Autodesk® Architectural Desktop

AEC Details eGuide

Based on Softdesk's S8 AEC Tools Details Module(DS) Release 8.x. Portions of this material may not be up to date or apply to the Architectural Desktop version of this software.

Product Overview

The *Details* portion of Architectural Desktop is a separate "module" or program carried over from the Softdesk S8 AEC Tools and has not been fully integrated with ADT. This software is designed for composing 2D construction industry details, such as wall sections or steel connection diagrams. The program is primarily database driven using dBase IV customizable files found in the **Autodesk Architectural Desktop 3\programs\details** folder.
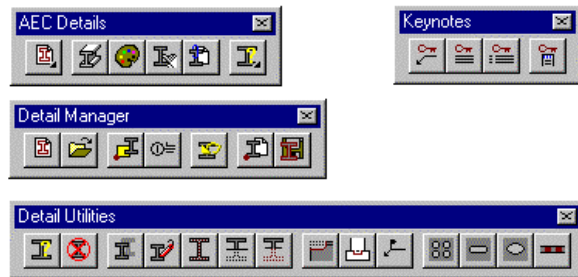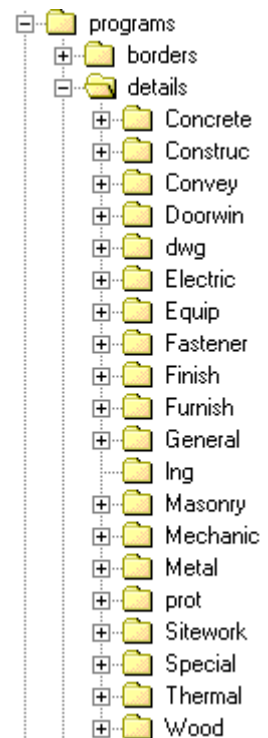
Software data files are separated into 19 folders of which 16 are based on the 16 CSI divisions, with one division for fasteners. There are two other program folders: **General** and **Prot**. General stores generic files which can be used as customization starting points. Prot holds the keynotes.dbf file in its **\zm\lng** folder. These folders store the *programs* used for the materials in that division.

Each *material folder* is further subdivided into 3 sub-folders: **data; dwg; and lng**. *Data* is used to store material libraries, catalogs, tables and supporting slide and slide libraries. *Dwg* holds any drawings referenced by the data files. *Lng* stores the information files (.ifm) for the material division in question.
☞ Note: Some divisions, such as *Masonry*, do not use dwg files, so they have no *dwg folders*.

New material catalogs (.cat) and information (.ifm) files can be be created "on the fly" inside of AutoCAD R14, while library (.lib) and database (.dbf) files must already exist. You must also use one of the supplied lisp programs in the **"division"** folder.

The material data database files can also use certain "reserved" column names to automatically perform certain functions. These names include: "R_SLIDE"; "R_BLOCK"; "R_PROGRAM"; "R_RASTER"; and "R_TEXTFILE". Existing material files can be edited from their respective material tables (once that material has been used).

<u>The Details Module's Use of Database Tables</u>

When creating a "material" in your drawing or attaching a material property to an AutoCAD entity, this "material information" is attached as extended entity data. This allows for the copying of entities with "attachments" within the current drawing, or the WBLOCKing of them to a new file, with no loss of data. Details accesses the database rows through index files (.cdx extension) of the same name. These require a unique **"key column"** of information. The key column must have a unique value for each database row. *Details* manages these keys for all tables except the material .dbf files. The first column of these tables is automatically used as the key. This value is stored with AutoCAD entities as they are created or linked to these tables. Always insure the first column of information clearly identifies the material and that it is not repeated within that table. If tables are changed, be sure to delete any index (.cdx) files. *Details* will automatically create a new index file if one is not found. If rows have been deleted from these .dbf files by an external program, the table should be packed by that program before accessing it with the *Details* program. If columns of a material database file need to modified, an external program, such as Microsoft Excel, is needed.
☙☀ **Warning: Do NOT change the name of or data type of any columns in any table provided with *Details*.**

