface burst of the explosive. Therefore, it is assumed that the distance between the explosive charge and the ground surface is very small compared to the distance from the charge to the building.

TOTAL DEPENDENCY PAIRS = "*". MAXIMUM ALLOWABLE IS 200. THE BDAMA ANALYSIS MODULES IS LIMITED TO 200 DEPEND.	**FATAL**

More dependencies have been generated than the *Analysis* module executable of the program (BDAMA) can process. Each Independent Component in the dependency spreadsheets for each wall/roof area constitutes a "dependency". See advice in Section 8.6.4.1 for reducing the total number of dependencies without affecting the accuracy of the calculated building damage.

ERROR MESSAGE	ERROR TYPE
WALL "**" CONTAINS UNDEFINED COORDINATES.	FATAL

Blank or unacceptable input exists for corner points in the specified wall area (**). Go to the *Wall/Roof Area Definition* spreadsheet in the *Building Geometry* section of the main *Preprocessor* menu and correct this.

WALL/ROOF "" HAS 4TH COORD. > 3RD COORD.** — **FATAL**

The fourth input corner point of the specified wall/roof area (**) has a greater x value in the local wall/roof area coordinate system than the third input corner. This probably indicates that the corner points were not input sequentially around the perimeter of the wall/roof area. Go to the *Wall/Roof Area Definition* spreadsheet in the *Building Geometry* section of the main *Preprocessor* menu and correct this.

WALL/ROOF "" HAS A NEGATIVE Y.** — **FATAL**

This indicates that the four corner points of the specified wall/roof area (**) are not sequentially around the perimeter of the area in a clockwise or counterclockwise manner as required. Go to the *Wall/Roof Area Definition* spreadsheet in the *Building Geometry* section of the main *Preprocessor* menu and correct this.

WALL/ROOF "" HAS ALL COORDINATES = 0** — **FATAL**

Go to the *Wall/Roof Area Definition* spreadsheet in the *Building Geometry* section of the main *Preprocessor* menu and define corner point coordinates for the specified wall/roof area (**).

WALL/ROOF "" HAS UNREADABLE COORDINATES.** — **FATAL**

It is possible that a non-numeric character or something which forms an invalid number has been entered for the specified wall/roof area (**). Return to the *Wall/Roof Area Definition* spreadsheet in the *Building Geometry* section of the main *Preprocessor* menu and check the input corner point coordinates and wall/roof name.

ERROR MESSAGE	ERROR TYPE

WALL/ROOF "" IS NOT COPLANAR.** — FATAL

The four coordinates of the specified wall/roof area (**) are not coplanar. Return to the *Wall/Roof Area Definition* spreadsheet in the *Building Geometry* section of the main *Preprocessor* menu and check the input corner point coordinates.

WALL/ROOF "" RE-ENTER COORDINATES CLOCKWISE OR COUNTERCLOCKWISE** — FATAL

The four corner points of the specified wall/roof area (**) are not input sequentially around the perimeter of the area in a clockwise or counterclockwise manner as required. Go to the *Wall/Roof Area Definition* spreadsheet in the *Building Geometry* section of the main *Preprocessor* menu and correct this.

WALL/ROOF AREA "": DEFINED BUT NOT USED** — WARNING

The specified wall/roof area (**) has been defined on the Wall/Roof Area Definitions spreadsheet, but has not been assigned any components. Was this area supposed to be used? If so, return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu and define the required components within the given wall/roof area. Otherwise, there is no problem.

WALL/ROOF AREA "": DEPENDENCIES HAVE NOT BEEN GENERATED** — WARNING

No dependencies have been generated for the specified wall/roof area (**). If the user forgot to generate dependencies for this wall/roof area return to the *Dependency* section under the *Building Geometry* section of the main *Preprocessor* menu and generate dependencies for the given wall/roof area. Some wall/roof areas will have no calculated dependencies and, in this case, this warning should be ignored.

ERROR MESSAGE	ERROR TYPE
WALL/ROOF AREA, ROW *: NAME IS RESERVED WORD, FRAME.	**FATAL**

Go to the *Wall/Roof Area Definition* spreadsheet in the *Building Geometry* section of the main *Preprocessor* menu and rename the wall/roof area in the specified row (*) currently named "FRAME".

WEIGHTING FACTOR ALERT. ALL COMPONENTS WEIGHTING FACTORS ARE AT THE DEFAULT VALUE OF 1.0. THIS IMPLIES THAT THE CALCULATED BUILDING DAMAGE GIVES TOO MUCH CONSIDERATION TO DAMAGED CLAPPED COMPONENTS AND NOT ENOUGH CONSIDERATION TO DAMAGED FRAMING COMPONENTS.	**WARNING**

See discussion of weighting factors in Section 8.5.2.

Z DIRECTION DISTANCE TO CHARGE (OFF GROUND) IS NEGATIVE. IT IS RESET TO 0 BY PROGRAM.	**WARNING**

A negative Z coordinate indicates that the charge is underground according to the assumed coordinate system in the FACEDAP program. **Such input charge locations are not allowed. Return** to the *Load Definition* section of the main *Preprocessor* menu and enter an acceptable Z coordinate for the charge location.

******, "*": COL. I IS UNDEFINED**	**FATAL**

The set of component properties with the specified Component Property Name (***) for the Component Type (*) has a blank field in the Ith column of the input spreadsheet screen. Return to the input spreadsheet for the specified component type under the *Component Properties* section of the main *Preprocessor* menu and define this value.

ERROR MESSAGE	ERROR TYPE

******, "*": COL. I ≤ 0** — FATAL

The set of component properties with the specified Component Property Name (***) for the specified Component Type (*) has a 0 or negative value in the Ith column of the input spreadsheet screen. Return to the input screen for the specified component type under the *Component Properties* section of the main *Preprocessor* menu and correct this value.

******, "*" IS A DUPLICATE NAME** — FATAL

The specified Component Type (***) has two property sets with the same Component Property Name (*). Return to the input screen for the specified component type under the *Component Properties* section of the main *Preprocessor* menu and re-name one of the property sets to a unique name.

***** & *** OF AREA "**" :IDENTICAL COORD.** — FATAL

The specified Wall/Roof area (**) has 2 components with the same local coordinates. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and correct the coordinates of the specified components.

***** OF AREA "**": BAD ASPECT RATIO** — FATAL

A minimum aspect ratio (the ratio of the short span to long span of a two-way component) of 0.09 is required for two-way components with **simple end conditions and a minimum value of 0.3 is required** for two-way components with fixed end conditions. The component with the specified ID number (***) in wall/roof area (**) should be redefined as a one-way component in the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu.

ERROR MESSAGE	ERROR TYPE
*** OF AREA "**": BLASTWARD WALL NOT DEFINED	FATAL

The specified component ID number (***), which is a frame component, does not have a defined Blastward Wall. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the FRAME option, and define the blastward wall for the specified component.

*** OF AREA "**": C/C SPACING ≤ 0.	FATAL

The component with the ID number (***) in the Wall/Roof area (**) is designated as a "master" component but it has a center-to-center spacing equal to or less than 0. A positive spacing is necessary to generate additional components. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and correct this value or make the component a unique component.

*** OF AREA "**": C/C SPACING NOT DEFINED	FATAL

The component with the ID number (***) in the Wall/Roof area (**) is designated as a "master" component but it has an undefined center-to-center spacing. This value is necessary to generate additional components. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and correct this value or make the component a unique (non-master) component.

*** OF AREA "**": COMP. PROPERTY NAME NOT DEFINED	FATAL

The component with the ID number (***) in the Wall/Roof area (**) does not have an input Component Property Name. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and **input a Component Property Name for the specified component by** pressing the space bar in the correct column and select from the display of acceptable Component Property Names.

ERROR MESSAGE	ERROR TYPE
***** OF AREA "**": COMP. TYPE-COMP. PROP. MISMATCH**	**FATAL**

The component with the ID number (***) in the Wall/Roof area (**) has an input Component Property Name that was input for a Component Type different than that of the component. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, check that the input component type is correct for the specified component, and input a Component Property Name by pressing the space bar in the correct column and selecting an acceptable Component Property Name from the displayed list.

***** OF AREA "**": COMPONENT TYPE NOT DEFINED** — **FATAL**

The component with the ID number (***) in the Wall/Roof area (**) does not have an input Component Type. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and input a Component Type for the specified component by pressing the space bar in the correct column and selecting from a display of acceptable Component Property Types.

***** OF AREA "**": COMPS. NOT GENERATED FROM MASTER** — **FATAL**

The component with the ID number (***) defined in the wall/roof area (**) is a "master" component but no generated components exist for it. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and generate components by placing the cursor in the row of the specified component and pressing F9, or change the component to a unique (non-master) component.

ERROR MESSAGE	ERROR TYPE
***** OF AREA "**": END 1 NOT DEFINED OR INVALID'**	**FATAL**

The input local coordinates of the endpoint input for the Frame or Interior Column type of component with the specified component ID number (***) the in wall/roof area (**) have not been defined or have fields containing invalid data. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area and define the first endpoint coordinates.

***** OF AREA "**": ENDPOINT NOT DEFINED** — **FATAL**

This is similar to the error message above.

***** OF AREA "**": ENDPOINTS NOT DEFINED** — **FATAL**

The local coordinates of one of the endpoints input for the component with the specified ID number (***) in the wall/roof area (**) have not been defined or have fields containing invalid data. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and define the endpoint coordinates.

***** OF AREA "**": ENDPOINTS NOT DIAGONAL** — **FATAL**

The local coordinates of the endpoints input for the two-way type component with the specified ID number (***) in the wall/roof area (**) define a line parallel to the local x or y coordinate system. Therefore, they do not define corner points of a planar surface as they should for a two-way component. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and redefine the endpoint coordinates.

ERROR MESSAGE	**ERROR TYPE**

***** OF AREA "**": FRAME HEIGHT ≤ 0** — **FATAL**

The input frame height for the Frame type of component with the specified component ID number (***) has not been defined as a positive value as it required. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the FRAME option, and input an acceptable height.

***** OF AREA "**": FRAME HEIGHT NOT DEFINED** — **FATAL**

This is similar to the error preceding error message.

***** OF AREA "**": INDEPENDENT ID# "***" HAS NO CORRESPONDING DEPENDENT ID#** — **FATAL**

The component ID number shown above has been entered as an Independent Component in the *Dependencies* input screens for the specified wall/roof area (**) and no such component ID number is located in the component list. Return to the *Dependencies* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area (**), find the specified Independent Component ID number ("***") in the row of the specified Dependent Component number (***) and correct it or delete it.

***** OF AREA "**": INDEPENDENT ID# "***" NOT UNIQUE FOR DEPENDENT ID#** — **FATAL**

The Independent Component ID # (***) is defined twice for the same Dependent Component. Return to the *Dependencies* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area (**), and delete or correct the second occurrence of the specified Independent ID#.

ERROR MESSAGE	ERROR TYPE
*** OF AREA "**": MATERIAL-COMP. TYPE MISMATCH'	FATAL

The specified component ID number (***) in the specified Wall/Roof area (**) has an input Material Type which is not consistent with the input Component Type. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, check that the input material type is correct for the specified component, and input a component type by pressing the space bar in the correct column and selecting an acceptable component type from a displayed list.

*** OF AREA "**": NO GENERATED COMPONENTS FOUND	FATAL

The specified component ID number (***) defined in wall/roof area (**) is a "master" component but no generated components exist for it. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and generate components by placing the cursor in the row of the specified component and pressing F9 if this component was correctly designated as a master component. If the component is not intended to be a master component, correct this in the appropriate column of the input screen.

*** OF AREA "**": NOT IN COMPONENT PROPERTIES	FATAL

The component with the ID number (***) in the Wall/Roof area (**) has an input Component Property Name which does not have a defined set of properties. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, check that the input component type is correct for the specified component, and input a Component Property Name by pressing the space bar in the correct column to select from a display of acceptable names. Also check that this component property set been accidentally deleted in the *Component Properties* section in the main *Preprocessor* menu.

ERROR MESSAGE	ERROR TYPE

***** OF AREA "**": NUMBER GENERATED ≤ 0.** — **FATAL**

The component with the ID number (***) in the Wall/Roof area (**) in designated as a "master" component but it has a zero or negative number of additional components to be generated. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and input a positive non-zero value or make the component a unique (non-master) component.

***** OF AREA "**": NUMBER ADD. REPEATED GROUPS (i.e., Generated Components) NOT DEFINED** — **FATAL**

This is similar to the preceding error message.

***** OF AREA "**": "X" NOT WITHIN WALL BOUNDS** — **FATAL**

The component with the ID number (***) in the Wall/Roof area (**) has a local x coordinate which is outside the bounds of the specified wall/roof area (**). Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and correct the x coordinate of the specified component. It is also possible that the global coordinates of the specified wall/roof area are not correct in the *Wall/Roof Area Definitions* section under the *Building Geometry* section of the main *Preprocessor* menu.

***** OF AREA "**": "Y" NOT WITHIN WALL BOUNDS** — **FATAL**

This is similar to the preceding message except that it pertains to the local y coordinate of the specified component.

ERROR MESSAGE	ERROR TYPE
***** OF AREA "**", TYPE ****: 2ND SET OF INPUT COORD. BLANKED BY PROGRAM**	**WARNING**

The component with the ID number (***), of the component type (****), in the Wall/Roof area (**) is an interior column or frame type of component which should only have one endpoint defined. Two endpoints were defined and the second one has been deleted by the program. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area or the FRAME option, and check that the intended coordinates are still entered.

***** OF AREA "**", TYPE ****: IDENTICAL ENDPOINTS**	**FATAL**

The component with the ID number (***), of the component type (****), in the specified Wall/Roof area (**) has two identical endpoints. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and correct one of the endpoints.

***** OF AREA "**", TYPE ****: NOT ALLOWED IN A ROOF AREA**	**FATAL**

The component with the ID number (***), of the component type (****), in the Wall/Roof area (**) is located in a non-vertical (i.e., roof) wall/roof area. Components with the word "exterior column" in their component type names must be placed in a vertical (i.e., wall) wall/roof area. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and delete the specified component and relocate in a wall area or change the component type.

ERROR MESSAGE	ERROR TYPE
***** OF AREA "**", TYPE ****: MUST LOCATE IN A ROOF AREA**	**FATAL**

The component with the ID number (***), of the component type (****), in the Wall/Roof area (**) is located in a vertical (i.e., wall) wall/roof area. Components with the word "interior column" in their component type names must be placed in a non-vertical (i.e., roof) wall/roof area. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and delete the specified component and relocate in a roof area or change the component type.

***** OF AREA "**", TYPE ****: NOT HORIZONTAL OR VERTICAL (INPUT AT AN ANGLE)'**	**WARNING**

The component with the ID number (***), of the component type (****), in the Wall/Roof area (**) has endpoints which do not define a line up parallel to the local x or y axis of the wall/roof area coordinate system. This type of component almost always does line up with the local x or y axis. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and check the input endpoint coordinates of the specified component.

***** OF AREA "**", TYPE ****: USUALLY NOT ON ROOF**	**WARNING**

The component with the ID number (***), of the component type (****), in the Wall/Roof area (**) is located in a non-vertical (i.e., roof) wall/roof area. This type of components is not usually located in a roof. Return to the *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and check that there is not an error.

ERROR MESSAGE	ERROR TYPE
***** OF AREA "**", TYPE ****: USUALLY NOT ON WALL AREA**	**WARNING**

The component with the ID number (***), of the component type (****), in the Wall/Roof area (**) is located in a vertical (i.e., wall) wall/roof area. This type of components is not usually located in a wall. **Return to the** *Component Geometry* section under the *Building Geometry* section of the main *Preprocessor* menu, select the specified wall/roof area, and check that there is not an error.

***** OF BLAST WALL "**", TYPE ****: BLASTWARD WALL IS ROOF, SHOULD BE A WALL**	**FATAL**

The frame type component with the specified ID number (***) and of the given component type (****) has an input blastward wall/roof area (**) which is non-vertical (i.e., a roof). The blastward wall area must always be a vertical area. Return to the *Component Geometry* section **under the** *Building Geometry* section of the main *Preprocessor* menu, select the FRAME option, and reselect the blastward wall using the space bar to display a list of acceptable wall/roof area names which can be selected.

Note:
- * - Indicates Component Property Name
- ** - Indicates Wall/Roof Area Name
- *** - Indicates ID #
- **** - Indicates Component Type

12.4 Error Messages in Analysis Module

ERROR MESSAGE	ERROR TYPE

CANNOT FIND INDEPENDENT ID # FOR DEPENDENT ID # *** — FATAL

The specified Independent ID# was not found. Check the Dependency spreadsheet to see if the Independent ID was input incorrectly. Most probably changes were made to the Component Geometry Definitions in a given wall/roof area and then the F9 key was not used in the *Dependencies* option of the *Building Geometry* option off the main *Preprocessor* menu to regenerate dependencies considering the changed geometry.

FACE.CFG FILE NOT FOUND! — FATAL

The FACE.CFG file is not in the program directory path. Exit to main menu and use configuration option (under Utilities) to check path and check the FACE.CFG file is in intended area on hard disk or that it has not been created.

NO INPUT FILE ENTERED ON COMMAND LINE! MAKEBDMA TERMINATING — FATAL

This error occurs when MAKEBDMA.EXE is run stand-alone (outside of the FACEDAP program) and no file name is specified on the command line. MAKEBDMA is the program which writes the input file for the analysis executable BDAMA.EXE.

INPUT FILE SELECTED IN MAIN PROGRAM NOT FOUND! MAKEBDMA TERMINATING — FATAL

This error occurs when MAKEBDMA.EXE cannot find the selected file. This error should only occur if MAKEBDMA is run outside of FACEDAP and the file name specified on the command line cannot be found. The file on the command line must specify the full path with the file name. The data directory path is not added by MAKEBDAM.

12.4 Error Messages in Analysis Module

ERROR MESSAGE	ERROR TYPE
CANNOT FIND INDEPENDENT ID # FOR DEPENDENT ID # ***	**FATAL**

The specified Independent ID# was not found. Check the Dependency spreadsheet to see if the Independent ID was input incorrectly. Most probably changes were made to the Component Geometry Definitions in a given wall/roof area and then the F9 key was not used in the *Dependencies* option of the *Building Geometry* option off the main *Preprocessor* menu to regenerate dependencies considering the changed geometry.

FACE.CFG FILE NOT FOUND! — **FATAL**

The FACE.CFG file is not in the program directory path. Exit to main menu and use configuration option (under Utilities) to check path and check the FACE.CFG file is in intended area on hard disk or that it has not been created.

NO INPUT FILE ENTERED ON COMMAND LINE! MAKEBDMA TERMINATING — **FATAL**

This error occurs when MAKEBDMA.EXE is run stand-alone (outside of the FACEDAP program) and no file name is specified on the command line. MAKEBDMA is the program which writes the input file for the analysis executable BDAMA.EXE.

INPUT FILE SELECTED IN MAIN PROGRAM NOT FOUND! MAKEBDMA TERMINATING — **FATAL**

This error occurs when MAKEBDMA.EXE cannot find the selected file. This error should only occur if MAKEBDMA is run outside of FACEDAP and the file name specified on the command line cannot be found. The file on the command line must specify the full path with the file name. The data directory path is not added by MAKEBDAM.

12.5 Error Messages in Postprocessor

Explanations of errors in which can occur during use of the Postprocessor to view output from Single Component Analyses are shown in Section 12.5.1. Explanations for errors which can occur during use of the Postprocessor to view output from Building Analyses are shown in Section 12.5.2.

12.5.1 Error Messages in Postprocessor Related to Single Component Analyses

ERROR MESSAGE	ERROR TYPE
FACE.CFG file not found!	**FATAL**
The FACE.CFG file is not in the program directory path. Exit to main menu and use configuration option to check path and check the FACE.CFG file exists in the intended directory.	
INPUT FILE SELECTED IN MAIN PROGRAM NOT FOUND! POSTPROCESSOR TERMINATING.	**FATAL**
Exit to FACEDAP main menu and use configuration option to check path for the data file directory and check the data files stored in this directory.	
THE BUILDING ANALYSIS INPUT FILE YOU HAVE RETRIEVED CANNOT BE USED IN THE SINGLE COMPONENT POSTPROCESSOR.	**FATAL**
OUTPUT FILE, &&&.PST, DOES NOT EXIST. EXECUTE BDAMA PROCESS PROGRAM FROM FACEDAP. POSTPROCESSOR TERMINATING.	**FATAL**
The ANALYSIS option in the FACEDAP main menu must be exercised before the building damage can be viewed.	
TOTAL NUMBER COMPONENTS FROM BDAMA ARE DIFFERENT THAN FROM THE PREPROCESSOR INPUT FILE. MAKE SURE EXECUTABLE HAS BEEN RUN SINCE LAST PREPROCESSOR UPDATES.	**FATAL**
Rerun the ANALYSIS option in the FACEDAP main menu to ensure that consistent versions of the input and output are in the data file directory.	

ERROR MESSAGE	ERROR TYPE
PRINTING OF GENERATED REPORT WILL BE INITIATED UPON RETURNING TO THE FACEDAP MAIN MENU.	**WARNING**

Printing does not begin until the user exits the Postprocessor.

12.5.2 Error Messages in Postprocessor Related to Building Analyses

ERROR MESSAGE	ERROR TYPE
ALL COMPONENTS WEIGHTING FACTORS ARE AT THE DEFAULT VALUE OF 1.0. THIS IMPLIES THAT THE CALCULATED BUILDING DAMAGE GIVES TOO MUCH CONSIDERATION TO DAMAGED CLADDING COMPONENTS AND NOT ENOUGH CONSIDERATION TO DAMAGED FRAMING COMPONENTS.	**WARNING**

See discussion of weighting factors in Section 8.5.2.