Material Tables can have additional column of information added, the columns can be rearranged, and rows can be added and rearranged.
☙☀ **Warning: Files not to be edited are *.lib*, *.cat*, and *.pal* files, as well as a few .dbf files**

<u>Composing Materials</u>

Details are composed from materials selected from the above mentioned material tables. To the right is a sample of the Material Retrieval dialog box. This box will be explored further later.
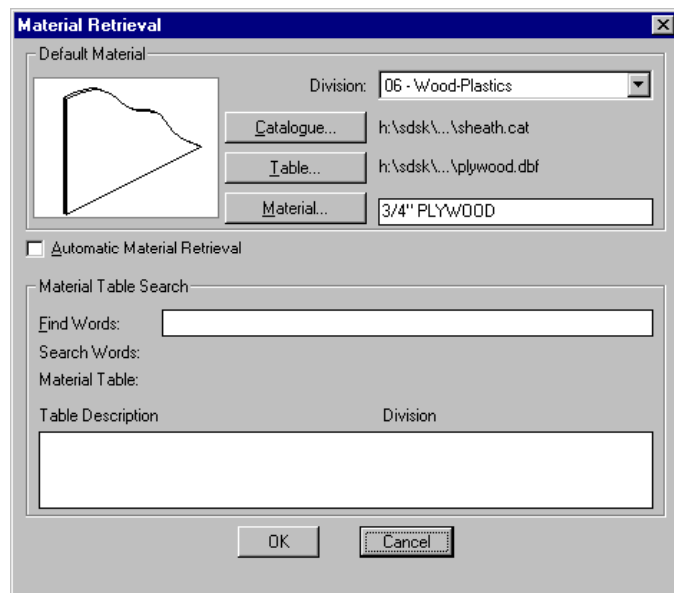
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You select your current material to be drawn from 17 divisions. Within each are one or more catalogues, or subdivisions. For example the Wood Division is subdivided into Finish Carpentry, Lumber, Sheathing, etc. Each of these catalogues is further