BUILDING SUSTAINED NO SIGNIFICANT DAMAGE. PROCEED TO ALL COMPONENTS FOR PRINTING.	**WARNING**

The most damage components is not printed when all components have 0% damage.

BUILDING SUSTAINED NO SIGNIFICANT DAMAGE. PROCEED TO ALL COMPONENTS FOR REVIEWING.	**WARNING**

The most damage components is not shown when all components have 0% damage.

FACE.CFG FILE NOT FOUND!	**FATAL**

The FACE.CFG file is not in the program directory path. Exit to main program and use Configuration option under Utilities to check path and check the FACE.CFG file is in intended area on hard disk or that it has not been created.

INPUT FILE SELECTED IN MAIN PROGRAM NOT FOUND! BDAMPOST TERMINATING	**FATAL**

THIS PAGE INTENTIONALLY LEFT BLANK

12.5 Error Messages in Postprocessor

ERROR MESSAGE	ERROR TYPE
ALL COMPONENTS WEIGHTING FACTORS ARE AT THE DEFAULT VALUE OF 1.0. THIS IMPLIES THAT THE CALCULATED BUILDING DAMAGE GIVES TOO MUCH CONSIDERATION TO DAMAGED CLADDING COMPONENTS AND NOT ENOUGH CONSIDERATION TO DAMAGED FRAMING COMPONENTS. See discussion of weighting factors in Section 8.5.2.	**WARNING**
BUILDING SUSTAINED NO SIGNIFICANT DAMAGE. PROCEED TO ALL COMPONENTS FOR PRINTING. The most damage components is not printed when all components have 0% damage.	**WARNING**
BUILDING SUSTAINED NO SIGNIFICANT DAMAGE. PROCEED TO ALL COMPONENTS FOR REVIEWING. The most damage components is not shown when all components have 0% damage.	**WARNING**
FACE.CFG FILE NOT FOUND! The FACE.CFG file is not in the program directory path. Exit to main program and use Configuration option under Utilities to check path and check the FACE.CFG file is in intended area on hard disk or that it has not been created.	**FATAL**
INPUT FILE SELECTED IN MAIN PROGRAM NOT FOUND! BDAMPØST TERMINATING	**FATAL**

ERROR MESSAGE	ERROR TYPE
MAXIMUM NUMBER OF COMPONENTS HAS EXCEEDED ARRAY DIMENSION. ONLY THE FIRST 150 COMPONENTS WILL BE DISPLAYED. CONSULT PROGRAMMER FOR INCREASING DIMENSION OF ARRAYS OR SUBDIVIDE CURRENT WALL INTO 2 WALLS IN THE PREPROCESSOR.	**WARNING**
NO FRAME COMPONENTS IN CURRENT BUILDING No frame components can be viewed if none specified in component definitions.	**WARNING**
NO COMPONENTS DAMAGED ON WALL/ROOF AREA ""** If all components in a wall/roof area have 0% computed damage, no component damage screen is shown.	**WARNING**
OPTIONTB.DAT FILE NOT FOUND IN PROGRAM DIRECTORY - UNABLE TO PROCEED WITHOUT THIS FILE! The OPTIONB.DAT file is not in the program directory path. Exit to main program and use configuration option to check path and check the OPTIONB.DAT file is in intended area on hard disk.	**FATAL**
OUTPUT FILE, &&&.PST, DOES NOT EXIST. EXECUTE BDAMA PROCESS PROGRAM FROM FACEDAP. BDAMPOST TERMINATING. The ANALYSIS option in the main menu must be exercised before the building damage can be viewed.	**FATAL**
PRINTING OF GENERATED REPORT(S) WILL BE INITIATED UPON RETURNING TO THE BDAM MAIN MENU. Printing does not begin until the user exits the Postprocessor.	**WARNING**

ERROR MESSAGE	ERROR TYPE
TOTAL NUMBER COMPONENTS FROM BDAMA ARE DIFFERENT. THAN FROM THE PREPROCESSOR INPUT FILE. MAKE SURE EXECUTABLE HAS BEEN RUN SINCE LAST PREPROCESSOR UPDATES. Rerun the ANALYSIS option in the Main Menu to ensure that consistent versions of the input and output are in the data file directory.	**FATAL**
WARNING !!! CURRENT OUTPUT FILE IS OLD. THE PREPROCESSOR FILE HAS NOT BEEN VALIDATED AND THE SOLUTION MODULE HAS NOT BEEN RUN SINCE THE PREPROCESSOR WAS LAST ENTERED. DO YOU WANT TO LOOK AT RESULTS ANYWAY?	**WARNING**

Note: ** - Indicates Wall/Roof Area Name
&&& - File name

13.0 EXAMPLE PROBLEM

The FACEDAP program is used in this example to calculate the blast damage to the building shown on the following page. This example illustrates the use of each basic option in the program. However, the example building does not contain each of the 24 different component types. Input of component properties for each of the 24 different component types is illustrated in Sections 8.5.2.1 through 8.5.2.24. The discussion within this example includes most of the main points in Sections 7.0 through 11.0 of this manual, which describe input of information defining the building and explosive charge and viewing program output which shows the calculated building and component blast damage. Whenever the explanation provided in this example problem is not sufficient, the user should refer to the Table of Contents to find more discussion on a given topic. The User's Manual is sectioned so that the section titles in the Table of Contents refer to each major program option which the user can select.

Section 6.0 of the manual describes the use of keystrokes required to traverse the different types of input and output screens used in the program. Sections 1.0 through 5.0 give the user an overview of the FACEDAP program and describes the installation of the program onto the user's PC. This example problem does not describe the program installation. Section 12.0 of the manual contains a listing and explanation of all error messages. Finally, Section 14.0 of the manual shows drawings of thirteen light industrial buildings. The FACEDAP input files for each of these buildings are included on the FACEDAP program disk. These files must be unarchived from the BUILDING.ARC file as discussed in Chapter 3.0. After these files are analyzed with the FACEDAP program, the program input and output can be printed out as described in Sections 8.10 and 11.4 of this manual and this information can be used along with the drawings and building descriptions in Section 14.0 as additional example problems.

STEP 1 Obtain or draw a general diagram of the building.

A plan view of the Example Building which will be analyzed, with cross sections showing typical components, is shown in the next two figures. This building has a reinforced concrete moment resisting frame with marginally reinforced CMU in-fill walls and a lightweight reinforced concrete roof. The 12 ft high, one-story building, covers 1600 square feet. It is Example Building #2 in Section 14.0. The analysis shown in this example assumes that the frame components will respond to the blast load primarily as individual columns and beams subjected to flexure. As is discussed below, and in Section 8.6.3, buildings with frames should be analyzed twice, first assuming that the frame members respond as individual components as is illustrated in this example, and then assuming the components respond as frames subjected to lateral loading.

STEP 2 Begin the FACEDAP program by typing the word FACEDAP and then pressing the enter or return key while in the directory where the FACEDAP.EXE file resides. All files required by the program must be in the data directory path that is defined in the configuration program FACECFG or in the *Configuration* option under *Utilities* in the main FACEDAP program menu. The user should see Section 3.0 for instructions on how to check and reset the program paths if necessary.

13.0 EXAMPLE PROBLEM

The FACEDAP program is used first in this example to calculate the blast damage to the building shown on the following page, and then it is used to calculate the damage to a single component in the building - the reinforced concrete exterior column in the middle of the long side of the building (Component #20 in the figure on the next page). This example illustrates the use of each basic option in the program. However, the example building does not contain each of the 24 different component types. Input of component properties for each of the 24 different component types is illustrated in Sections 8.5.2.1 through 8.5.2.24. The discussion within this example includes most of the main points in Sections 7.0 through 11.0 of this manual, which describe input of information defining the building and explosive charge and viewing program output which shows the calculated building and component blast damage. Whenever the explanation provided in this example problem is not sufficient, the user should refer to the Table of Contents to find more discussion on a given topic. The User's Manual is sectioned so that the section titles in the Table of Contents refer to each major program option which the user can select.

Section 6.0 of the manual describes the use of keystrokes required to traverse the different types of input and output screens used in the program. Sections 1.0 through 5.0 give the user an overview of the FACEDAP program and describes the installation of the program onto the user's PC. This example problem does not describe the program installation. Section 12.0 of the manual contains a listing and explanation of all error messages. Finally, Section 14.0 of the manual shows drawings of thirteen light industrial buildings. The FACEDAP input files for each of these buildings are included on the FACEDAP program disk. These files must be unarchived from the BUILDING.ZIP file as discussed in Chapter 3.0. The BUILDING.ZIP file also contains the file COMPCHCK.BLG, which contains single component analysis input for a large number of individual components. After these files are analyzed with the FACEDAP program, the program input and output can be printed out as described in Sections 8.10 and 11.4 of this manual and this information can be used along with the drawings and building descriptions in Section 14.0 as additional example problems. The first part of the example problem, where the FACEDAP program is used to analyze a building, follows next. The portion of the example problem where FACEDAP is used to analyze a single component is in Section 13.2.

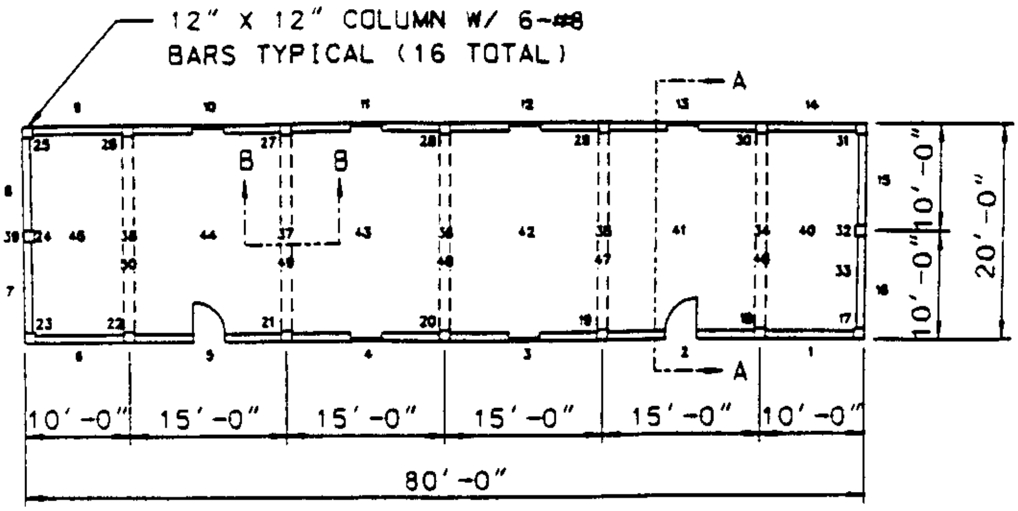
13.1 Building Analysis Example Problem

STEP 1 Obtain or draw a general diagram of the building.

A plan view of the Example Building which will be analyzed, with cross sections showing typical components, is shown in the next two figures. This building has a reinforced concrete moment resisting frame with marginally reinforced CMU in-fill walls and a lightweight reinforced concrete roof. The 12 ft high, one-story building, covers 1600 square feet. It is Example Building #2 in Section 14.0. The analysis shown in this example assumes that the frame components will respond to the blast load primarily as individual columns and beams subjected to flexure. As is discussed below, and in Section 8.6.3, buildings with frames should be analyzed twice, first assuming that the frame members respond as individual components as is illustrated in this example, and then assuming the components respond as frames subjected to lateral loading.

STEP 2 Begin the FACEDAP program by typing the word FACEDAP and then pressing the enter or return key while in the directory where the FACEDAP.EXE file resides. All files required by the program must be in the data directory path that is defined in the configuration program FACECFG or in the *Configuration* option under *Utilities* in the main FACEDAP program menu. The user should see Section 3.0 for instructions on how to check and reset the program paths if necessary.

THIS PAGE INTENTIONALLY LEFT BLANK



ROOF PLAN

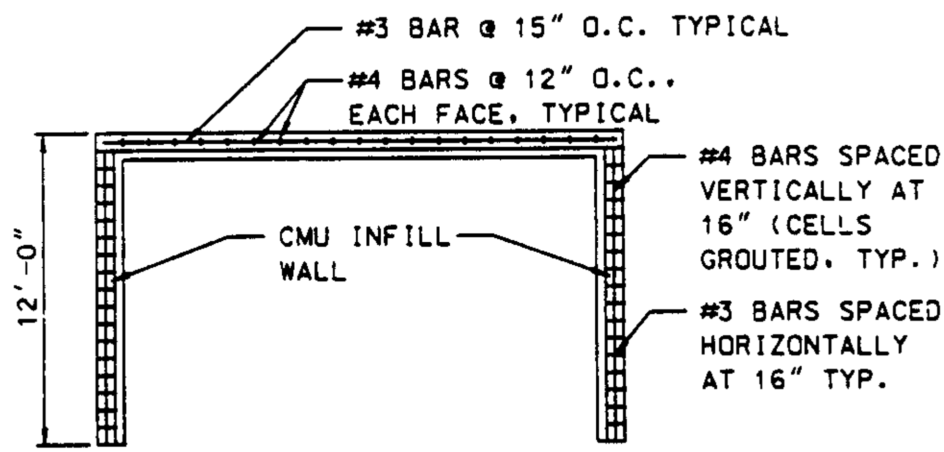
COMPONENTS 1 - 16 ARE WALL PANELS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 17 - 32 ARE EXT. COLUMNS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 33 - 39 ARE REINFORCED CONCRETE BEAMS PROCEDING RIGHT TO LEFT

COMPONENTS 40 - 45 ARE LIGHTWEIGHT ROOF PANELS PROCEDING RIGHT TO LEFT

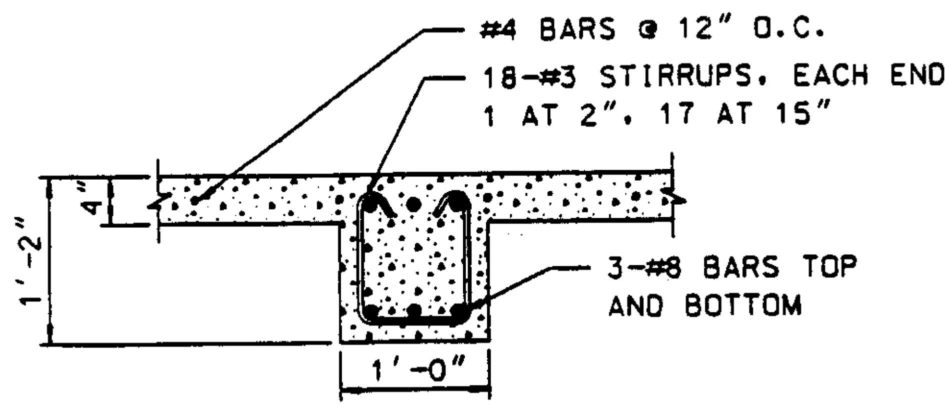
COMPONENTS 46 - 50 ARE CONCRETE FRAMING MEMBERS PROCEDING RIGHT TO LEFT



SECTION A-A

NOTES:
- 4000 PSI CONCRETE STRENGTH
- 60 KSI REINFORCEMENT STRENGTH
- 2000 PSI COMPRESSIVE CMU STRENGTH
- 200 PSI CMU RUPTURE MODULUS

Example Building Plan and Details



SECTION B-B

Example Building Details (Continued)

STEP 3 Assuming the user does not have an existing data file describing the desired building, the *Input Definition* option off the *Preprocessor* option in the main FACEDAP menu should be selected. If a data file for a similar building exists, or if one of the 13 common building types described Section 14.0 is similar to the building of interest, the *Retrieve* option under *File* in the main FACEDAP program can be used to retrieve the similar input file. The input in the similar building can then be modified with the Input Definition option using the method described below.

Step 4 Select the *Input Definition* option under the *Preprocess* option in the main FACEDAP program menu. Then, select the *Building Analysis* option from the *Problem Type* screen which is displayed by the program. The program will display the preprocessor main menu for input of a building analysis problem. The first step in the input definition is to enter or modify the problem title and description using the *Problem Title* option off the main menu of the preprocessor.

The problem is entitled "Example Building #2", the charge weight of 1000 lbs and the standoff of 70 ft are also mentioned in the problem description.

Step 5 Use the F2 key to return to the main preprocessor menu. **The F2 key is typically the escape key off all spreadsheet and form-style input screens in the FACEDAP program.** Select the *Load Definition* option off the main preprocessor menu and enter the charge weight and charge location.

In this example problem, a 1000 lb explosive charge will be considered at a 70 ft standoff from the center of the long sidewall (the South Wall) of the building. The charge location will be assigned the coordinates (0,0,0) in this example. The global coordinates defining the building location will be chosen so that the building will be located 70 ft from the charge as described above.

There are several restrictions on the allowable charge location which are discussed in Section 8.4. The input charge location must be **outside** the building which is analyzed for blast damage and it must not be directly over the building roof. Also, it is recommended that the user only analyzes cases where the scaled standoff between the charge and the building is at least 3.0 $ft/lb^{1/3}$. The scaled standoff is the minimum distance between the charge and the building (in the units of feet) divided by the cube root of the charge weight (in the units of pounds). The validation module of the program will issue a fatal error message, which precludes entry into the Analysis module, if the scaled standoff is less than 1.0 $ft/lb^{1/3}$ or if the charge location is inside the building or above the building roof. A warning message will be issued if the scaled standoff is less than 3.0 $ft/lb^{1/3}$.

Finally, the units of feet and pounds must be used for the *Load Definition* input. A printout of the *Problem Title* and *Load Definition* input follows.

STEP 3 Assuming the user does not have an existing data file describing the desired building, the *Input Definition* option off the *Preprocessor* option in the main FACEDAP menu should be selected. If a data file for a similar building exists, or if one of the 13 common building types described Section 14.0 is similar to the building of interest, the *Retrieve* option under *File* in the main FACEDAP program can be used to retrieve the similar input file. The input in the similar building can then be modified with the Input Definition option using the method described below.

Step 4 Select the *Input Definition* option under the *Preprocess* option in the main FACEDAP program menu. The first step in the input definition is to enter or modify the problem title and description using the *Problem Title* option off the main menu of the preprocessor.

The problem is entitled "Example Building #2", the charge weight of 1000 lbs and the standoff of 70 ft are also mentioned in the problem description.

Step 5 Use the F2 key to return to the main preprocessor menu. **The F2 key is typically the escape key off all spreadsheet and form-style input screens in the FACEDAP program.** Select the *Load Definition* option off the main preprocessor menu and enter the charge weight and charge location.

In this example problem, a 1000 lb explosive charge will be considered at a 70 ft standoff from the center of the long sidewall (the South Wall) of the building. The charge location will be assigned the coordinates (0,0,0) in this example. The global coordinates defining the building location will be chosen so that the building will be located 70 ft from the charge as described above.

There are several restrictions on the allowable charge location which are discussed in Section 8.4. The input charge location must be **outside** the building which is analyzed for blast damage and it must not be directly over the building roof. Also, it is recommended that the user only analyzes cases where the scaled standoff between the charge and the building is at least 3.0 $ft/lb^{1/3}$. The scaled standoff is the minimum distance between the charge and the building (in the units of feet) divided by the cube root of the charge weight (in the units of pounds). The validation module of the program will issue a fatal error message, which precludes entry into the Analysis module, if the scaled standoff is less than 1.0 $ft/lb^{1/3}$ or if the charge location is inside the building or above the building roof. A warning message will be issued if the scaled standoff is less than 3.0 $ft/lb^{1/3}$.

Finally, the units of feet and pounds must be used for the *Load Definition* input. A printout of the *Problem Title* and *Load Definition* input follows.

```
                  PROBLEM DESCRIPTION REPORT
                         EXAMPL2.BLG
                  Example Building No. 2


                   *----------------------*
                   | PROBLEM DESCRIPTION  |
                   *----------------------*

This structure has a reinforced concrete moment resisting frame with
              marginally reinforced CMU in-fill walls.
 Charge weight = 1000 lbs. at a standoff of 70 ft. from South Wall

                   *----------------------*
                   |    LOAD DEFINITION   |
                   *----------------------*

               Charge Weight  :        1000.0  lbs

               Charge Location:
                    X Coordinate           .0  ft
                    Y Coordinate           .0  ft
                    Z Coordinate           .0  ft
```

STEP 6 Examine the building drawing and identify all the components loaded directly by the blast load. These are the components which will by input into the program. Interior columns which directly support the roof should also be analyzed with the program. After these components are identified, they should be grouped by component type. Within each component type, components with identical cross sectional and material properties should be grouped together and each of these subgroups should be given a short, descriptive name. Table 8 lists all the 24 different component types which can be analyzed with the FACEDAP program. These names, called the Component Property Names, will be used to identify the different sets of component properties.

The relevant component properties (primarily cross sectional and material properties) required by the program for each component type are shown in Section 8.5.2.1 through 8.5.2.24. In some cases a property is required, such as total weight, which is dependent on component length. If there is any required component property value which is not common between any two building components, then the two components must be subgrouped separately and given separate Component Property Names. If two components have identical component properties, they are both part of the same subgroup. A listing of the components in the example building follows.

Components in the Example Building

Marginally reinforced CMU in-fill walls, analyzed as two-way reinforced masonry with simple supports.

Reinforced concrete columns, analyzed as exterior reinforced concrete columns, fixed supports, 15' or 10' loaded widths.

Reinforced concrete beams, fixed supports, 15' or 12.5' loaded widths.

Lightweight reinforced concrete roof sections, 4" thick, analyzed as one-way reinforced concrete slabs.

Note: The corner columns of the Example Building are not input into this example problem because the in-plane support provided in both directions by the attached wall panels is considered sufficient to preclude blast damage. The edge beams which bear on the masonry endwalls along their length also have not been input since they are continuously supported by the wall.

STEP 7 Select the *Component Properties* option off the main Preprocessor menu and enter required properties for each separate set of component properties identified in Step 6. These property sets are input into separate spreadsheets for each component type. The input screens for each of the 24 different component types is accessed by first selecting the basic Material Type (concrete, steel, masonry, or wood) of the component off the first submenu of the *Component Properties* option and then selecting the specific Component Type from a displayed list of component types with the selected Material Type. **All input into these Component Property input screens must be in the units shown in the header of each column.**

The *Component Property* input screens for each Component Type are similar to spreadsheets. They contain separate columns for each required input property. The properties identifying each of the subgroups of separate, or unique component property sets identified in Step 6 for a given Component Type are input into separate rows of the Component Property input screens along with the Component Property Name assigned in Step 6. **No two rows should have the exact same input properties since a given set of component properties only needs to be defined once.** The component property set will be assigned later to all components in the building with the given set of properties using the input Component Property Name. When all unique property sets for the selected Component Type have been input, press the F2 key to return to the submenus and select another Component Type until all the unique property sets identified in Step 6 have been entered.

In the *Component Property* input screens, the F1 key can be used to see a listing of all the function keys which are available to expedite spreadsheet input. The F7 key can be used to calculate these input properties which have an asterisk in the column header. This calculation uses input values in columns to the left of the column of interest and default formulas programmed into the FACEDAP program. The default formulas are shown for each Component

Type in Sections 8.5.2.1 through 8.5.2.24. Default values are also automatically written to the input screens for some required properties. Any user input will overwrite these values. Input of component properties is discussed in more detail in Section 8.5.2. This discussion should be read carefully prior to *Component Property* input. It contains information describing inputs such as weighting factors, tension membrane and arching response, etc. which may not be self evident to the user. The user can rely on the column specific help messages which appear at the bottom of each input screen for some additional guidance. A sample of the *Component Property* input for the Example Building follows showing the two reinforced concrete exterior column property sets which were input. The user can print out all the *Component Property* input by retrieving the file EXAMPL2.BLG, which is on the FACEDAP program disk (see Section 3.0 for instructions on copying off the program disk), and using the Print Report option in the Preprocessor to print the input. This option is explained in Section 8.10.

COMPONENTS PROPERTIES REPORT 1 OF 2
R/C Exterior Column
EXAMPL2.BLG
Example Building No. 2

Component Property Name	Weighting Factor	Column Width (in)	Column Thickness (in)	Loaded Width (ft)	Total Weight (lb)	Concrete Compressive Strength (psi)
ECOL 1	4.00	12.00	12.00	10.00	12000.	4000.
ECOL 2	4.00	12.00	12.00	15.00	18000.	4000.

COMPONENTS PROPERTIES REPORT 2 OF 2
R/C Exterior Column
EXAMPL2.BLG
Example Building No. 2

Component Property Name	Steel Yield Strength (psi)	Depth to Tensile Steel (in)*	Area of Tensile Steel (in^2)*	Moment of Inertia (in^4)*	Boundary Condition
ECOL 1	60000.	10.00	2.37	1150.	FIXED
ECOL 2	60000.	10.00	2.37	1150.	FIXED

STEP 8 Examine the building geometry and divide the building into convenient wall and roof areas.

The Example Building is divided into large, planar four-sided wall/roof areas, defined by four corner points, with a minimum of one area per wall and one area in the roof. Within this restriction it is generally advisable to use a minimum number of wall/roof areas for the sake of simplicity. Other restrictions are listed in Section 8.6.1. Since no graphical presentation of input is available, it is recommended that each wall/roof area should be sketched as shown in step 9.

STEP 9 Label each corner of each Wall/Roof area with its global coordinates and assign numbers (1 through 4) to each corner point which will be used to define the local coordinate system in each area. The following figure shows how this step has been carried out for the Example Building.

Since the location (0,0,0) was chosen as the charge location and a 70 ft standoff is desired to the center of the south sidewall of the building, the global coordinates of the building have been chosen as shown in the following figure. **The global X-Y plane must lie in the plane of the ground and the global Z direction must be positive upward from the ground.** It is recommended that the global X and Y axes be chosen to coincide with two perpendicular walls in the building whenever possible. All coordinates must be entered into the program in the units of feet. The global coordinates of the building corners should be used to determine the global coordinates of the corners of each user defined wall/roof area.

The numbers 1 through 4 which are assigned to the corner points of each wall/roof area are very important because they define the local coordinate system within each wall/roof area used for component geometry input. These numbers show the order in which the four corner points will be input into the FACEDAP program. Since all wall/roof areas must be planar, only two local coordinates are necessary to define individual building components within each wall/roof area. Since all building components must lie within a wall/roof area, all components can be defined in terms of the two local coordinates rather than three global coordinates. **The local x direction extends from the first input wall/roof area corner point to the second input corner point, positive towards Corner No. 2. Also, the first corner point becomes (0,0) in the wall/roof local coordinate system. The positive local y direction is perpendicular to the local x direction and extends from the first input corner point towards the fourth input corner point.** In the sketches of each wall/roof area which follow, the corner points labeled No. 1 and No. 2 have been chosen so that they define the local x direction along the bottom edge each wall area and along the front side (nearest charge) of the roof area. The local y direction in each wall area is positive straight upward from the ground. The wall/roof area corners have also been numbered so that the local x and y axes are parallel to each other and positive in the same direction of parallel wall areas. This is usually a good approach for setting up a set of simple, consistent local coordinate systems in the wall/roof areas of a typical rectangular building.

The corner points must be numbered in a sequential order around the perimeter of the wall/roof area in either a clockwise or counterclockwise manner. The points numbered 1 and 3 must be at opposite corners of the wall/roof areas and the same is true for the points numbered 2 and 4.

STEP 10 Select the *Building Geometry* option off the main Preprocessor menu to begin the building geometry input. This input defines the location of all the components within the building relative to each other and relative to the explosive charge. Also, the Component Property Name of the set of component properties that apply for each component are assigned during *Building Geometry* input. After the *Building Geometry* option is selected, the main Building Geometry menu is displayed on the screen showing the three types of building geometry input: 1) Wall/Roof Area Definition; 2) Component Geometry Definition; and 3) Dependencies. These should be selected in the sequence listed above and all input should be completed into one option before the next option is selected. Therefore, the user should select the *Wall/Roof Area Definition* option to initiate building geometry definition. This will cause a spreadsheet input screen to be displayed. Input the global coordinates of the four corner points for each wall/roof area into the spreadsheet in the order of the numbers assigned to each corner point in Step 9 and input a short descriptive name. Each wall/roof area should be entered into a separate row of the spreadsheet. Use the F2 key to exit the spreadsheet and return to the main Building Geometry menu.

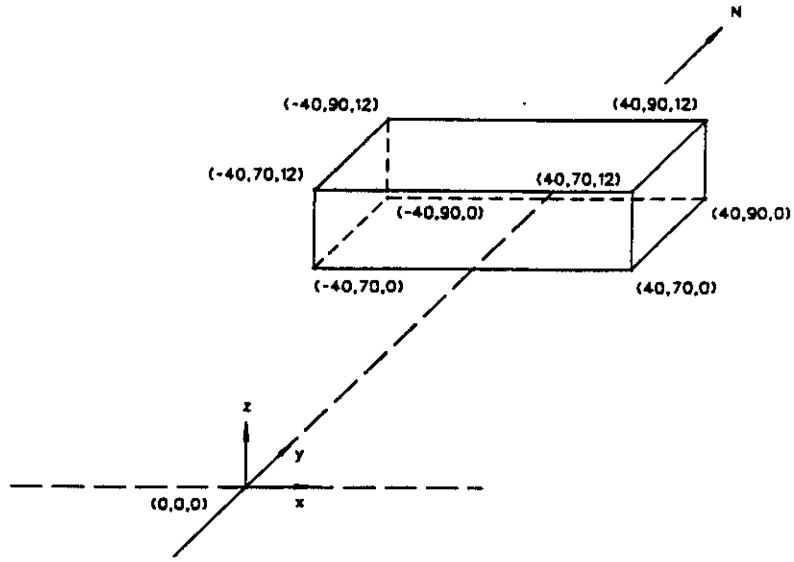
The global coordinates of the four corners of each wall/roof area are input sequentially around the perimeter of the area in the order shown in the following figure. This ordering defines the local x and y axis as shown in the figure for each wall/roof area. The name assigned to each wall/roof area is used by both the Preprocessor and Postprocessor to refer to the given area. A printout of the wall/roof area input for the Example Building follows.

DAMAGED BUILDING COMPONENTS REPORT
WALL/ROOF AREA "ROOF"
EXAMPL2.BLG
Example Building No. 2

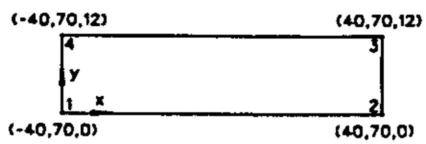
Component Type	Percent Damage (%)	Component Local Coordinates x1 (ft)	y1 (ft)	x2 (ft)	y2 (ft)	Repair or Replace	P-i Diagram Terms pbar	ibar
RC1WI	60	.0	5.0	10.0	5.0	1	5.1	3.4
RC1WI	30	.0	15.0	10.0	15.0	0	4.1	3.1
RCBMI	30	10.0	.0	10.0	20.0	0	3.6	2.7
RC1WI	60	10.0	5.0	25.0	5.0	1	12.8	3.5
RC1WI	60	10.0	15.0	25.0	15.0	1	10.2	3.2
RCBMI	30	25.0	.0	25.0	20.0	0	4.8	3.1
RC1WI	60	25.0	5.0	40.0	5.0	1	13.8	3.6
RC1WI	60	25.0	15.0	40.0	15.0	1	10.8	3.3
RCBMI	30	40.0	.0	40.0	20.0	0	5.0	3.1
RC1WI	60	40.0	5.0	55.0	5.0	1	13.8	3.6
RC1WI	60	40.0	15.0	55.0	15.0	1	10.8	3.3
RCBMI	30	55.0	.0	55.0	20.0	0	4.8	3.1
RC1WI	60	55.0	5.0	70.0	5.0	1	12.8	3.5
RC1WI	60	55.0	15.0	70.0	15.0	1	10.2	3.2
RCBMI	30	70.0	.0	70.0	20.0	0	3.6	2.7
RC1WI	60	70.0	5.0	80.0	5.0	1	5.1	3.4
RC1WI	30	70.0	15.0	80.0	15.0	0	4.1	3.1

BLAST LOAD REPORT
WALL/ROOF AREA "SOUTH WALL"
EXAMPL2.BLG
Example Building No. 2

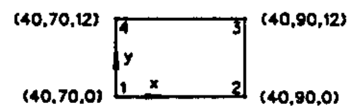
Component Type	Blast Load Peak Pressure (psi)	Impulse (psi-ms)	End or Opposite Corner Points in Local Wall Area Coordinates x1 (ft)	y1 (ft)	x2 (ft)	y2 (ft)
MAR2WI	43.5	229.3	.0	.0	10.0	12.0
RCECI	46.7	236.8	10.0	.0	10.0	12.0
MAR2WI	51.3	246.9	10.0	.0	25.0	12.0
RCECI	55.1	254.8	25.0	.0	25.0	12.0
MAR2WI	57.6	259.9	25.0	.0	40.0	12.0
RCECI	58.5	261.7	40.0	.0	40.0	12.0
MAR2WI	57.6	259.9	40.0	.0	55.0	12.0
RCECI	55.1	254.8	55.0	.0	55.0	12.0
MAR2WI	51.3	246.9	55.0	.0	70.0	12.0
RCECI	46.7	236.8	70.0	.0	70.0	12.0
MAR2WI	43.5	229.3	70.0	.0	80.0	12.0



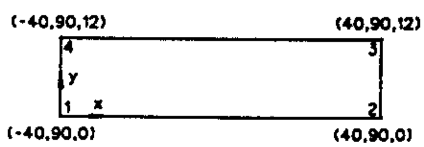
THREE-DIMENSIONAL VIEW OF BUILDING IN
GLOBAL COORDINATE SYSTEM



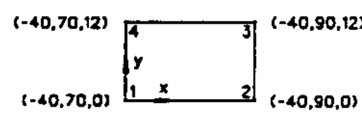
Wall/Roof Area 1 (South Wall)



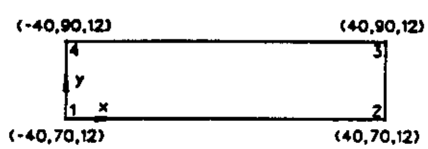
Wall/Roof Area 2 (East Wall)



Wall/Roof Area 3 (North Wall)



Wall/Roof Area 4 (West Wall)



Wall/Roof Area 5 (Roof)

Example Building Wall/Roof Area

STEP 11 Select the *Component Geometry Definitions* option off the main Building Geometry menu. This will cause a listing of the names of all the wall/roof areas defined in Step 10 to be displayed on the screen as well as the Frame option. The user should sequentially select each of the wall/roof areas, input information defining all the components located within the selected wall/roof area, and then press the F2 key to return to the list of all wall/roof areas again. This should be continued until all building components have been entered into all wall/roof areas and frames in the building have been defined. Each component should be defined only within one wall/roof area. Components along wall/roof area boundaries can be input into either area. Reinforced concrete frames and steel frames typically do not lie within a single wall/roof area and therefore all frames must be defined using the Frame option shown along with all the named wall/roof areas. In some cases moment resisting frames are entered as complete frame components and in other cases the columns and beams which make up the frame are entered as individual column and beam components as discussed below.

When each wall/roof area is selected, the *Component Definition* spreadsheet is displayed. This spreadsheet has columns for each required input. The first eight columns information for a single component as discussed in this paragraph. The last four columns read information that the Preprocessor uses to generate additional components if the this capability is invoked by the user. These columns are discussed in subsequent paragraphs. All **one-way slab, one-way masonry wall components, beam, exterior column, wood wall, and wood roof components** should be defined in the first eight columns of the *Component Definition* spreadsheet terms of: 1) the component material, 2) the component type, 3) the Component Property Name of the property set defining the properties of the component, and 4) the LOCAL x and LOCAL y coordinates of the two endpoints of the components. **In the case of one-way slabs, one-way masonry walls components, wood walls and wood roofs,** the end points which define the span of the component and lie at the center of the component should be input. Long one-way walls or slabs should be divided into separate components which have a relatively uniform blast load over their full width. Also, the component end points should match those of any component which supports, or is supported by the component. Usually ten to twenty foot spacing between adjacent components will cause a nearly uniform blast load over each component. The spacing between these one-way components is not dependent on the section width, which is a separate component property.

All **two-way components** should be defined in terms of items 1 through 3 above and, 4) the LOCAL x and LOCAL y coordinates of two OPPOSITE corner points of the components. All **interior column components** should be defined in terms of items 1 through 3 above and, 4) the LOCAL x and LOCAL y coordinates of the top endpoint of the column where it intersects the roof, or the center point of the roof area supported by the column.

The input of a group of identical components spaced relative to each other at uniform spacing within a wall/roof area is much easier if the capability of the FACEDAP code to automatically generate components is used. If there are a number of identical components within a wall/roof area at uniform spacing then, 1) locate the component CLOSEST to (0,0) in the wall/roof local coordinate system, this becomes the "master component", 2) enter information for this component into the first eight columns of the *Component Definition* spreadsheet as explained in the previous paragraphs, 3) use the space bar key in the ninth column to indicate that the component is a master component which will be used for component generation, 4) enter additional information into the last three columns of the *Component Definition* spreadsheet including the uniform center to center spacing between the similar components, the direction of spacing between components (local x or local y direction), and the number of additional components (NOT including the master component which is being entered) that should be generated. Then, the user must make sure the cursor is still on the row with the master component just entered and press the F9 key. This causes the Preprocessor to generate the additional components. If a keystroke (any keystroke) is selected followed by the F10 key, the *Master Component* spreadsheet is displayed which shows all the components generated from the given master component. This may be edited by the user in the same manner as the previous component definition spreadsheet. If the components are ever regenerated, all edits will be overwritten. The F2 key returns the user to the *Component Definition* spreadsheet. The F10 key can be used any time to access the *Master Component* spreadsheet showing the components generated from the master component in the row of the *Component Definition* spreadsheet where the cursor is when the F10 key is pressed. If any changes are made to the master component, the user should either regenerate all the previously generated components with the F9 key, or make the changes to each generated component on the *Master Component* spreadsheet.

All **frame components** must be entered into the FRAME component definition spreadsheet. Frame components should be defined in terms of items 2 and 3 above and, 4) the name of the wall area closest to the blast which contains an exterior column of the frame (the blastward wall), and 5) the LOCAL x and LOCAL y coordinates of the ground level end point of the exterior frame column in the blastward wall. The x and y coordinates of the column base point will be in the local coordinate system of the blastward wall. In the case of a building with frames, the user should make two separate runs with the FACEDAP code considering the columns and beams which make up the frame in the first run as separate exterior and interior columns and roof beams and then, in second run, DELETING these frame components from the wall/roof component definition input spreadsheets (using the Control D keystroke) and defining the frames in the FRAME component definition spreadsheet. It is very important that the user does not double define components both as exterior columns and roof beams and by defining the frame which includes these columns and beams. This is discussed further in Section 8.6.3.2. The input in this example problem is that for the run where the frame components are considered as separate columns and beams responding in flexure to the blast load. EXAMPL2A.BLG on the FACEDAP program disk contains the input for the case where the column and beam components which make up the frames are deleted from the input deck as individual components and the moment resisting frames are input as frame components. A sample of the printout of the *Component Property* input for the Example Building follow. The user can printout the all the *Component Property* input by reading the file EXAMPL2.BLG off the FACEDAP program disk (see Section 3.0 for instructions on copying off the program disk) and using the Print Report option in the Preprocessor to print the input. This option is explained in Section 8.10.

A sample of the *Component Geometry* input for the Example Building follows showing the components defined in the South Wall. The first three components shown are master components which were used to generate the remainder of the components. The user can print out all the *Component Geometry* input by retrieving the file EXAMPL2.BLG, which is on the FACEDAP program disk (see Section 3.0 for instructions on copying off the program disk), and using the Print Report option in the Preprocessor to print the input. This option is explained in Section 8.10.

COMPONENT GEOMETRY REPORT 1 OF 2
WALL/ROOF AREA "SOUTH WALL"
EXAMPL2.BLG
Example Building No. 2

Item #	Component Material Type	Component Type	Component Property Name	Component Id. Number
1	MASONRY	MAR2WI	MWAL 1	01.001.00
2	CONCRETE	RCECI	ECOL 2	01.004.00
3	MASONRY	MAR2WI	MWAL 1	01.003.00
4	CONCRETE	RCECI	ECOL 2	01.004.01
5	MASONRY	MAR2WI	MWAL 1	01.003.01
6	CONCRETE	RCECI	ECOL 2	01.004.02
7	MASONRY	MAR2WI	MWAL 1	01.003.02
8	CONCRETE	RCECI	ECOL 2	01.004.03
9	MASONRY	MAR2WI	MWAL 1	01.003.03
10	CONCRETE	RCECI	ECOL 2	01.004.04
11	MASONRY	MAR2WI	MWAL 1	01.001.01

COMPONENT GEOMETRY REPORT 2 OF 2
WALL/ROOF AREA "SOUTH WALL" REPORT 2 OF 2
EXAMPL2.BLG
Example Building No. 2

Item #	End or Opposite Corner Points in Local Wall Area Coordinates x1 (ft)	y1 (ft)	x2 (ft)	y2 (ft)	Master Component for Comp. Generation	C/C Spacing (ft)	Local Dir. of C/C Spacing	# of Comp. to be Gen.
1	.0	.0	10.0	12.0	Yes	70.0	Local x	1
2	10.0	.0	10.0	12.0	Yes	15.0	Local x	4
3	10.0	.0	25.0	12.0	Yes	15.0	Local x	3
4	25.0	.0	25.0	12.0	-	-	-	-
5	25.0	.0	40.0	12.0	-	-	-	-
6	40.0	.0	40.0	12.0	-	-	-	-
7	40.0	.0	55.0	12.0	-	-	-	-
8	55.0	.0	55.0	12.0	-	-	-	-
9	55.0	.0	70.0	12.0	-	-	-	-
10	70.0	.0	70.0	12.0	-	-	-	-
11	70.0	.0	80.0	12.0	-	-	-	-

STEP 12 After all the *Component Geometries* have been defined, select the *Dependencies* option from the main *Building Geometry* menu. A screen will be displayed with the names of all wall/roof areas defined by the user and the word "Frame". This screen is used to view and generate the component dependencies. The component dependencies define how the building components supports, and is supported by, other components. They are used to calculate any cascading failure, or secondary failure of components due to loss of support caused by 100% damage to supporting components as explained in Section 8.6.4. The wall/roof areas in the menu should be selected one-by-one. After a wall/roof area has been selected one of two possibilities will occur. If there are no preexisting dependencies for the selected wall/roof area, the Preprocessor will automatically generate the dependencies of each component in the selected wall/roof area. After the components are generated, the *Dependencies* spreadsheet showing each component in the wall/roof area (called the Dependent Component in the spreadsheet) and the ID Numbers of the building components it is supported by (called Independent Components) automatically comes to the screen. Some components are not supported by any other component and therefore they do not have any Independent Components.