divided into one or more database material tables. For example, Sheathing has tables for Plywood and Gypsum Sheathing. To access a material table, first choose the proper Catalogue, then Table, then Material.
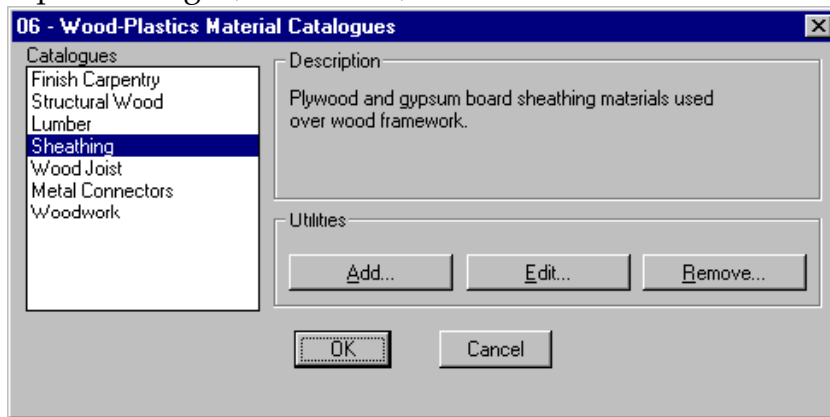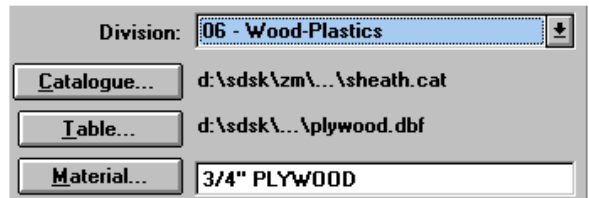
The Catalogue Dialog Box
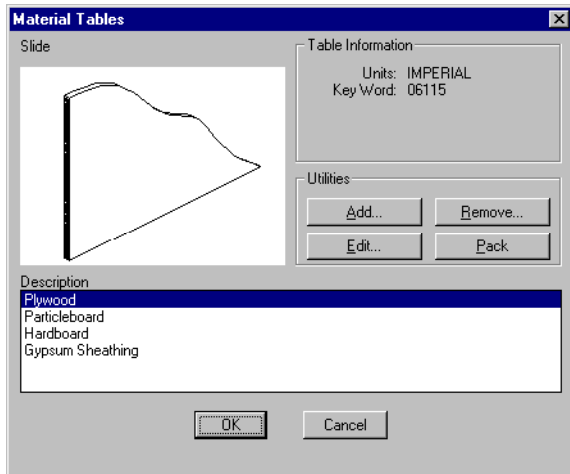
The Table Dialog Box   The Material Dialog Box
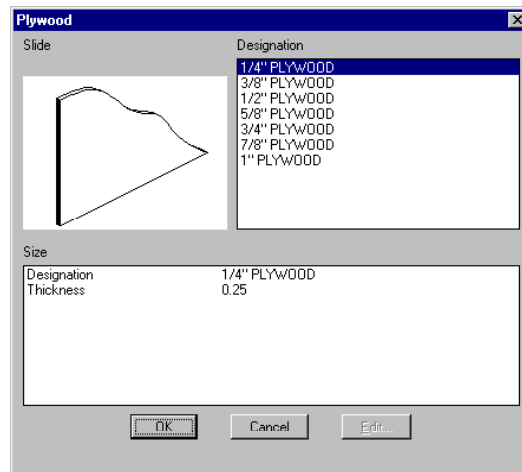
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To access the Softdesk material table database editor, select the edit button in the Material dialog box. You can edit, add new items, copy or delete existing items.
☞ Note: The Change Table button works with the *Material Edit* command
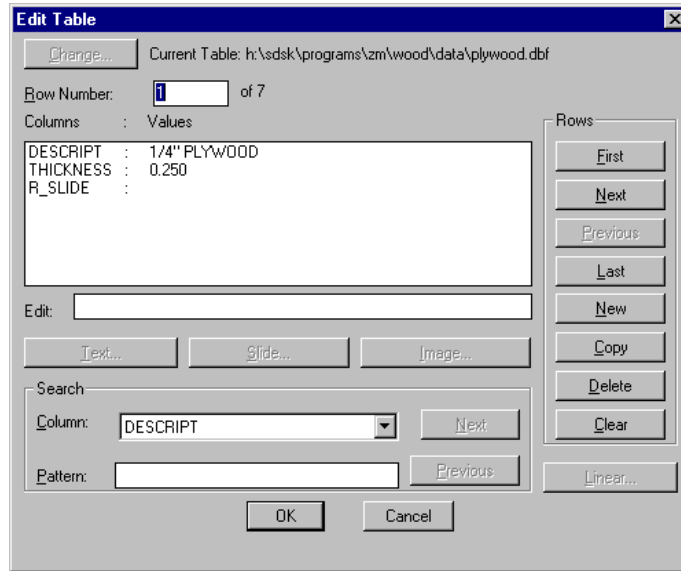
The Edit Table Dialog box

Details Settings

The Details module has been fully integrated into AEC Tools, and therefore its configuration variables are also included with those of AEC Tools. Layer settings are within the AEC Tools Project Settings dialog box. Details layers are divided into the 16 CSI divisions, plus Fasteners. Each division is further divided into 5 layers: wide, medium, thin, fine, and hatch.
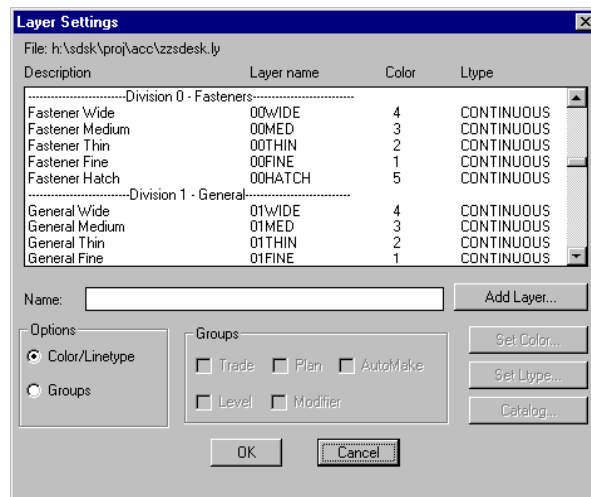
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Settings specific to Details are accessed
from the AEC>Detail Manager>Project
Settings pulldown menu item. There are
many fewer items to configure than in AEC
Tools.
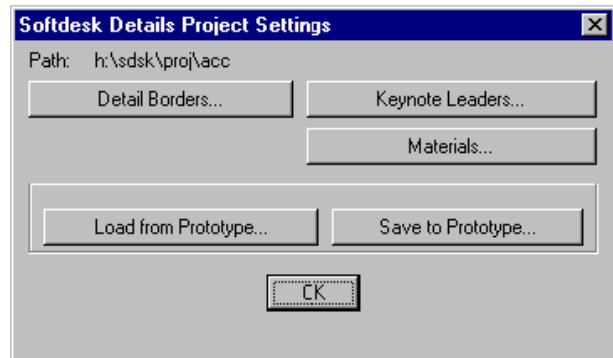
Detail Borders

Detail borders let's you choose the border type to
place around details with the "Create Titles"
command from the AEC>Detail Manager pulldown
menu.
☞ Note: This dialog box can also be accessed from
the Title Settings... option in the AEC>Detail Manager
pulldown menu.
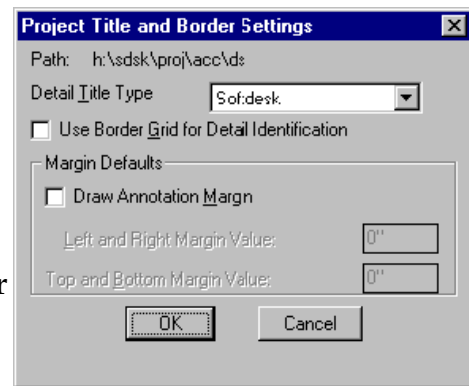
There are 3 types available: Softdesk, AIA ConDoc,
and Title Only. Below is a description and illustration
of each one.
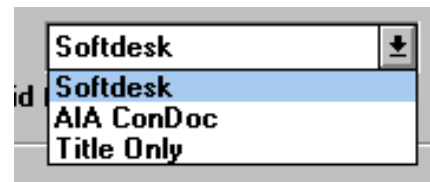
> **Softdesk Title:** This creates a title block and border. Prompts are as follows:
>> Detail identification: this is the character placed in the bubble
>> Detail title: this is placed next to the bubble. The style used is TITLEBLKL.
>> Detail scale <1/8" = 1'-0">: - this is placed below the title. The style used is TITLEBLKM.
>> Detail reference: this is placed in the lower right corner of the border. The style used is TITLEBLKM.
>> Drawing Sheet: prompted for, but not used
>> First corner point: pick the first corner for the border.
>> Second corner point: pick the second corner for the border.
> A border line is drawn on the BORDERLINE layer. If the Margin Defaults toggle
> is on, a rectangle is drawn on the NOPLOT layer within the outer border to the
> "plot width" specified.

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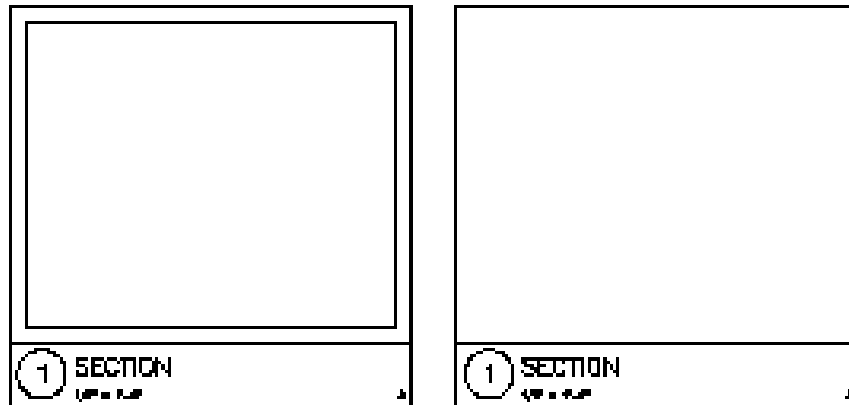
Softdesk with Border   Softdesk without Border