The other possibility is that dependencies have been previously generated for the selected wall/roof area. The Preprocessor does not automatically overwrite existing dependencies since these may include user edits. Therefore, the previously generated dependencies are displayed and the user must use the F9 key to generate a new set of dependencies which take into account the effects of any new components or component locations input since the file was last saved. Selection of the Frame option from the menu of wall/roof areas results in a screen which allows the user to either turn off all frame dependencies or to allow the FACEDAP program to calculate frame dependencies. For typical situations, it is recommended that the program be allowed to calculate frame dependencies. This is the default.

Whenever dependencies are generated by the Preprocessor, the user should check that each Dependent Component is supported by the listed Independent Components. Also, the user should check that no components which provide support are missing. More information on how the Independent Components are determined is given in Section 8.6.4.1.1. The F7 key should be used to find out more information about each Independent Component. Information on the wall/roof area, local coordinates, and component type of each Independent Component is provided on the screen if the user positions the cursor on the Independent Component ID Number of interest and presses the F7 key. The local coordinates which are displayed show the local end or corner points of the Independent Component in the local coordinate system of the wall/roof area where this component was defined. The user can delete an Independent Component ID number from this spreadsheet screen as shown in Table 7 in Section 6.5. The user can also enter an Independent Component ID number or overwrite an Independent Component ID number. Finally, the user can see the full component type name of the Dependent Component (which is abbreviated in the first column of the *Dependencies* spreadsheet) by positioning the cursor over the abbreviated name and pressing the space bar key.

A sample of the *Dependencies* input for the Example Building follows showing the dependencies calculated for components in the South Wall. No consideration of the roof support is calculated by the algorithm used by the Preprocessor to calculate dependencies as discussed in Section 8.6.4.1.1. The user can print out all the *Dependencies* input by retrieving the file EXAMPL2.BLG, which is on the FACEDAP program disk (see Section 3.0 for instructions on copying off the program disk), and using the Print Report option in the Preprocessor to print the input. This option is explained in Section 8.10.

DEPENDENCY REPORT 1 OF 1
WALL/ROOF AREA "SOUTH WALL"
EXAMPL2.BLG
Example Building No. 2

Dependent Component Type	End or Opposite Corner Points in Local Wall Area Coordinates x1 (ft)	y1 (ft)	x2 (ft)	y2 (ft)	Dependent Component Id. Number	Supporting Component ID Number(s) (Independent Comp. ID Number(s))	
MAR2WI	.0	.0	10.0	12.0	01.001.00	01.004.00	04.001.00
RCECI	10.0	.0	10.0	12.0	01.004.00		
MAR2WI	10.0	.0	25.0	12.0	01.003.00	01.004.00	01.004.01
RCECI	25.0	.0	25.0	12.0	01.004.01		
MAR2WI	25.0	.0	40.0	12.0	01.003.01	01.004.01	01.004.02
RCECI	40.0	.0	40.0	12.0	01.004.02		
MAR2WI	40.0	.0	55.0	12.0	01.003.02	01.004.02	01.004.03
RCECI	55.0	.0	55.0	12.0	01.004.03		
MAR2WI	55.0	.0	70.0	12.0	01.003.03	01.004.03	01.004.04
RCECI	70.0	.0	70.0	12.0	01.004.04		
MAR2WI	70.0	.0	80.0	12.0	01.001.01	01.004.04	02.001.00

STEP 13 This completes building geometry input. Return to the main Preprocessor menu and save the input information with the *Save Input* option. The user must input the save file name without an extension and the Preprocessor automatically assigns the .BLG extension to the saved input file. The user can print the input as explained in Section 8.10.

STEP 14 Return to the main FACEDAP menu and select the *Input Validation* option under *Preprocess*.

This option checks the input in a given input file for errors before the input is submitted to the Analysis module of the FACEDAP code. An input deck cannot be analyzed unless the *Input Validation* option has been exercised and no fatal errors were located. After it has completed its validation process, the FACEDAP code prints a report to the screen which summarizes all errors and warnings. Instructions for moving within the review window appear at the bottom of the window. The escape key returns the user to the main FACEDAP program menu after the user has answered a dialog message which instructs the program to either print or not to print the error summary. If there are errors the user should return to the *Input Definition* option under *Preprocess* in the main program menu, make the necessary corrections, and repeat the validation.

STEP 15 After the input has been validated with no errors, the user should select the *Analysis* option of the main FACEDAP menu. This will cause the Analysis module to determine the blast damage caused to the input building from the input explosive charge. The Analysis module creates an output deck which can be read by the Postprocessor.

STEP 16 After the analysis is complete, the user should select the *Postprocessor* option in the main FACEDAP menu. This will cause a menu to appear with the following options: *Building Damage; Component Damage; Blast Load Information;* and *Print Reports*. The *Building Damage* option displays the calculated values of four building damage parameters including the overall percentage of building damage caused by the blast loads. See Section 11.1 for an explanation of these parameters. Selection of the *Component Damage* and *Blast Load Information* will cause additional submenus to appear. If the *Component Damage* option is selected, these submenus allow the user to see either the most damaged components in the entire building or to see the damage level calculated for each building component. If *Blast Load Information* option is selected, these submenus allow the user to see the blast load calculated on each building component. Both the *Component Damage* and *Blast Load Information* options display blast damage or blast loads on each building component in separate spreadsheets for which contain information on components within each user defined wall/roof area. The *Print Reports* option, which is explained in Section 11.4, can be used to print out all the output displayed by the other options.

A sample of the Postprocessor output for the Example Building follows. The user can print out all the Postprocessor output by retrieving the file EXAMPL2.BLG, which is on the FACEDAP program disk (see Section 3.0 for instructions on copying off the program disk), validating and analyzing the input, and using the Print Report option in the Postprocessor to print the output.

BUILDING DAMAGE SUMMARY REPORT
EXAMPL2.BLG
Example Building No. 2

```
*----------------------*
| PROBLEM DESCRIPTION  |
*----------------------*
```

This structure has a reinforced concrete moment resisting frame with marginally reinforced CMU in-fill walls.
Charge weight = 1000 lbs. at a standoff of 70 ft. from South Wall

```
*----------------------*
|   LOAD DEFINITION    |
*----------------------*

Charge Weight   :        1000.0  lbs

Charge Location:
     X Coordinate            .0  ft
     Y Coordinate            .0  ft
     Z Coordinate            .0  ft

*----------------------------------------------*
| BUILDING DAMAGE INFORMATION BASED ON SUMMED  |
|    COMPONENT DAMAGE AND WEIGHTING FACTORS    |
*----------------------------------------------*

Percent Building Damage                          : 49. %
Replacement Factor                               : 30. %
Percent of Building Floor Reuseable w/o Repair   : 87. %

*----------------------------------------------*
|      LEVEL OF PROTECTION BASED ON OVERALL    |
|         PERCENTAGE OF BUILDING DAMAGE        |
*----------------------------------------------*

Building Level of Protection                     : Medium
```

```
                BUILDING DAMAGE SUMMARY REPORT
                        EXAMPL2.BLG
                  Example Building No. 2


                  *----------------------*
                  | PROBLEM DESCRIPTION |
                  *----------------------*

This structure has a reinforced concrete moment resisting frame with
            marginally reinforced CMU in-fill walls.
 Charge weight = 1000 lbs. at a standoff of 70 ft. from South Wall

                  *----------------------*
                  |   LOAD DEFINITION    |
                  *----------------------*

             Charge Weight  :       1000.0  lbs

             Charge Location:
                  X Coordinate         .0  ft
                  Y Coordinate         .0  ft
                  Z Coordinate         .0  ft

        *----------------------------------------------*
        | BUILDING DAMAGE INFORMATION BASED ON SUMMED  |
        |   COMPONENT DAMAGE AND WEIGHTING FACTORS     |
        *----------------------------------------------*

       Percent Building Damage                       : 49. %
       Replacement Factor                            : 30. %
       Percent of Building Floor Reuseable w/o Repair : 87. %

        *----------------------------------------------*
        |      LEVEL OF PROTECTION BASED ON OVERALL    |
        |          PERCENTAGE OF BUILDING DAMAGE       |
        *----------------------------------------------*

       Building Level of Protection                  : Medium
```

13.2 Single Component Analysis Example Problem

STEP 1 If the user is starting with this part of the example problem, see Steps 2 and 3 from Section 13.1 to begin running the FACEDAP program. The file COMPCHCK.BLG in the archived file BUILDING.ZIP on the FACEDAP program disk contains the input described in this example.

STEP 2 Select the *Input Definition* option under the *Preprocess* option in the main FACEDAP program menu. Then, select the *Single Component Analysis* option from the *Problem Type* screen which is displayed by the program.

STEP 3 The program will display the Preprocessor main menu for single component analysis. The first step in the input definition is to enter or modify the problem title and description using the *Problem Title* option off the main menu.

STEP 4 Use the F2 key to return to the main preprocessor menu. **The F2 key is typically the escape key off all spreadsheet and form-style input screens in the FACEDAP program.** Select the *Load Definition* option off the main preprocessor menu and enter the charge weight, standoff between the charge and center of the component to be analyzed, and the type of blast pressure (i.e., free-field or reflected).

In this example problem, a 1000 lb explosive charge will be considered at a 70 ft standoff from the center of the component of interest. This component is the exterior reinforced concrete column in the middle of the long sidewall of the building shown in Section 13.1 (Component #20 in the illustration of the building). The component is in the sidewall facing the charge so that it will be loaded by a reflected blast pressure.

There are several restrictions on the allowable charge location which are discussed in Section 8.0.2.3. It is recommended that the user only analyzes cases where the scaled standoff between the charge and the building is at least 3.0 $ft/lb^{1/3}$ and less than 100 $ft/lb^{1/3}$. The scaled standoff is the minimum distance between the charge and the building (in the units of feet) divided by the cube root of the charge weight (in the units of pounds). The validation module of the program will issue a fatal error message, which precludes entry into the Analysis module, if the scaled standoff is less than 1.0 $ft/lb^{1/3}$ or if the charge location is inside the building or above the building roof. A warning message will be issued if the scaled standoff is less than 3.0 $ft/lb^{1/3}$.

Finally, the units of feet and pounds must be used for the *Load Definition* input. A printout of the *Load Definition and Problem Title* input follows.

```
                    PROBLEM DESCRIPTION REPORT
                          COMPCHCK.BLG
     Example File with Single Component Property Input for All Components


                      *----------------------*
                      | PROBLEM DESCRIPTION  |
                      *----------------------*

       Current Standoff = 70 ft, Current Charge Weight = 1000 lbs
     Current Selected Component for Analysis is Exterior R/C Column
                      component named 'ECOL2'

                      *----------------------*
                      |   LOAD DEFINITION    |
                      *----------------------*

              Charge Weight          :        1000.0  lbs

              Charge Standoff        :          70.0  ft

              Type of Blast Pressure:  Free-Field
```

STEP 5 Select the *Component Properties* option off the main Preprocessor menu and enter required properties for the component of interest (the reinforced concrete exterior column). These property sets are input into separate spreadsheets for each component type. The input screens for each of the 24 different component types is accessed by first selecting the basic Material Type (i.e., concrete for this case) of the component off the first submenu of the *Component Properties* option and then selecting the specific Component Type (i.e. exterior R/C column) from a displayed list of component types constructed from the selected Material Type. This causes the screen for input of Exterior R/C Concrete Columns to be displayed. **All input into these Component Property input screens must be in the units shown in the header of each column.** When all property sets for the selected Component Type have been input, press the F2 key to return to the submenus. The user may define multiple components, assigning each component a name in the first column of the Component Property spreadsheets. The *Component Property* input for the reinforced concrete exterior column of interest follows.

COMPONENTS PROPERTIES REPORT 1 OF 2
R/C Exterior Column
COMPCHCK.BLG
Example File with Single Component Property Input for All Components

Component Property Name	Column Height (ft)	Column Width (in)	Column Thickness (in)	Loaded Width (ft)	Total Weight (lb)	Concrete Compressive Strength (psi)
ECOL 2	12.00	12.00	12.00	15.00	18000.	4000.

COMPONENTS PROPERTIES REPORT 2 OF 2
R/C Exterior Column
COMPCHCK.BLG
Example File with Single Component Property Input for All Components

Component Property Name	Steel Yield Strength (psi)	Depth to Tensile Steel (in)*	Area of Tensile Steel (in^2)*	Moment of Inertia (in^4)*	Boundary Condition
ECOL 2	60000.	10.0000	2.3700	1150.0	FIXED

STEP 6 Select the *Component Selection* option off the main Preprocessor menu and select the component to be analyzed by making appropriate choices from the pop-up menus associated with each input on the screen. Position the cursor on each input, starting from the top of the screen, press the space bar to activate the pop-up menus, and pick the appropriate material type, component type, and component name, respectively, for each of the three required inputs. It is important that all three inputs are always compatible with each other. The *Component Selection* input for the reinforced concrete exterior column of interest follows.

```
                    COMPONENT SELECTION REPORT
                         COMPCHCK.BLG
Example File with Single Component Property Input for All Components


          Material Name :  CONCRETE

          Component Type:  R/C Exterior Column

          Component Name:  ECOL 2
```

STEP 7 Save the input using the *Save Input File* option off the main preprocessor menu. The *Save File* name must begin with the four letter string 'COMP'. Then, return to the main FACEDAP menu and select the *Input Validation* option under *Preprocess*. For a brief explanation of the Validation procedure, see Step 14 in Section 13.1.

STEP 8 After the input has been validated with no errors, the user should select the *Analysis* option of the main FACEDAP menu. This will cause the Analysis module to determine the blast damage caused to the component selected in Step 6 from the input explosive charge. The Analysis module creates an output deck which can be read by the Postprocessor.

STEP 9 After the analysis is complete, the user should select the *Postprocessor* option in the main FACEDAP menu. This will cause a menu to appear with the following options: *Component Damage; Blast Load Information;* and *Print Reports*. The *Component Damage* option displays the calculated values of the component damage level and level of protection, the Pbar and Ibar values, and the blast load on the center point of the component. The *Blast Load Information* option simply displays a screen showing the user the input blast load parameters. The *Print Reports* option can be used to print out all the output displayed by the other options. The Postprocessor output for this example problem follows.

SINGLE COMPONENT DAMAGE SUMMARY REPORT
COMPCHCK.BLG
Example File with Single Component Property Input for All Components

```
*---------------------*
| PROBLEM DESCRIPTION |
*---------------------*
```

Current Standoff = 70 ft, Current Charge Weight = 1000 lbs
Current Selected Component for Analysis is Exterior R/C Column
component named 'ECOL2'

```
*---------------------*
|   LOAD DEFINITION   |
*---------------------*
```

Charge Weight	:	1000.00	lbs
Charge Standoff	:	70.00	ft
Type of Blast Pressure	:	Free-Field	

```
*--------------------------------------------*
|          SINGLE COMPONENT DAMAGE           |
*--------------------------------------------*
```

Component Damage Level	:	30. %
Component Level of Protection (L.O.P.)	:	Medium
Pbar Term	:	3.026
Ibar Term	:	2.481
Peak Plast Pressure	:	19.779 psi
Blast Impulse	:	110.692 psi-ms

14.0 THIRTEEN EXAMPLE BUILDINGS

Thirteen building types which are considered to be representative of common industrial facilities are presented graphically and summarized on the following pages. A summary of the buildings and the construction types which they represent is shown below in Table 24. These buildings have been input into the input files EXAMPL1.BLG through EXAMPL13.BLG on the FACEDAP program disk. See Section 3.0 for instructions on how to copy these files off the disk. These files correspond to the Building Numbers 1 through 13 in Table 24. The files EXAMPL2A.BLG and EXAMPL7A.BLG contain the input for Buildings No. 2 and No. 7 for the analyses where the frames are considered as frame components. The files EXAMPL2.BLG and EXAMPL7.BLG contain the input for the analyses of these buildings where the frame components are input as individual beam and column components responding in flexure, or buckling, to the applied blast load. This type of input for buildings with moment resisting frames is discussed in Section 8.6.3. The doors and windows of the buildings are not analyzed with the FACEDAP program. If only a rough estimate of door and window damage is desired, the doors can be assumed to fail if they are loaded by a peak blast pressure greater than 2 psi and windows can be assumed to fail at peak pressures above 1 psi. Otherwise, a more detailed analysis is required using other analysis methods.

The file COMPCHCK.BLG on the BUILDING.ZIP file on the FACEDAP program disk contains single component analysis input for a large number of components.

Table 23. A Summary Description of the Thirteen Building Types

Building No.	Page No.	Description
1	170	One-story, large (> 6000 ft^2), reinforced concrete building.
2	173	One-story, small (< 6000 ft^2), reinforced concrete and reinforced masonry building with moment resisting reinforced concrete frames.
3	176	One-story, small (< 6000 ft^2) unreinforced masonry building.
4	179	One-story, small (< 6000 ft^2), unreinforced clay brick building.
5	182	One-story, small (< 6000 ft^2), metal stud wall building.
6	185	Two-story, small (< 6000 ft^2), reinforced concrete building.
7	188	One-story, small (< 6000 ft^2), pre-engineered (Butler® type) building with steel moment resisting frames.
8	191	One-story, large (> 6000 ft^2) metal stud wall building.
9	194	Two-story, small (< 6000 ft^2, timber building.
10	196	One-story, large (> 6000 ft^2), tilt-up reinforced concrete building.
11	198	One-story, large (> 6000 ft^2), heavy timber building.
12	201	Two-story, large (> 6000 ft^2), steel frame building.
13	204	One-story, large (> 6000 ft^2), prestressed concrete (double "T") building.

14.0 THIRTEEN EXAMPLE BUILDINGS

Thirteen building types which are considered to be representative of common industrial facilities are presented graphically and summarized on the following pages. A summary of the buildings and the construction types which they represent is shown below in Table 24. These buildings have been input into the input files EXAMPL1.BLG through EXAMPL13.BLG on the FACEDAP program disk. See Section 3.0 for instructions on how to copy these files off the disk. These files correspond to the Building Numbers 1 through 13 in Table 24. The files EXAMPL2A.BLG and EXAMPL7A.BLG contain the input for Buildings No. 2 and No. 7 for the analyses where the frames are considered as frame components. The files EXAMPL2.BLG and EXAMPL7.BLG contain the input for the analyses of these buildings where the frame components are input as individual beam and column components responding in flexure, or buckling, to the applied blast load. This type of input for buildings with moment resisting frames is discussed in Section 8.6.3. The doors and windows of the buildings are not analyzed with the FACEDAP program. If only a rough estimate of door and window damage is desired, the doors and windows can be assumed to fail if they are loaded by a peak blast pressure greater than 2 psi. Otherwise, a more detailed analysis is required using other analysis methods.

Table 24. A Summary Description of the Thirteen Building Types

Building No.	Page No.	Description
1	170	One-story, large (> 6000 ft^2), reinforced concrete building.
2	173	One-story, small (< 6000 ft^2), reinforced concrete and reinforced masonry building with moment resisting reinforced concrete frames.
3	176	One-story, small (< 6000 ft^2) unreinforced masonry building.
4	179	One-story, small (< 6000 ft^2), unreinforced clay brick building.
5	182	One-story, small (< 6000 ft^2), metal stud wall building.
6	185	Two-story, small (< 6000 ft^2), reinforced concrete building.
7	188	One-story, small (< 6000 ft^2), pre-engineered (Butler® type) building with steel moment resisting frames.
8	191	One-story, large (> 6000 ft^2) metal stud wall building.
9	194	Two-story, small (< 6000 ft^2, timber building.
10	196	One-story, large (> 6000 ft^2), tilt-up reinforced concrete building.
11	198	One-story, large (> 6000 ft^2), heavy timber building.
12	201	Two-story, large (> 6000 ft^2), steel frame building.
13	204	One-story, large (> 6000 ft^2), prestressed concrete (double "T") building.

BUILDING NO. 1

This structure has reinforced concrete walls and columns. The roof is constructed of lightweight concrete over corrugated metal decking and is supported by open web steel joists. This one story building is 9,600 square feet and is 12 feet in height. Plan view and sections follow. This structure is comprised of the following components. Due to the large number of building components, the roof panels and joists are grouped with four components per group and only the middle component is input into the program. However, the weighting factor assigned to this component is multiplied by four to account for the components in each group which are not entered. This approach is discussed in Section 8.1.6. Also, corner columns are not input because the in-plane support provided in both directions by the attached wall panels is considered sufficient to preclude blast damage.

Component

19' x 12' x 6" reinforced concrete wall panels, reinforced with #4 rebar @ 8" E.F.E.W., considered simply supported, two-way panels, 19' x 12' loaded area.