☞ Note: If "Use Border Grid for Identification" is toggled on, the "identification prompt" is skipped initially, and the routine looks for the "row" and "column" number from the sheet border. If not found, you will be prompted to enter a value.

AIA ConDoc: These prompts are similar to the Softdesk detail border, though in a different order.
Detail identification: placed near the bottom left corner of the border
Detail title: placed next to the identification response
Detail scale <1/8" = 1'-0">: - placed below the identification response
Detail reference: placed below the title
Drawing Sheet: prompted for, but not used
First corner point: pick the first corner for the border.
Second corner point: pick the second corner for the border.
A border line is drawn on the BORDERLINE layer. If the Margin Defaults toggle is on, a rectangle is drawn on the NOPLOT layer within the outer border to the "plot width" specified.

☞ Note: If "Use Border Grid for Identification" is toggled on, the "identification prompt" is skipped initially, and the routine looks for the "row" and "column" number from the sheet border. If not found, you will be prompted to enter a value.

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AIA ConDoc with Border   AIA ConDoc without Border

Title Only: This places a bubble with associated text on the drawing. Prompts
are similar to AIA ConDoc.
Detail identification: placed near the bottom left corner of the border
Detail title: placed next to the identification response
Detail scale <1/8" = 1'-0">: - placed below the identification response
Detail reference: placed below the title
Drawing Sheet: prompted for, but not used
Move objects: reposition the title

☞ Note: If "Use Border Grid for Identification" is
toggled on, the "identification prompt" is skipped
initially, and the routine looks for the "row" and
"column" number from the sheet border. If not found,
you will be prompted to enter a value.
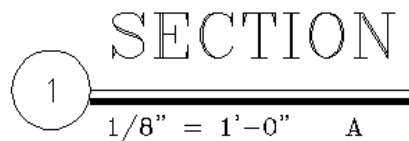
Keynote leaders

This affects how notes are created using
the Keynote Leaders command from the
AEC>Keynoting pulldown menu. This
dialog box can also be access from this
same pulldown menu.Annotations
options include: Key Only; Notes Only;
Key and Notes. Key Word Source
includes options for: Information file for
material; Database table file for material;
and Automatically enter keynotes file.

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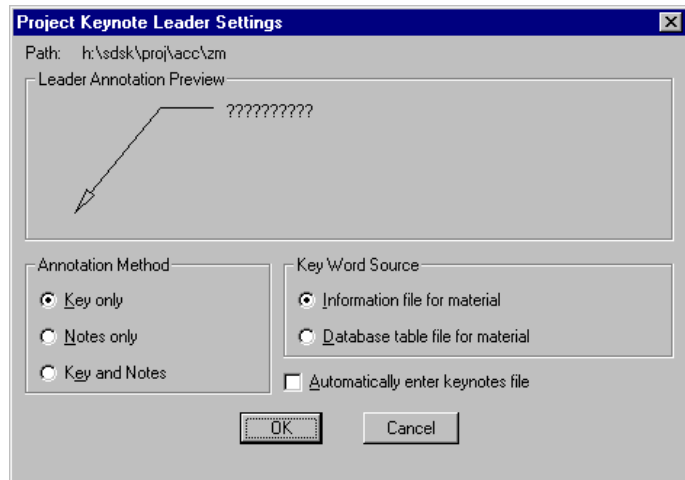
All entities created with *Details* have database records attached to them. Keynoting allows you merely to pick that entity, even in an XREF, and it will be annotated. Options within the Keynote Leader Settings dialog box include:

>   Path: indicates the location of the keynotes.dbf file

>   Key Only: Leader command adds one line of text. Type depends on Key Word Source variable:

>   >   • Information file for material: annotated material key word is retrieved from the *material tables* **.ifm** file. Every material table has an associated **.IFM** file. Each .IFM file contains a keyword specific to that material and is associated with every row in the table file. This option uses this keyword to find the associated keynote in the *keynotes.dbf* file
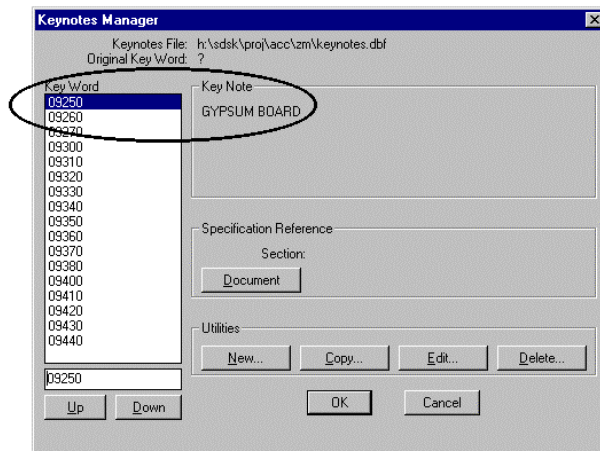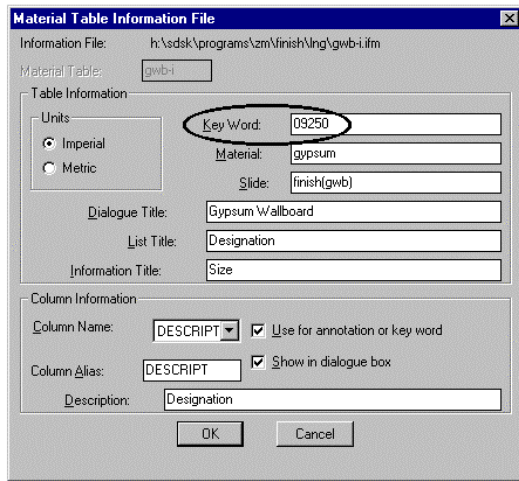
>   >   • Database table file for material: This sets the source for the keyword to a column in the material .dbf file, which is the first column by default, unless the material .IFM file specifies an alternate column.
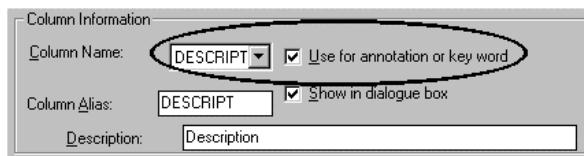
The .IFM File
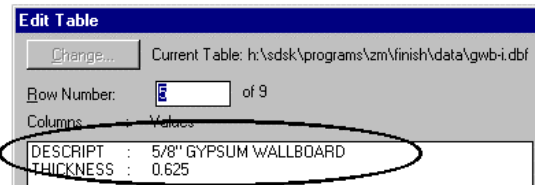
The Database file

>   **Notes Only**: This adds one note to end of leader, based on setting for Key Word Source (.IFM or .DBF file), then provides the option for additional lines of text. If .IFM is toggled, the description comes from the *keynotes.dbf* file.

Key and Notes: When using the database option under Key Word Source, this is no different than notes only. For *.IFM* option, it uses the CSI number in *.ifm* file after placing the leader, then the related text from the *keynotes.dbf* on the next line, then allows you to add additional line(s) of text.