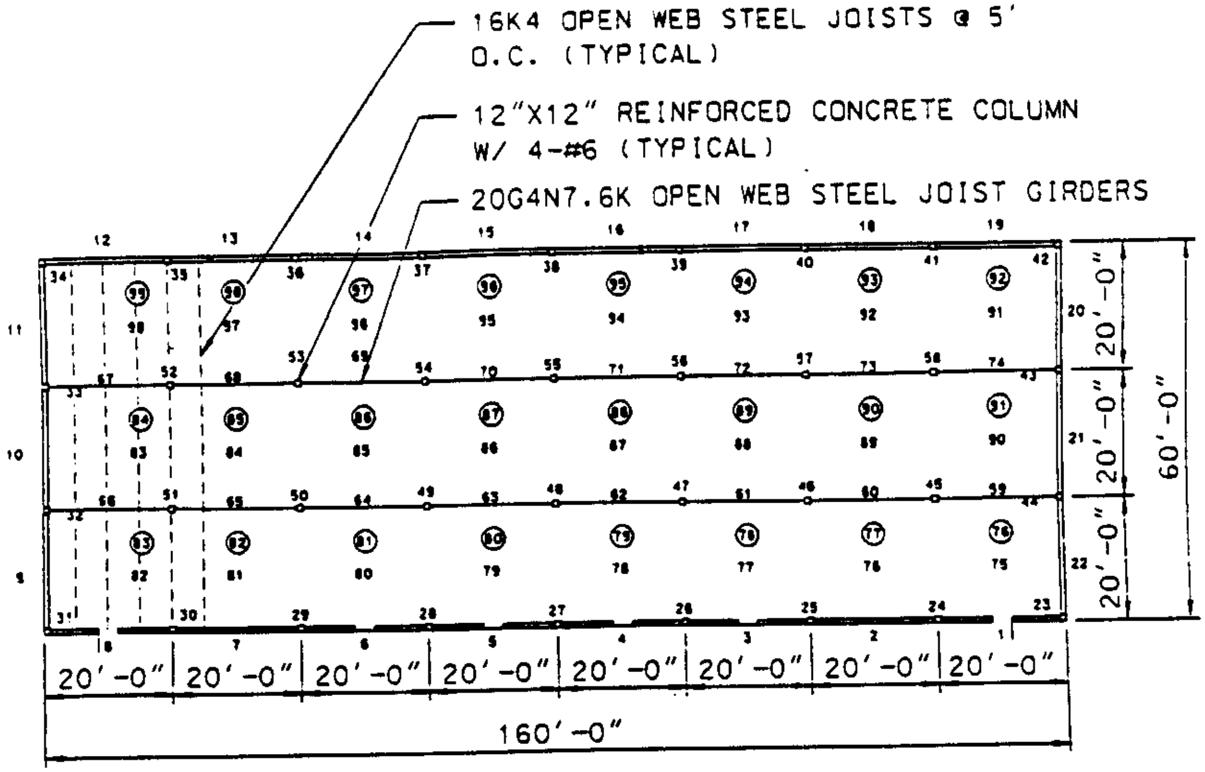
Reinforced concrete exterior columns, 12" x 12" section with four #6 rebar, 12' span, assume fixed-simple supports, 12' x 19' loaded area.

Reinforced concrete interior columns, 12" x 12" section with four #6 rebar, 12' height, assume fixed-simple supports with no sidesway, consider buckling, 20' x 20' loaded roof area.

Open web steel joist girders spanning between columns, 20' span at 20' on center, 20' x 20' loaded area, 20G4N7.6K joists (SJI designation).

Open web steel joists directly supporting roof panels, 20' span at 5' on center, 20' x 5' loaded area, 16K4 joists (SJI designation).

Lightweight reinforced concrete/corrugated metal decking roof panels, 5' span, fixed supports, Vulcraft 0.6C22 decking with maximum 4" concrete cover.



ROOF PLAN

COMPONENTS 1 - 22 ARE WALL PANELS STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.

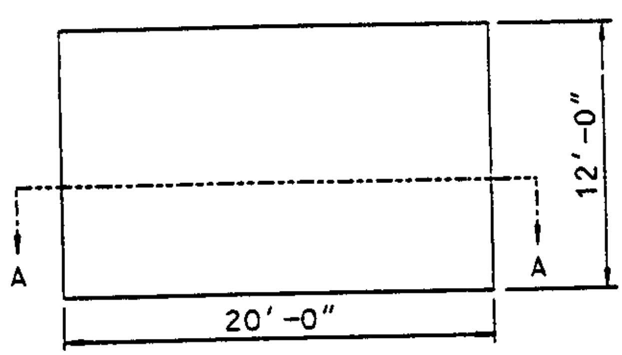
COMPONENTS 23 - 44 ARE EXT. COLUMNS STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.

COMPONENTS 45 - 58 ARE INT. COLUMNS STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.

COMPONENTS 59 - 74 ARE JOIST GIRDERS STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.

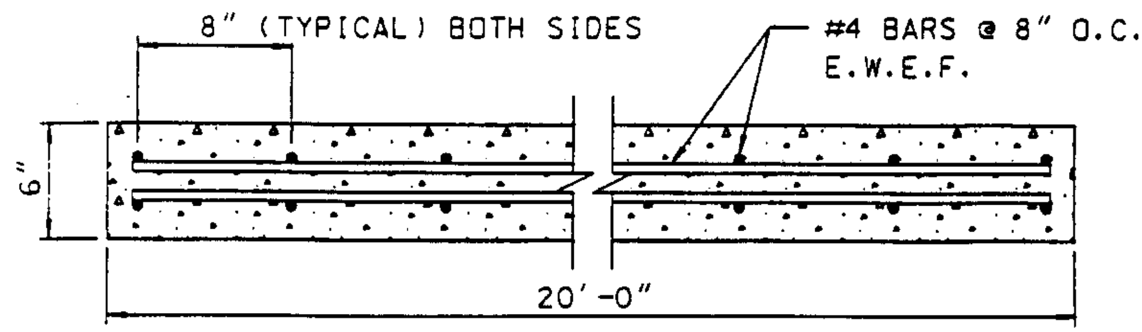
COMPONENTS 75 - 91 ARE STEEL JOIST STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.

COMPONENTS (76) - (99) ARE ROOF PANELS STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.

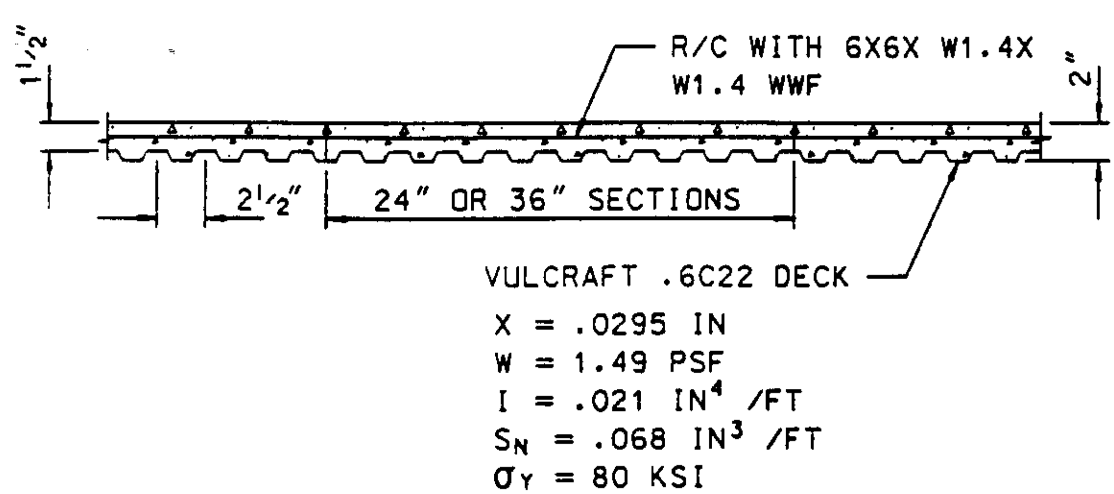


TYPICAL REINFORCED CONCRETE WALL PANEL

Building Type #1 Plan and Details



SECTION A-A



TYPICAL SECTION THROUGH ROOF DECK

Building Type #1 Details (continued)

BUILDING NO. 2

This structure has a reinforced concrete moment resisting frame with marginally reinforced CMU in-fill walls. The roof is lightweight reinforced concrete. This one-story building has 1600 square feet and is 12 feet in height. Plan view and sections follow. Corner columns are not input because the in-plane support provided in both directions by the attached wall panels is considered sufficient to preclude blast damage. Also, the edge beams which bear on the masonry endwalls along their length have not been input since they are continuously supported by the wall.

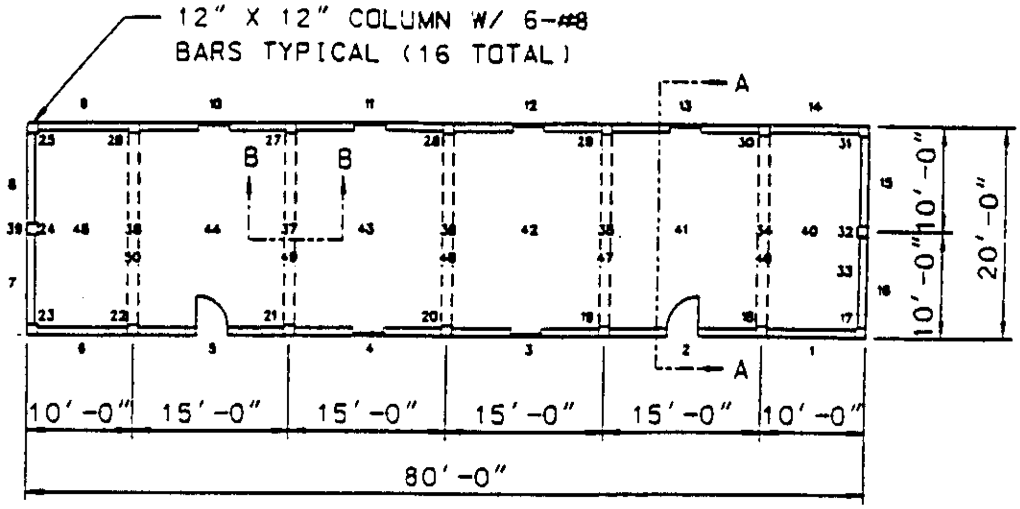
Component

Marginally reinforced CMU in-fill walls, two-way simple supports, 15' x 11' or 10' x 11' loaded area.

Reinforced concrete columns, consider as one-way in bending, include damage from frame side sway, fixed supports, 15' x 11' or 10' x 11' loaded area.

Reinforced concrete beams, analyzed assuming all are identical (end spans are actually shorter), fixed supports, 15' x 20' or 5' x 20' loaded area.

Lightweight reinforced concrete roof section, 4" thick, 10' or 15' span.



ROOF PLAN

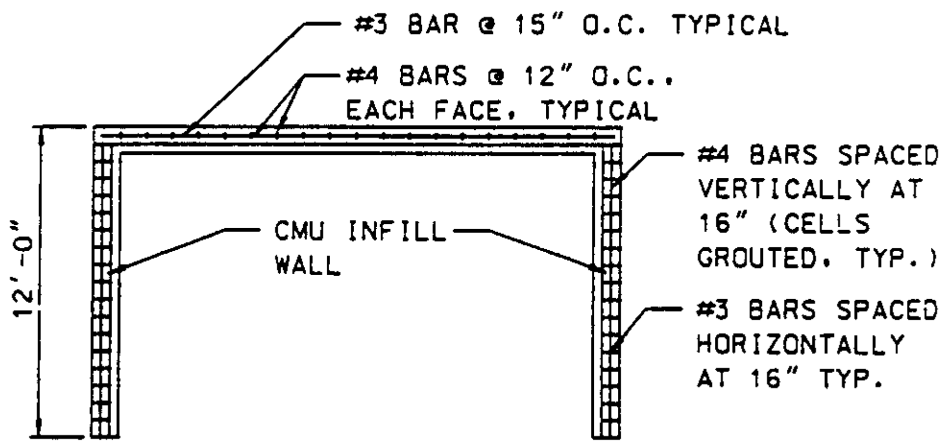
COMPONENTS 1 - 16 ARE WALL PANELS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 17 - 32 ARE EXT. COLUMNS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 33 - 39 ARE REINFORCED CONCRETE BEAMS PROCEDING RIGHT TO LEFT

COMPONENTS 40 - 45 ARE LIGHTWEIGHT ROOF PANELS PROCEDING RIGHT TO LEFT

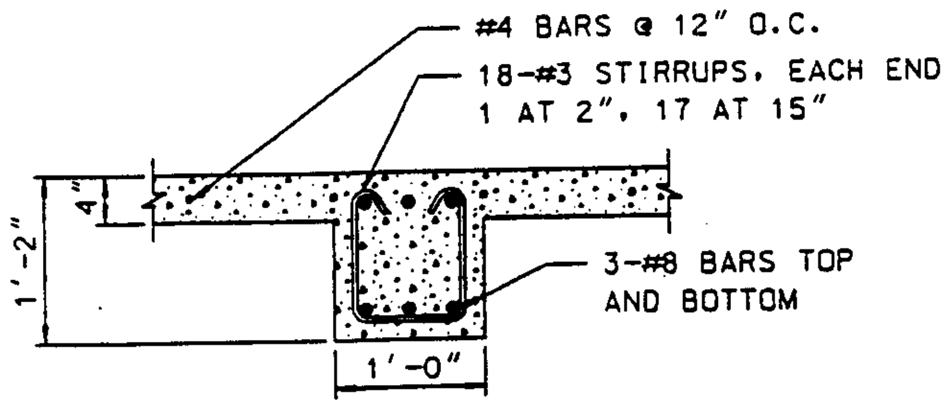
COMPONENTS 46 - 50 ARE CONCRETE FRAMING MEMBERS PROCEDING RIGHT TO LEFT



SECTION A-A

NOTES:
- 4000 PSI CONCRETE STRENGTH
- 60 KSI REINFORCEMENT STRENGTH
- 2000 PSI COMPRESSIVE CMU STRENGTH
- 200 PSI CMU RUPTURE MODULUS

Building Type #2 Plan and Details



SECTION B-B

Building Type #2 Details (continued)

This structure has unreinforced CMU bearing walls with reinforced CMU pilasters. All cells are grouted. The roof supports are open-web steel joists. The roof is constructed of lightweight concrete over corrugated metal decking. This one-story building has 1600 square feet and is 12 feet in height. Plan view and sections follow. Corner columns are not input because the in-plane support provided in both directions by the attached wall panels should preclude blast damage. Ten foot wide one-way reinforced concrete roof panels have been input since this causes the end points of the end panels to lie along the top edge of the masonry wall panels in the endwalls between the corner points of the two-way wall panels. This allows the Preprocessor to calculate the dependency between the roof panels and the wall panels (see Tables 16 and 17 for the rules used by the Preprocessor to calculate dependencies).

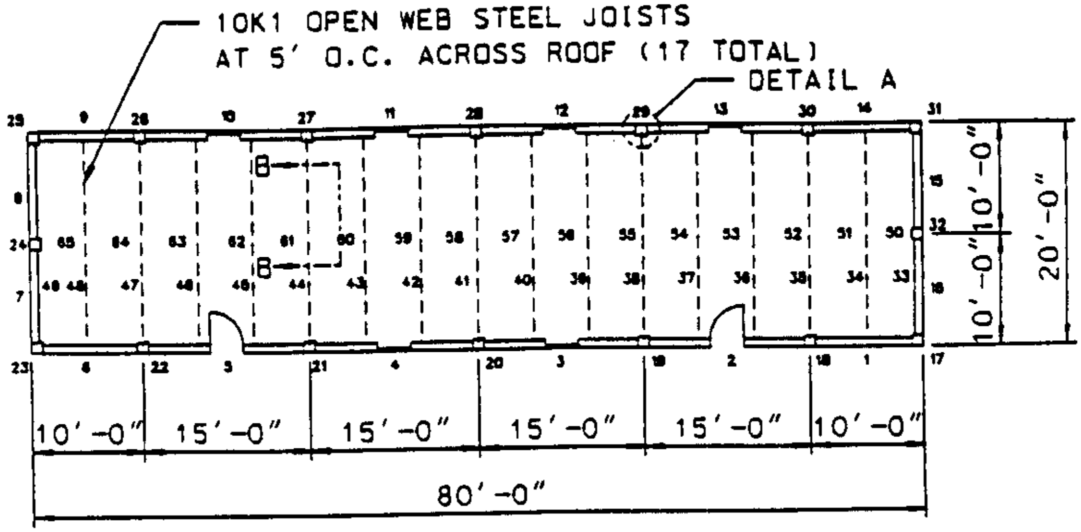
Component

13.7' x 12' x 7.625" CMU panels, unreinforced, single wythe, considered simply supported, two-way 13.7' x 12' loaded area.

12' x 15.625" x 15.625" CMU pilasters, considered simply supported beams, loaded area is 38% of 15' x 12' area, reinforced with four #8's at d = 12.625".

Open-web steel joists, 20' span at 5' on center, 20' x 5' loaded area, 10K1 joists (SJI designation).

Corrugated steel/concrete decking, 5' span, fixed supports, .6C22 panels with 1-1/2" lightweight concrete cover.



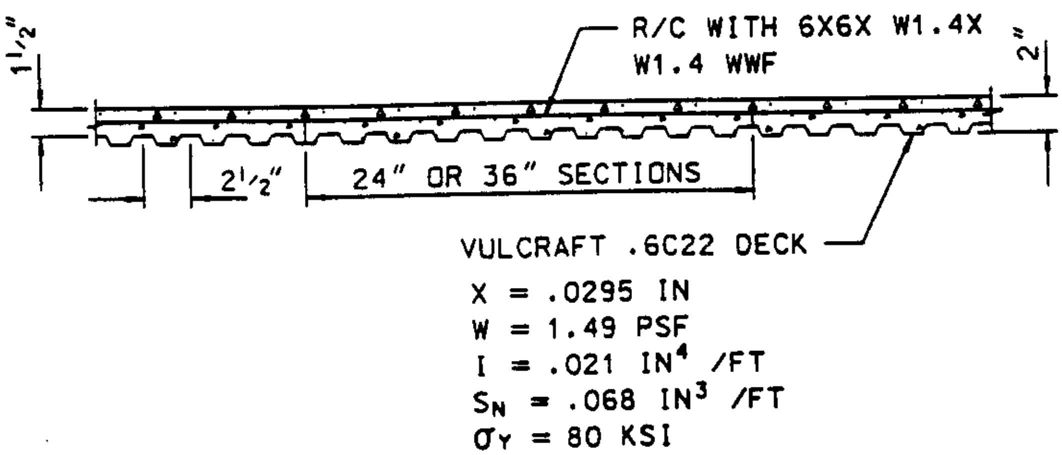
ROOF PLAN

COMPONENTS 1 - 16 ARE WALL PANELS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 17 - 32 ARE EXT. COLUMNS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 33 - 49 ARE STEEL JOIST PROCEDING RIGHT TO LEFT

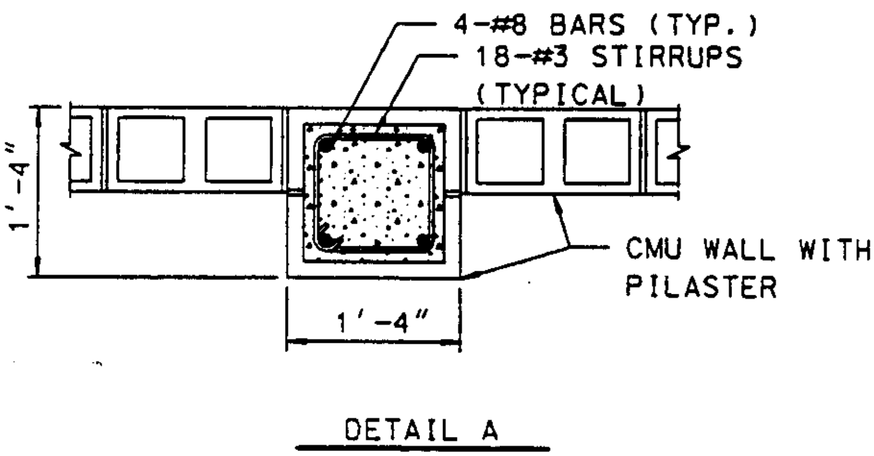
COMPONENTS 50 - 65 ARE LIGHTWEIGHT ROOF PANELS PROCEDING RIGHT TO LEFT



SECTION B-B

NOTES:
- 3000 PSI CONCRETE STRENGTH
- 40 KSI REINFORCEMENT STRENGTH
- 2000 PSI COMPRESSIVE CMU STRENGTH
- 200 PSI CMU RUPTURE MODULUS

Building Type #3 Plan and Details



Building Type #3 Details (continued)

BUILDING NO. 4

This structure has unreinforced cavity clay brick walls and brick pilasters. The roof is supported by open-web steel joists. The roof is constructed of corrugated metal panels. This one-story building has 1600 square feet and is 12 feet in height. Plan view and sections follow. Corner columns are not input because the in-plane support provided in both directions by the attached wall panels should preclude blast damage. The roof panels are input as described in Building No. 3.

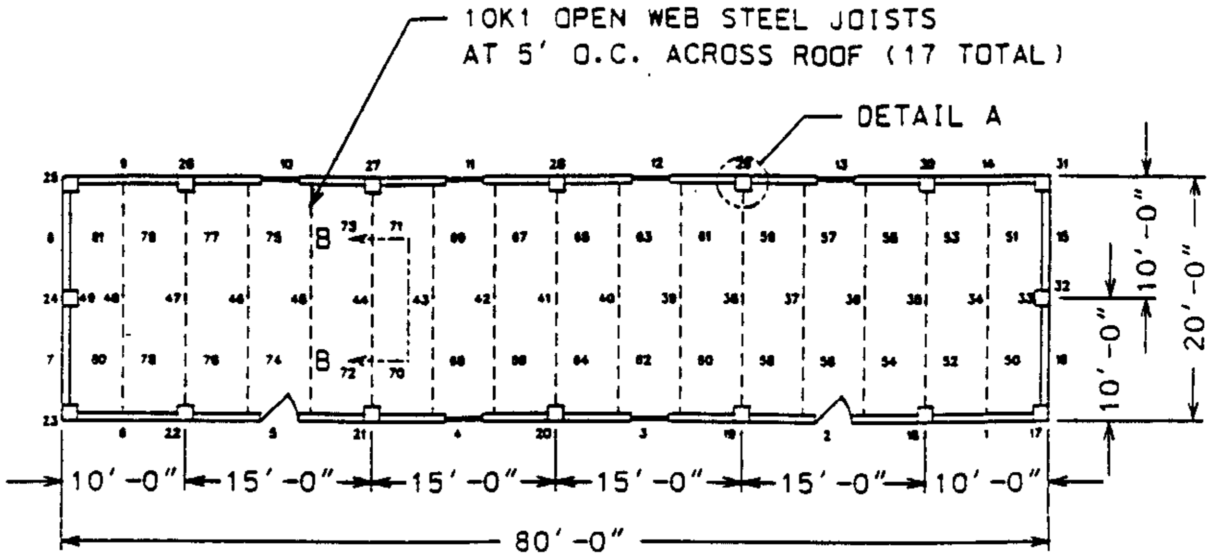
Component

13.7' x 12' x 8' clay brick cavity wall construction (double wythe), considered simply supported, two-way 13.7' x 12' loaded area.

12' x 16" x 16" clay brick pilasters, considered simply supported beams, loaded area is 38% of 15' x 12' area. Reinforcement is four #8's at d = 12.625".

Steel joists, 20' span, 20' x 5' loaded area, 10K1 joists (SJI designation).

Corrugated steel panels, 10' x 5' loaded area, Vulcraft .6C26' 0.0179" thick.



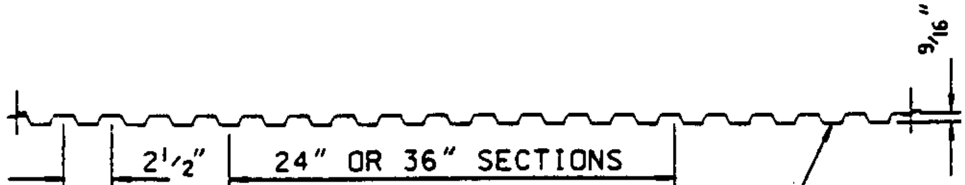
ROOF PANEL

COMPONENTS 1 - 16 ARE WALL PANELS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 17 - 32 ARE EXT. COLUMNS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 33 - 49 ARE STEEL JOIST PROCEDING RIGHT TO LEFT.

COMPONENTS 50 - 81 ARE METAL ROOF PANELS PROCEDING BOTTOM TO TOP AND RIGHT TO LEFT.



VULCRAFT .6C26 DECK

$W = 1.01$ PSF

$I = .013$ IN^4 /FT

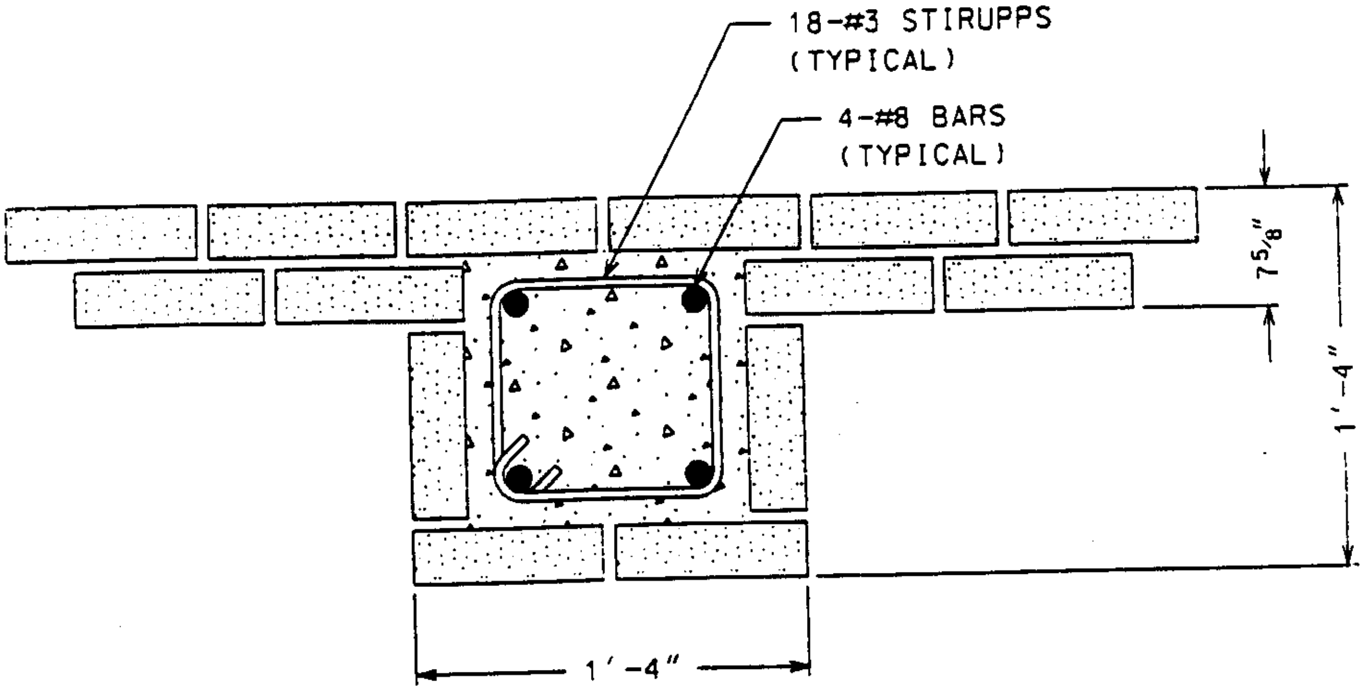
$S_N = .042$ IN^3 /FT

$\sigma_Y = 80$ KSI

SECTION B-B

NOTES:
- 2000 PSI CONCRETE STRENGTH
- 40 KSI REINFORCEMENT STRENGTH
- 200 PSI BRICK RUPTURE MODULUS

Building Type #4 Plan and Details



DETAIL A

Building Type #4 Details (continued)

BUILDING NO. 5

This structure has metal stud bearing walls with braced interior metal stud walls which provide lateral support. The roof is supported by open-web steel joists. The roof is constructed of corrugated metal panels. This one-story building has 1600 square feet and is 14 feet in height. Plan view and sections follow.

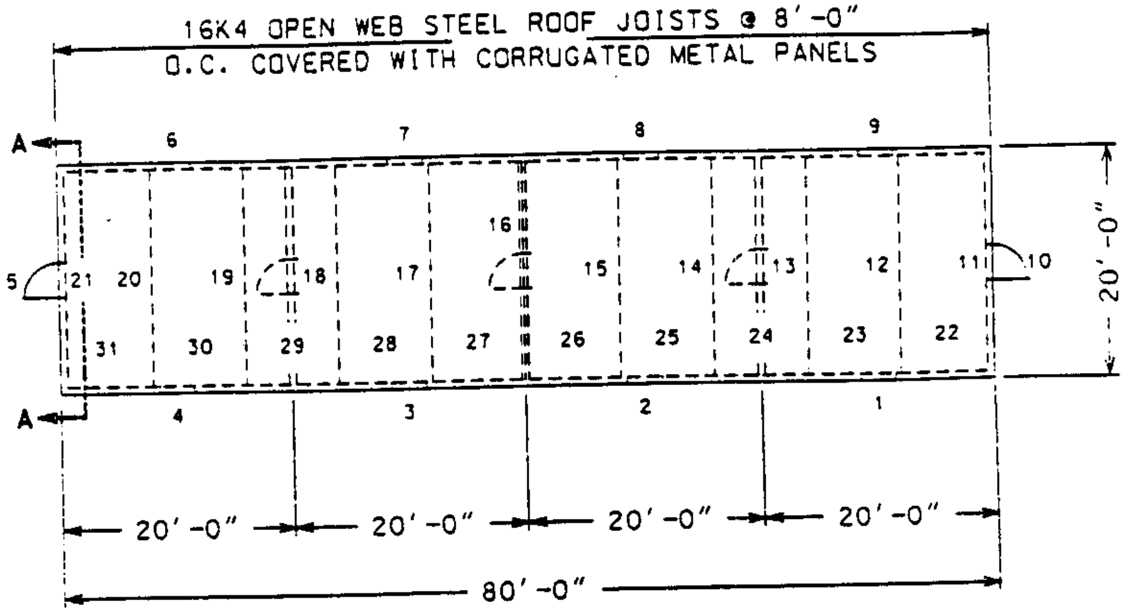
Component

Load bearing metal stud wall, 2" x 6" x 12 gage studs at 16" on center, 14' span, simple supports.

Non-loadbearing metal stud wall 2" x 6" x 12 gage studs at 16" on center, 14' span, simple supports.

Steel joists, 20' span, 20' x 8' loaded area, 16H4 joists (SJI designation).

Corrugated steel roof panels, 20' x 8' loaded area, H.H. Robertson Section 3-16, 16 gage thickness.

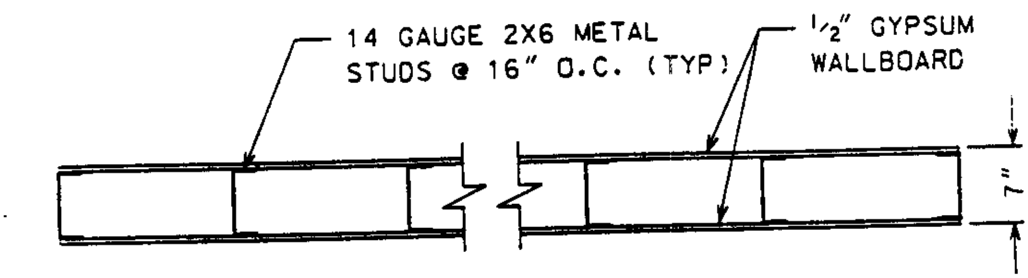


ROOF PLAN

COMPONENTS 1 - 10 ARE WALL PANELS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 11 - 21 ARE STEEL JOIST PROCEDING RIGHT TO LEFT.

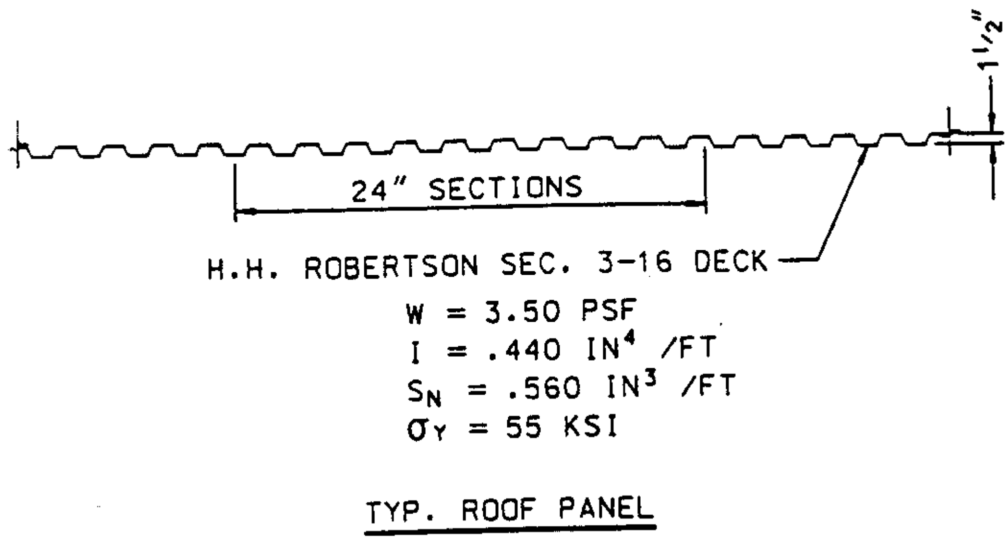
COMPONENTS 22 - 31 ARE METAL WALL PANEL PROCEDING RIGHT TO LEFT.



SECTION A-A

NOTE: METAL STUD WALLS SHALL BE COLD FORMED STEEL WITH MINIMUM YEILD 50,000 psi.

Building Type #5 Plan and Details



Building Type #5 Details (continued)

BUILDING NO. 6

This two story structure has 8" thick reinforced concrete load bearing walls in each story and a 4" lightweight concrete roof. The roof is supported by the walls and two reinforced concrete beams. An interior column is located in the center of floor area and supports the roof beams. This building has 900 square feet in each story for a total of 1800 square feet. Each story height is 14'. Plan view and sections follow. The dependencies for the roof panels (where they are supported by the wall panels) were input by hand into the *Dependencies* spreadsheet since these dependencies were not calculated by the Preprocessor.

Component

Reinforced concrete wall panels, one-way slabs, simple supports, 14' x 15'loaded area, first floor.

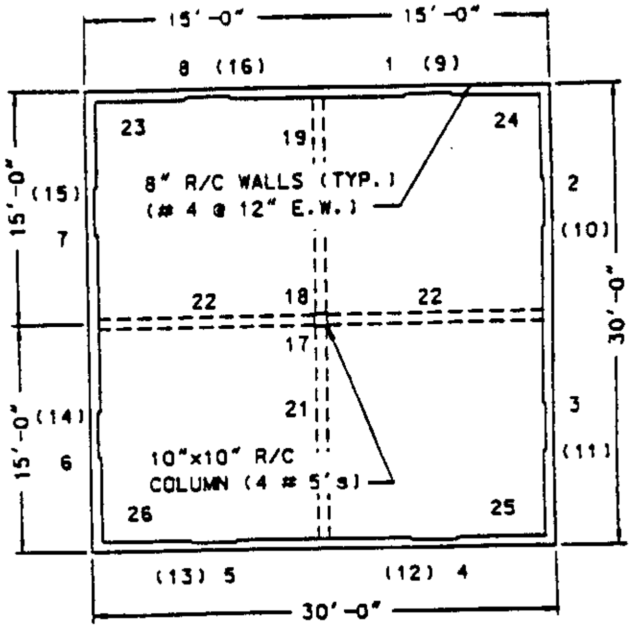
Reinforced concrete wall panels, one-way slabs, simple supports 14' x 15' loaded area, second floor.

4" lightweight concrete roof panel, two-way slab, simple supports, 15' x 15' loaded area.

12" x 8" reinforced concrete roof beam, simple supports, 112 square feet loaded area, 15' span.

Reinforced concrete interior column, second floor, consider buckling, 15' x 15' loaded roof area.

Reinforced concrete interior column, first floor, consider buckling 15' x 15' loaded area from both roof and second floor.



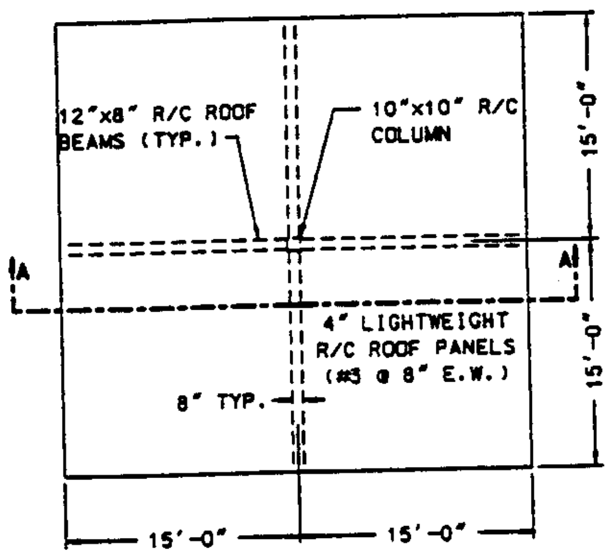
PLAN VIEW (1st. & 2nd. Floors)

COMPONENTS 1 - 16 ARE WALL PANELS STARTING FROM UPPER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 17 - 18 ARE INTERIOR COLUMNS PROCEDING BOTTOM TO TOP.

COMPONENTS 19 - 22 ARE CONCRETE BEAMS STARTING FROM THE TOP, PROCEDING CLOCKWISE.

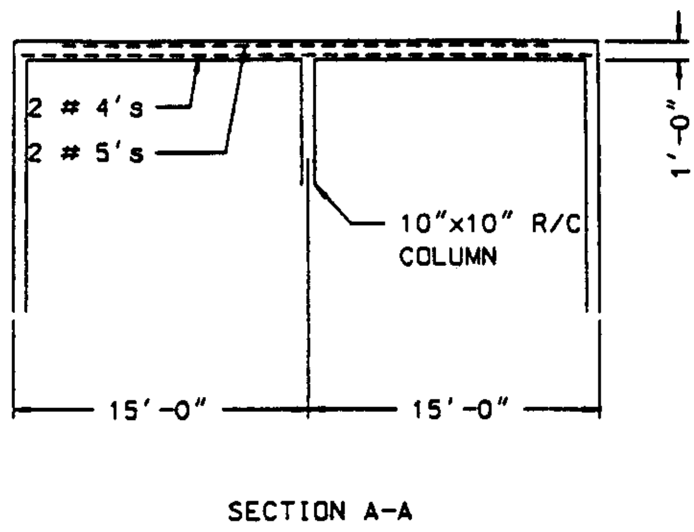
COMPONENTS 23 - 26 ARE ROOF PANELS STARTING FROM UPPER LEFT CORNER, PROCEDING CLOCKWISE.



ROOF PLAN

NOTES: fy = 60,000 psi
fc = 4,000 psi

Building Type #6 Plan



Building Type #6 Elevation (continued)

BUILDING NO. 7

This structure is a pre-engineered building as shown in the following drawings. This one-story building has 1600 square feet and is 12 feet in height.

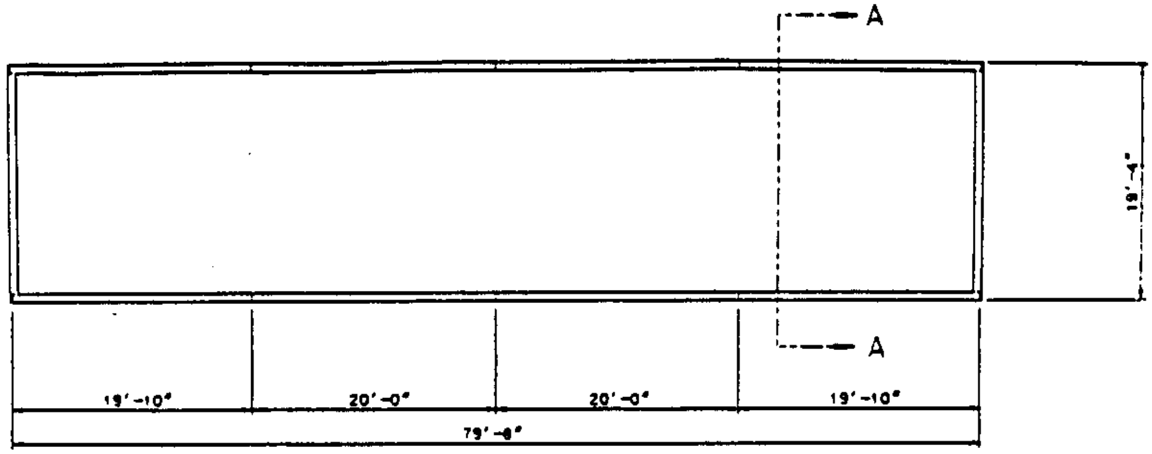
Component

Corrugated roof and wall panels, 4' x 20' loaded area, H.H. Robertson Section 3-22, 22 gage thickness.

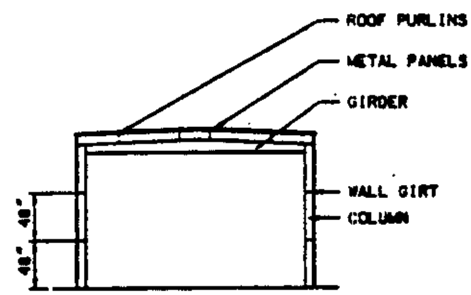
8" x 2" x 0.075" zee-section roof purlins, 4' x 20' loaded area, 20' span, simple supports.

8" x 2" x 0.060" zee-section wall girts (on side walls), 4' x 20' loaded area, 20' span, simple supports.

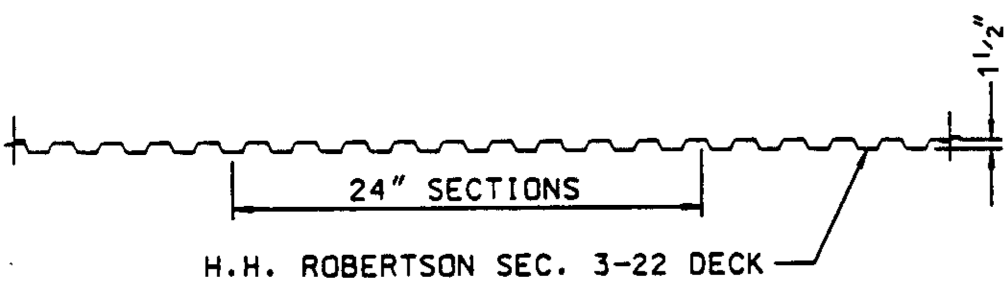
Moment resisting steel frame with members which may be approximated as W8x18 columns and a W12x14 girder. Girder span is 20' and column height is 12'. Column loaded area is 20' x 12' and girder loaded area is 20' x 20'.



PLAN VIEW



SECTION A-A



$W = 1.80$ PSF
$I = .180\ IN^4$ /FT
$S_N = .200\ IN^3$ /FT
$\sigma_Y = 55$ KSI

TYP. ROOF/WALL PANEL

Building Type #7 Plan and Details

COMPONENTS 1 - 10 ARE EXT. COLUMNS PROCEDING RIGHT TO LEFT.