Automatically enter keynotes file: This option automatically opens the Keynotes Manager dialog box, using the current material CSI number as "point of entry" if Key Word Source is set to Information file option, or the beginning of the file if Database table option is used. This provides a method to view, edit, use a different, or create a key and note before it is added to the leader.
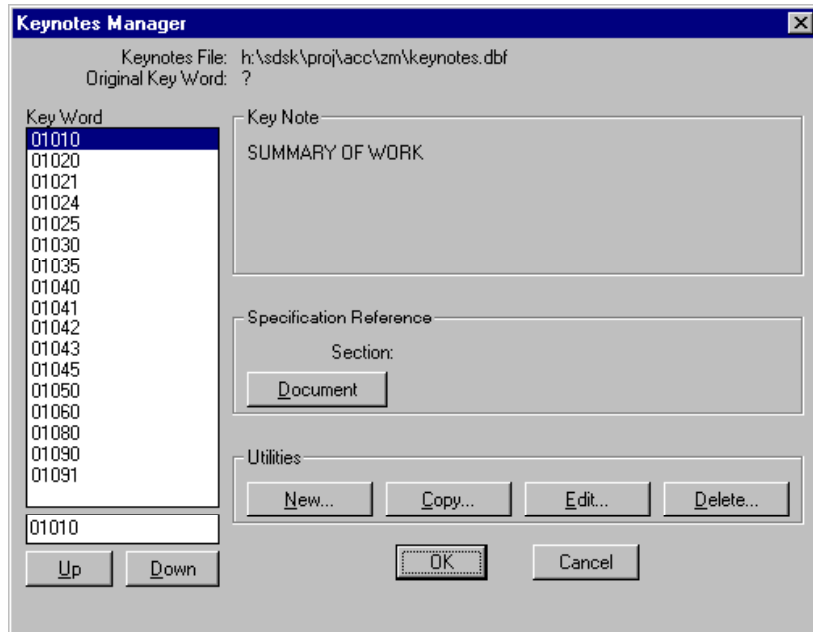
Materials

Materials sets options for material divisions which have them. Current divisions include: Fasteners; Concrete; Metals; Woods-Plastics; and Finishes.

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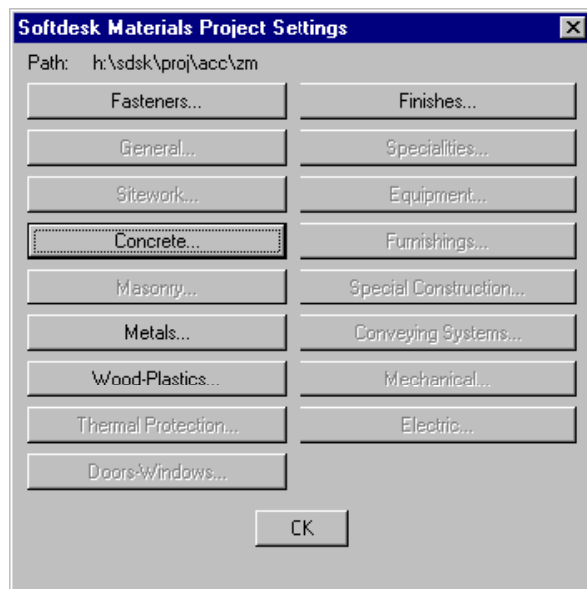
Creating Architectural Details

The Details program allows you one of three ways to create "intelligent" details:

1. You can begin composing a detail at you current drawing scale
2. You can use the Detail Manager to alter the plot scale for the detail you are about to create.
3. You can "attach" database records to existing details to make them *intelligent*.

You access commands for Details in either four toolbars, or from 4 cascading menus under the Details pulldown menu. The toolbars are: AEC Details; Detail Manager; Detail Utilities; and Keynotes, as shown to the right.
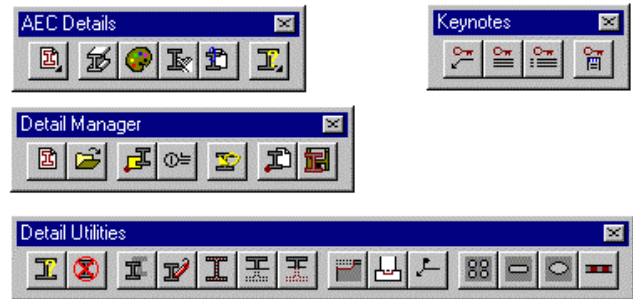
We will cover the casacading menus at the time the command is explained.