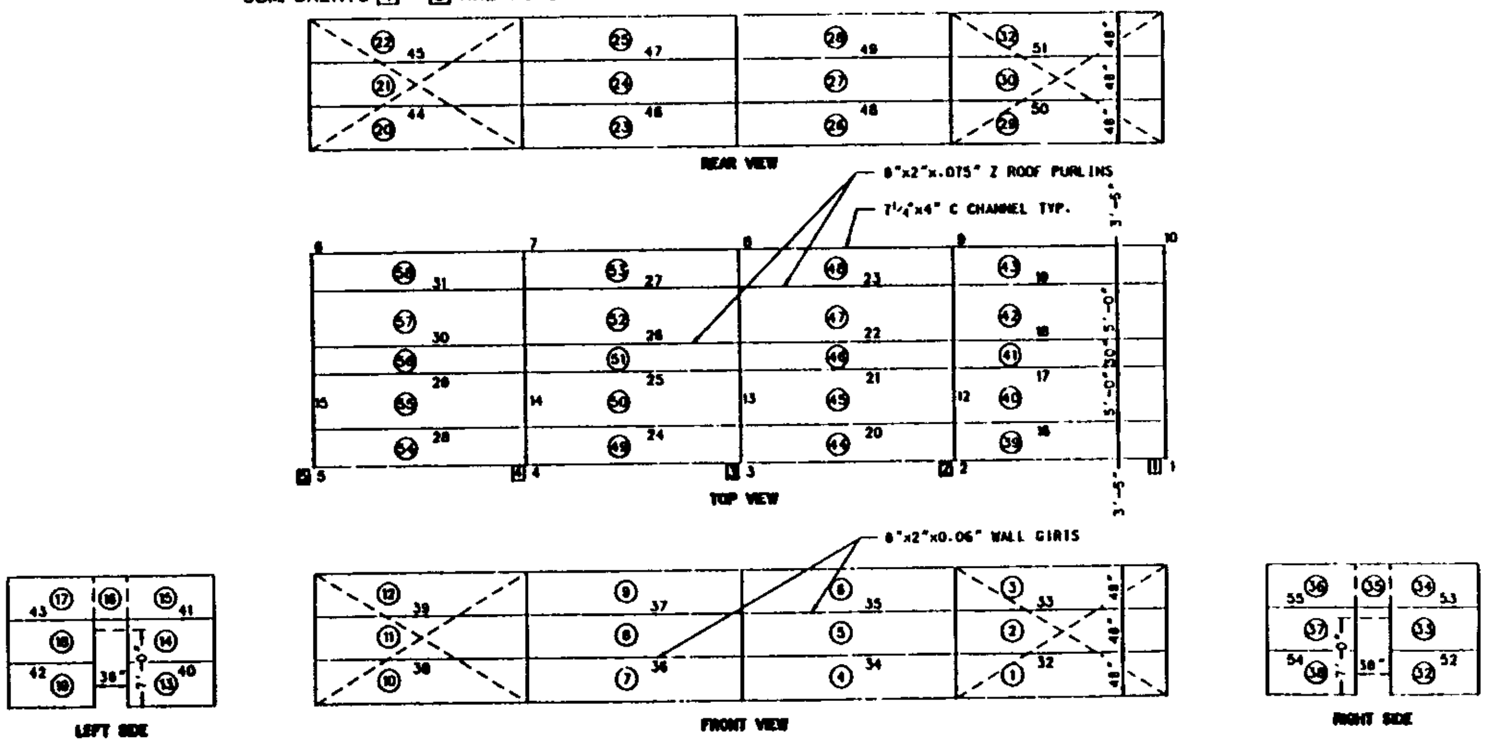
COMPONENTS 11 - 15 ARE ROOF BEAMS PROCEDING RIGHT TO LEFT.

COMPONENTS 16 - 31 ARE ROOF PURLINS PROCEDING BOTTOM TO TOP AND RIGHT TO LEFT.

COMPONENTS 32 - 55 ARE WALL GIRTS PROCEDING BOTTOM TO TOP AND RIGHT TO LEFT.

COMPONENTS (1) - (58) ARE METAL WALL/ROOF PANELS PROCEDING BOTTOM TO TOP AND RIGHT TO LEFT.

COMPONENTS [1] - [5] ARE MOMENT RESISTANT FRAMES PROCEDING RIGHT TO LEFT.



FRAMING PLAN

Building Type #7 Details (continued)

BUILDING NO. 8

This structure has metal stud bearing walls with braced interior metal stud walls which provide lateral support. The roof is supported by open-web steel joists. The roof is constructed of corrugated metal panels. This one-story building has 10,000 square feet and is 14 feet in height. Plan view and sections follow. Due to the large number of building components, the cladding and joist components are grouped with three components per group and only the middle component is input into the program. However, the weighting factor assigned to this component is multiplied by three to account for the components which are not entered. Wall panels which support roof components are assigned larger weighting factors than those which do not.

Component

Load bearing metal stud wall, 2" x 6" x 12 gage studs at 16" on center, 14' span, simple supports.

Non-loadbearing metal stud wall 2" x 6" x 12 gage studs at 16" on center, 14' span, simple supports.

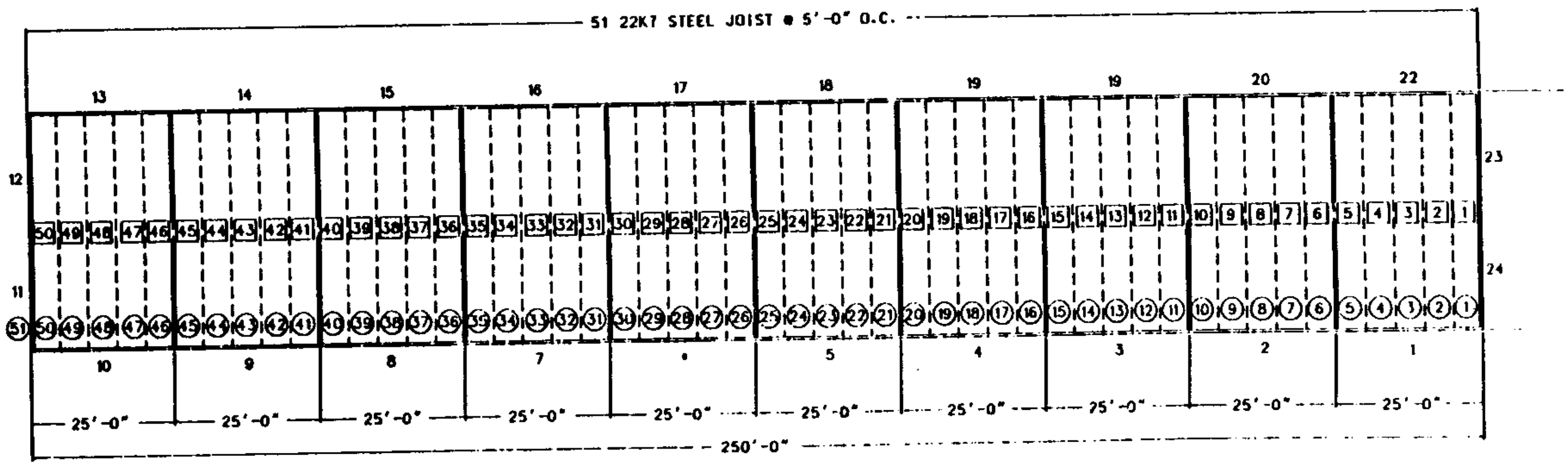
Steel joists, 20' span, 20' x 5' loaded area, 22H7 joists (SJI designation).

Corrugated steel roof panels, 40' x 5' loaded area, H.H. Robertson Section 3-16, 16 gage thickness.

COMPONENTS 1 - 24 ARE WALL PANELS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

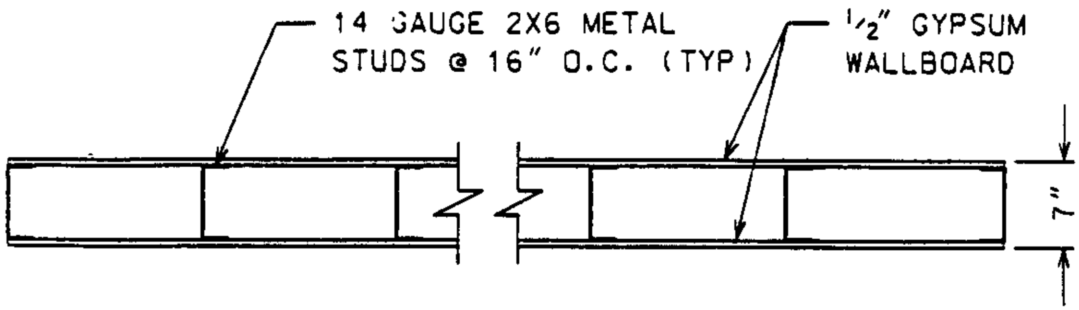
COMPONENTS (1) - (51) ARE STEEL JOIST PROCEDING RIGHT TO LEFT.

COMPONENTS [1] - [50] ARE METAL ROOF PANELS PROCEDING RIGHT TO LEFT.



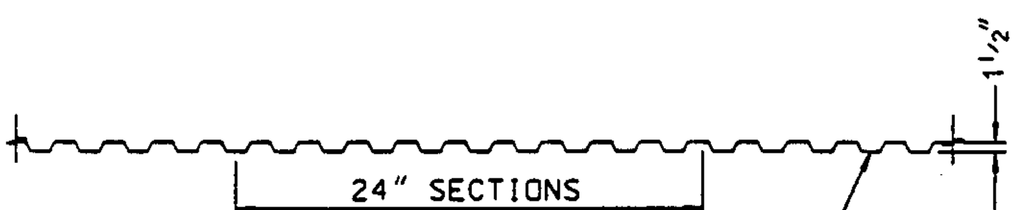
PLAN VIEW

Building Type #8 Plan



TYP. WALL SECTION

NOTE: METAL STUD WALLS SHALL BE COLD FORMED STEEL WITH MINIMUM YEILD 50.000 psi.



H.H. ROBERTSON SEC. 3-16 DECK

$W = 3.50$ PSF

$I = .440\ IN^4\ /FT$

$S_N = .560\ IN^3\ /FT$

$\sigma_Y = 55$ KSI

TYP. ROOF PANEL

Building Type #8 Details (continued)

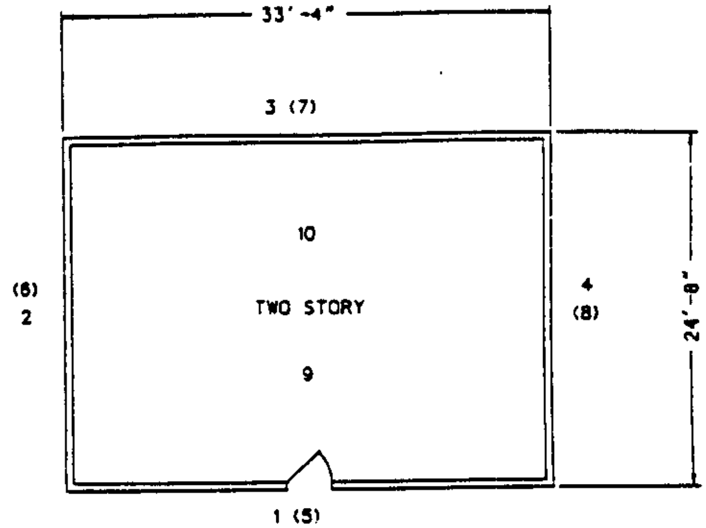
BUILDING NO. 9

This building has exterior stud, loadbearing walls with plank sheath siding. The roof is plywood decking with asphalt shingles supported by wood rafters. This building has 1600 square feet and has two stories. Plan view and sections follow. Wall panels which support roof components are assigned larger weighting factors than those which do not.

Component

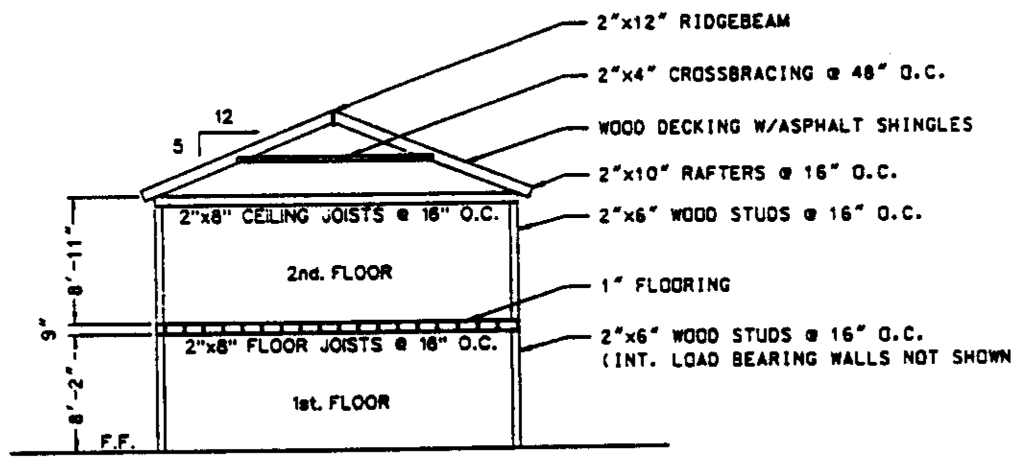
Wall panels, 2" x 6" studs at 16" on center with plank exterior sheathing, 8' span.

Roof panels, 2" x 10" joists at 16" on center with wood decking, 12.33' span.



PLAN VIEW

COMPONENT 1 - 8 ARE WALL PANELS STARTING AT THE BOTTOM, PROCEDING CLOCKWISE.
COMPONENT 9 - 10 ARE ROOF PANELS PROCEDING BOTTOM TO TOP.



WALL SECTION

Building Type #9 Plan and Elevation

BUILDING NO. 10

This building is constructed of tilt-up lightweight reinforced concrete panels supported by poured-in-place reinforced concrete columns. The roof is supported by open-web steel joists. The roof is constructed of lightweight concrete over corrugated metal decking. Interior columns are provided on a 20 foot by 20 foot grid. This building has 6000 square feet and is 14 feet in height. Plan view and sections follow. Due to the large number of building components, the cladding and joist components are grouped with three components per group and only the middle component is input into the program. However, the weighting factor assigned to this component is multiplied by three to account for the components which are not entered. This approach is discussed in Section 8.1.6. Also, corner columns are not input because the in-plane support provided in both directions by the attached wall panels should preclude blast damage.

Component

Tilt-up reinforced lightweight concrete wall panels, two-way slabs, simple supports, 14' x 20' loaded area.

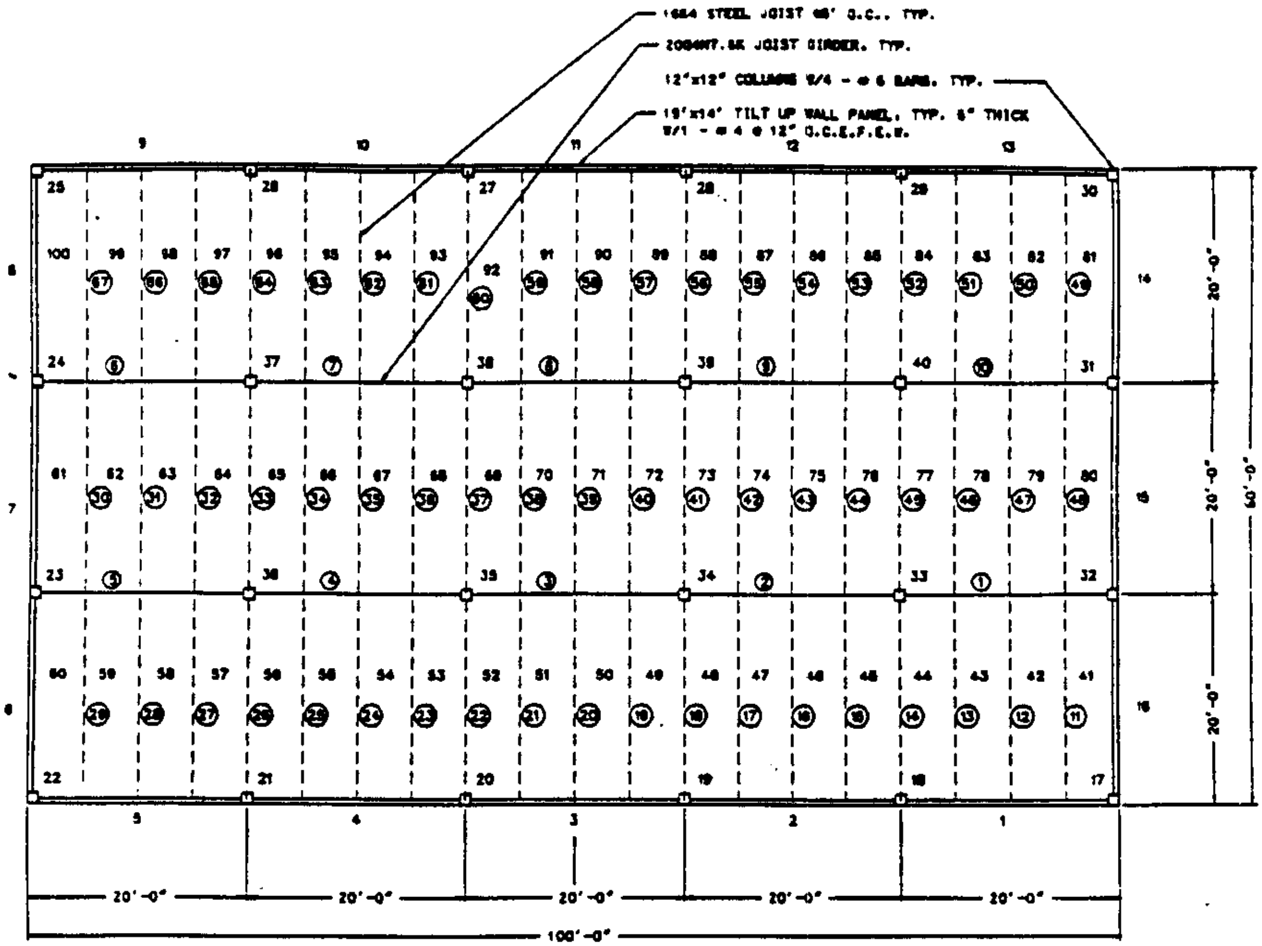
Reinforced concrete exterior columns, consider as one-way in bending, simple supports, 14' x 20' loaded area.

Reinforced concrete interior columns, consider buckling 20' x 20' loaded area.

Joist girder, 20' span at 20' on center, 20' x 20' loaded area, 20G4N7.6K.

Open-web steel joists, 20' span at 5' on center, 20' x 5' loaded area, 16K4 joists (SDI designation).

Corrugated steel/concrete decking, 5' span, fixed supports, Vulcraft 2VL22 deck with 2" lightweight concrete cover.



PLAN VIEW

COMPONENTS 1 - 16 ARE WALL PANELS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

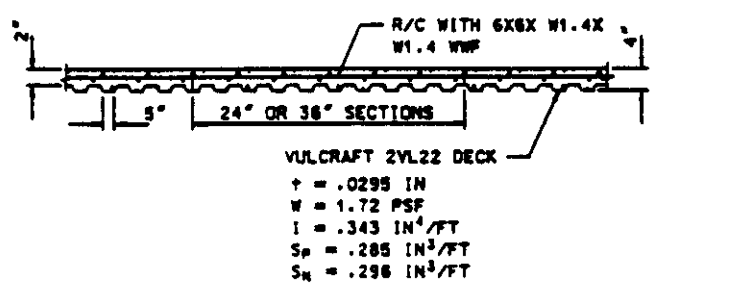
COMPONENTS 17 - 32 ARE EXT. COLUMNS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 33 - 40 ARE INT. COLUMNS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 41 - 100 ARE ROOF PANELS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 1 - 10 ARE JOIST GIRDERS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 11 - 67 ARE STEEL JOIST STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.



TYPICAL SECTION THROUGH ROOF DECK

Building Type #10 Plan and Details

BUILDING NO. 11

This building has exterior stud, loadbearing walls with interior timber columns on a 12 foot by 12 foot grid. The roof is plywood decking with asbestos shingles supported by wood joists. This one-story building has 7200 square feet and has a 10 foot ceiling height. Plan view and sections follow. The top chords of the roof trusses are entered into the program as wood beam components. Other truss components are not entered since there is no representative component type for them. Wood roof components are used to represent the portion of the roof which spans between trusses. Only three out of the four interior columns supporting each truss are input since this is necessary to cause the interior column end points to match with the end points of the input top chord beams in the truss. The true loaded area of the interior columns is entered as a component property and the weighting factor is increased by 4/3 so that the total contribution of the interior columns in the building damage summation routine is not underestimated.

Component

Wall panels, 2" x 6" wood studs at 16" on center with plywood sheathing, 10' span.

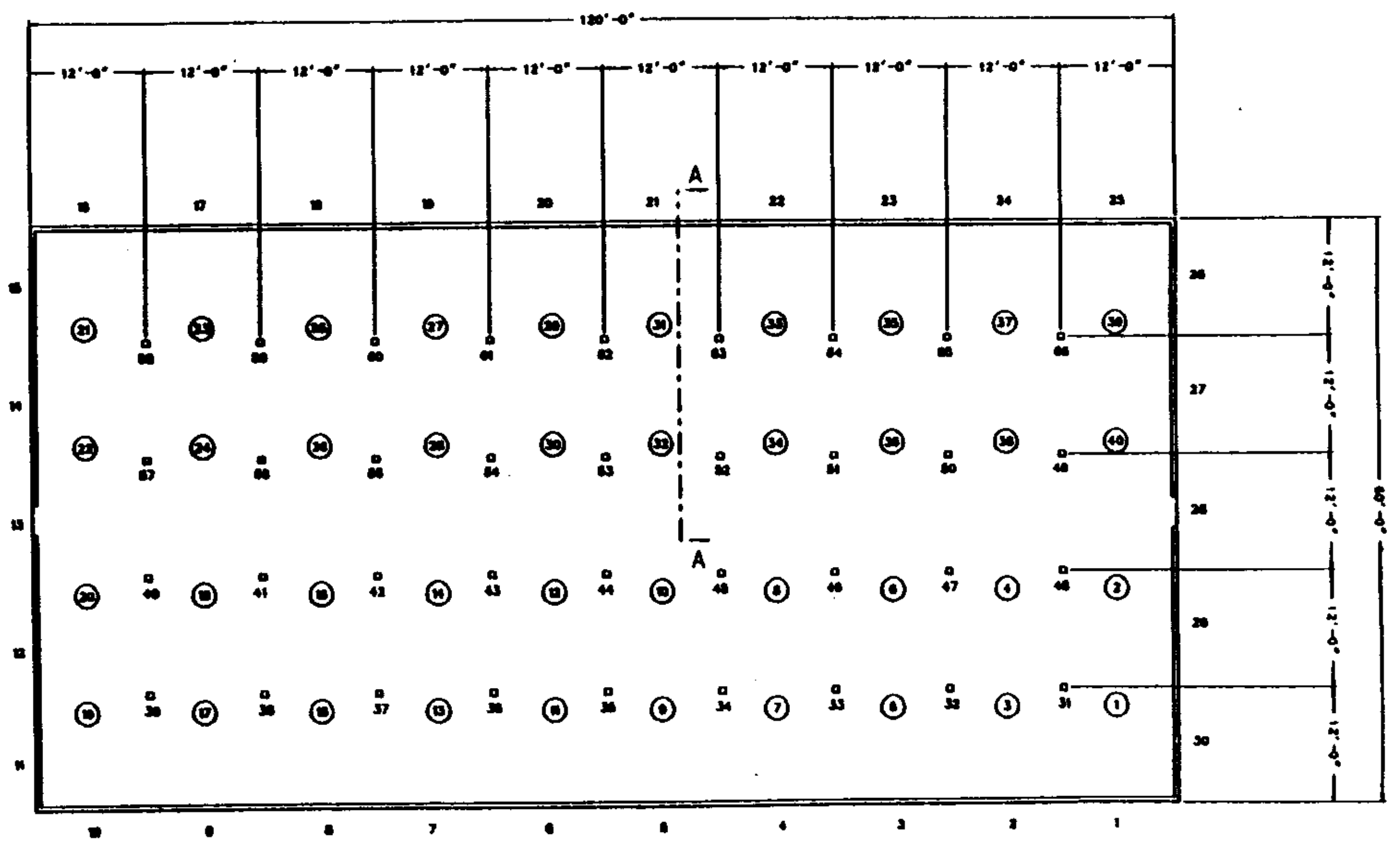
Roof panels, 2" x 10" joists at 16" on center with plywood decking, 15' span.

Interior column, 8" x 8" timber, consider buckling, 12' x 12' loaded area.

COMPONENTS 1 - 30 ARE WALL PANELS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

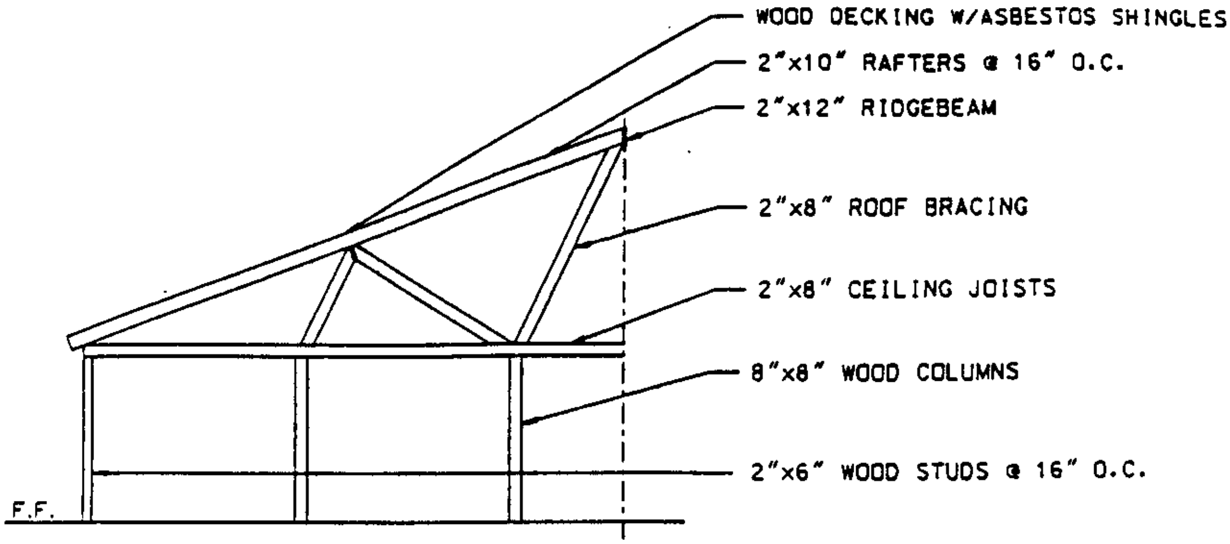
COMPONENTS 31 - 66 ARE INT. COLUMNS STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS (1) - (40) ARE ROOF BEAMS PROCEDING BOTTOM TO TOP AND RIGHT TO LEFT.

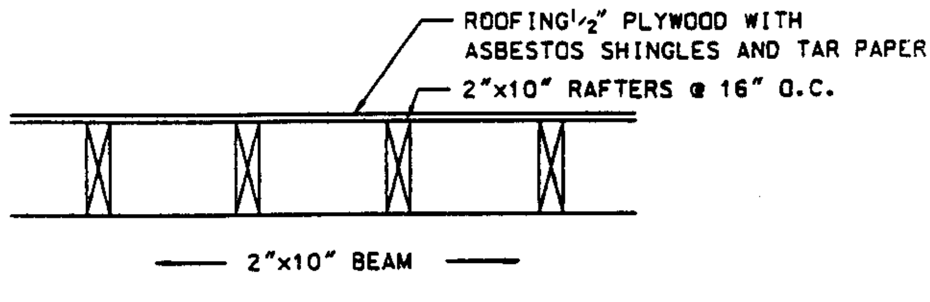


FRAMING PLAN

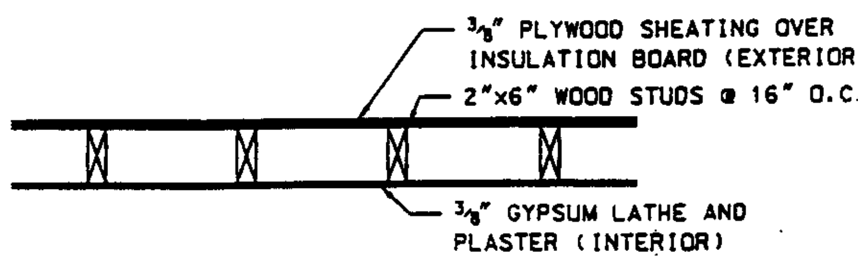
Building Type #11 Plan



SECTION A-A



TYP. ROOF SECTION



TYP. WALL SECTION

Building Type #11 Details (continued)

BUILDING NO. 12

This building has a steel framing members around marginally reinforced in-fill CMU walls. The roof is constructed of corrugated metal panels supported by open-web steel joists. This two-story building has 7000 square feet on each floor for a total of 14,000 square feet. Each story height is 12 feet. Plan view and sections follow. Due to the large number of roof components, the roof panel and joist components are grouped with two or three components per group and only the middle component is input into the program. However, the weighting factor assigned to this component is multiplied by two or three to account for the components which are not entered. Also, only the second floor is entered. The weighting factors of all wall components is multiplied by a factor of two to account for the first floor components which are not input. This approach is discussed in Section 8.1.6. Also, corner columns are not input because the in-plane support provided in both directions by the attached wall panels should preclude blast damage. For this building the CMU walls are assumed to act as shear walls so that the framing members to not need to resist lateral load.

Component

Marginally reinforced CMU in-fill walls, two-way slabs, simple supports, 12' x 20' or 12' x 28' or 12' x 14' loaded areas.

Exterior steel columns, including damage from side sway, assumed fixed supports, 20' x 12' laterally loaded area used for all columns.

Steel roof beams, assumed fixed supports, 28' x 20' or 14' x 20' loaded areas used for analysis.

Lightweight reinforced concrete roof section 4" thick, one-way slabs, 7' or 9'-4" span, 20' x 28' or 20' x 14' loaded sections used as one component.

Steel roof joists, assumed simple supports, 20' x 8' (average) loaded area 3 adjacent joists within 20' x 28' or 20' x 14' used as one component.

Interior steel columns, consider buckling and damage from side sway, 21' x 20' area loading column axially.

COMPONENTS (1) - (32) ARE CMU WALL PANELS STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.

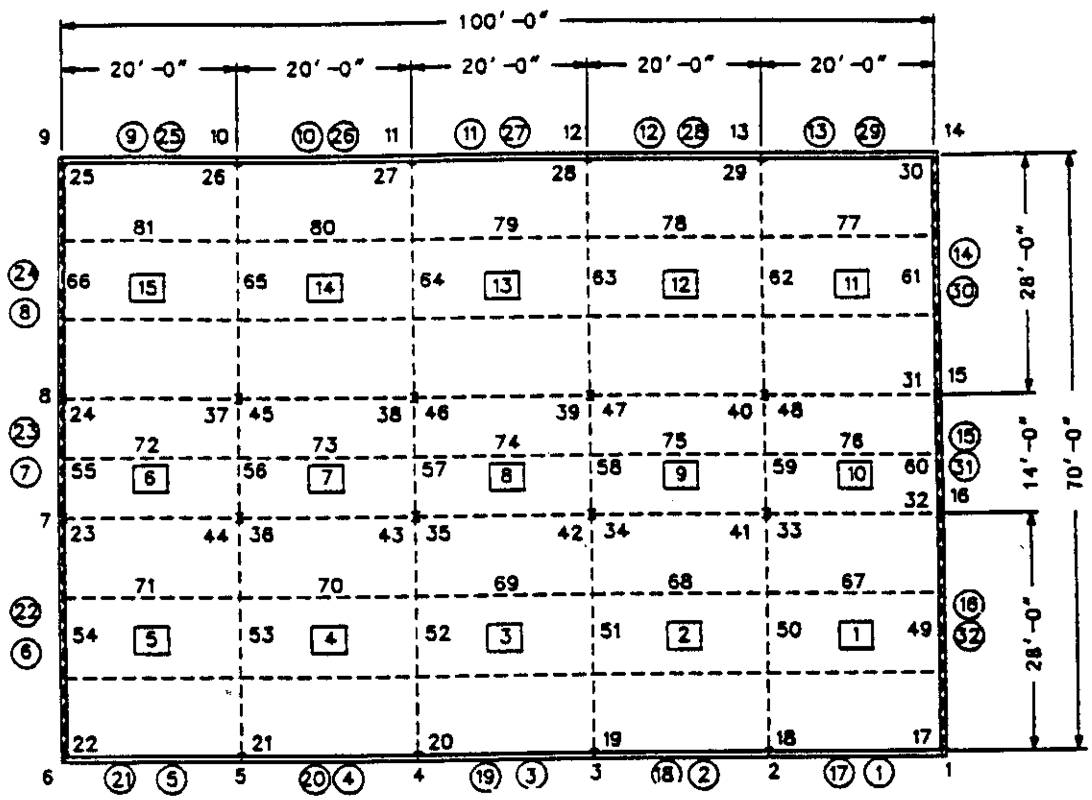
COMPONENTS 1 - 32 ARE EXT. COLUMNS STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.

COMPONENTS 33 - 48 ARE INT. COLUMNS STARTING FROM LOWER RIGHT CORNER. PROCEDINGS CLOCKWISE.

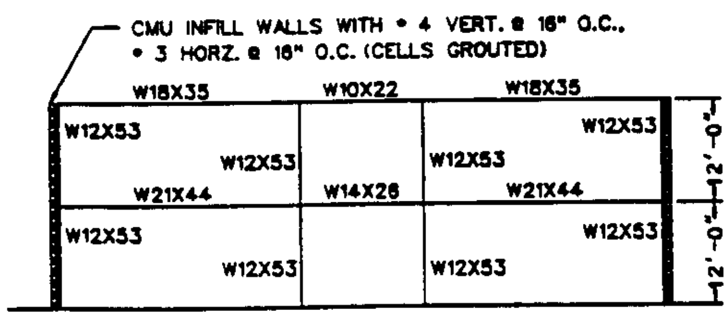
COMPONENTS 49 - 66 ARE ROOF BEAMS STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.

COMPONENTS 67 - 81 ARE STEEL JOISTS STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.

COMPONENTS [1] - [15] ARE ROOF PANELS STARTING FROM LOWER RIGHT CORNER. PROCEDING CLOCKWISE.



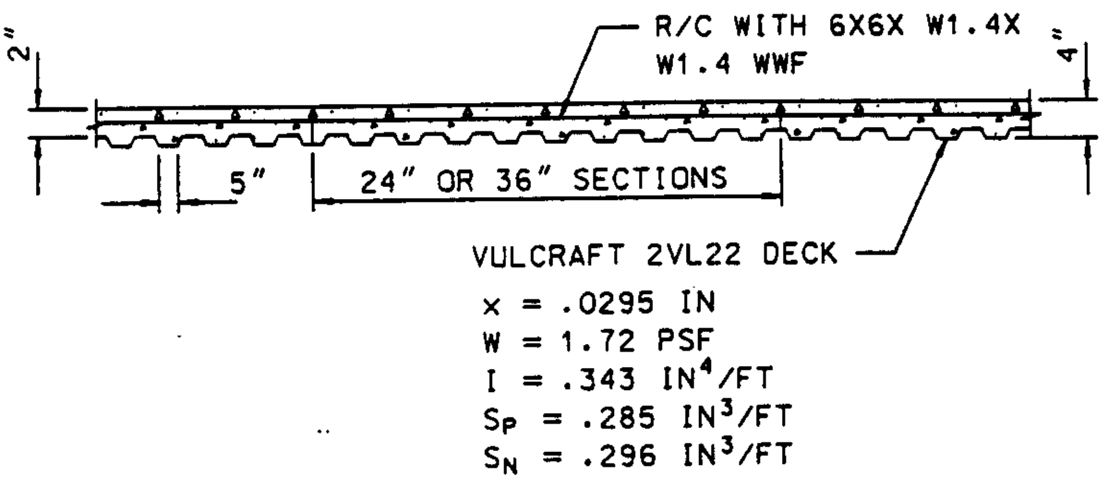
PLAN VIEW



BUILDING SECTION

NOTES:
- 4000 PSI CONCRETE STRENGTH
- 60 KSI REINFORCEMENT STRENGTH
- 2000 PSI COMPRESSIVE CMU STRENGTH
- 200 PSI CMU RUPTURE MODULUS
- 36 KSI STRENGTH FOR STRUCTURAL STEEL

Building Type #12 Plan and Elevation



TYPICAL SECTION THROUGH ROOF DECK

Building Type #12 Detail (continued)

BUILDING NO. 13

This building consists of prestressed concrete double "T" wall and roof sections. This one-story building has 6000 square feet of space, and has a story height of 12 feet. Plan view and sections follow. The dependencies between the roof beams and the wall beams which support them were input by hand into the *Dependencies* spreadsheet because the Preprocessor does not consider the unusual case of vertical, beam-type components in the walls.

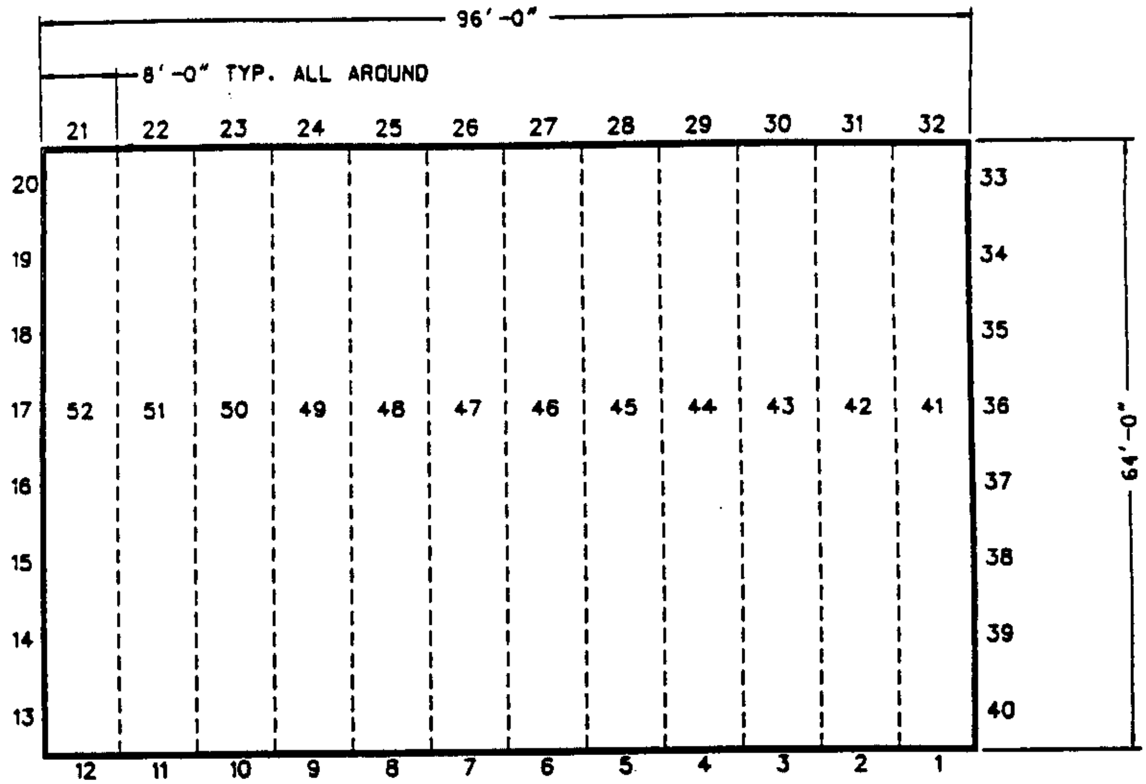
Component

Prestressed double tee roof panels, 64' span, simply supported, 8' width, 8' by 64' loaded areas, 8DT32 with 128-D1 strand pattern, no topping (32" depth).

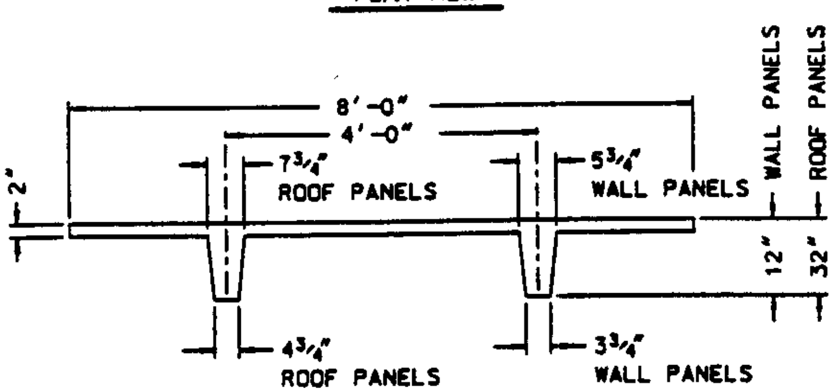
Prestressed double tee wall panels, 12' x 8', simple-fixed support condition, 8DT12 with 28-S strand pattern (12" depth).

COMPONENTS 1 - 40 ARE PRESTRESSED CONC. WALL PANELS. STARTING FROM LOWER RIGHT CORNER, PROCEDING CLOCKWISE.

COMPONENTS 40 - 52 ARE PRESTRESSED CONC. ROOF PANELS. PROCEDING RIGHT TO LEFT.



PLAN VIEW



WALL/ROOF SECTION

NOTES:
- 5000 PSI CONCRETE STRENGTH
- 270 KSI REINFORCEMENT STRENGTH

Building Type #13 Plan and Details

REFERENCES

1. Oswald, C.J., "FACEDAP Theory Manual," Contract No. DACA 45-91-D-0019, U.S. Army Corps of Engineers, Omaha District, April 1993.

2. Skerhut, D. and Oswald, C.J., "FACEDAP Programmer's Manual," Contract No. DACA 45-91-D-0019, U.S. Army Corps of Engineers, Omaha District, April 1993.

3. IOSUB: A Library of Input and Output SUBroutines for use with Microsoft® Fortan, Version 2.0 User's Manual, Business Systems Integration, San Antonio, TX.

4. Empty Shell Version 1.1 Documentation, Kandu, Inc., Hamilton, NJ.

5. WINLAC, WINdow Lite Analysis Code User Documentation, Sverdrup-Embassy Task Group, Washington, D.C., October 15, 1990.

6. Design Manual Army TM5-1300, NAVFAC P-397, and AFM-88-22: Structures to Resist the Effects of Accidental Explosions, Department of the Army, Navy, and Air Force, Washington, D.C., 1990.

7. Security Engineering Manual, U.S. Army Corps of Engineers Missouri River Division - Omaha District Protective Design Center, January 1990.

8. Sprague, E.T.E., 50-YEAR STEEL JOIST DIGEST, A Compilation of Specifications and Load Tables 1928-1978, Technical Digest No. 7, Steel Joist Institute, Myrtle Beach, SC, 1982.

9. Marchand, K.A., Cox, P.A., and Peterson, J.P., "Blast Analysis Manual Part I - Level of Protection Assessment Guide - Key Asset Protection Program Construction Option (KAPPCO)," Contract No. DACW 45-89-D-0139, Final Report prepared for the U.S. Army Corps of Engineers, Omaha District, PD-MCX and U.S. Forces Command Engineer, Fort McPherson, GA, July 19, 1991.