Material Compose:
The "heart" of the Details program is the Material Compose command. This command displays the Material Retrieval Dialog box. From here, you can select the material division, catalog, table, and the actual material item. Dialog boxes are provided to help navig ate the different catalogs, tables, and items. The following is one of two methods for finding a material:

• Select the desired material division from the Division drop-down list box.

• If the desired Catalog is not displayed, click the Catalog. The Material Catalog dialog box is displayed.

• If the desired Catalog is displayed, but the desired Table is not displayed, click the Table. The Material Table dialog box is displayed.

• If the desired Catalog and Table are displayed, but the desired Material is not displayed, click the Material. The Material dialog box is displayed.
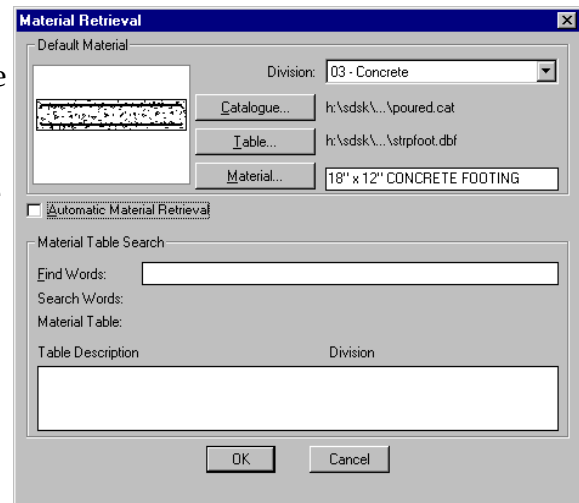
• If the desired Catalog, Table, and Material combination is displayed, click OK to continue. The system returns to your drawing and the associated drawing command is displayed on the command line.
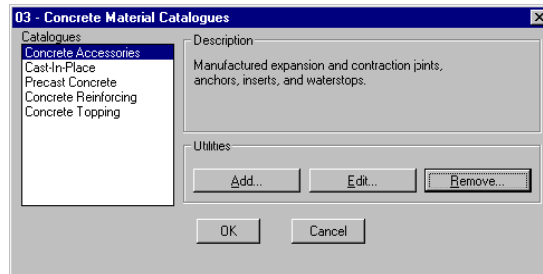
Material Catalogue Dialog Box
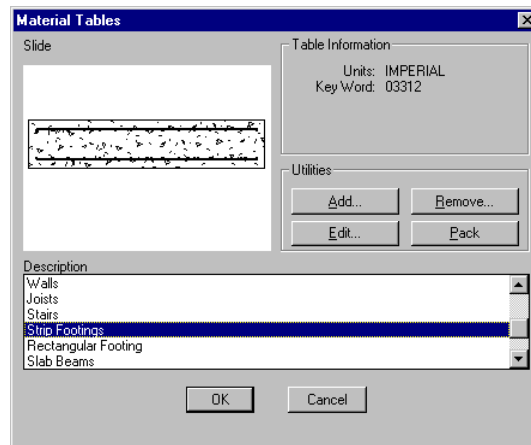
Material Table Dialog Box

Material Dialog Box

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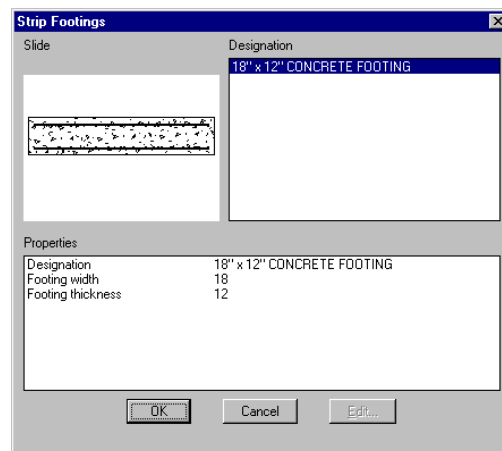
The second method for selecting and placing materials is through the use of the *Find Words* section. The following is a more in depth discussion of each method, as well as the various sections of these dialog boxes.

The Material Retrieval dialog box is divided into two sections which provide two methods of material retrieval. The upper section provides an interface to browse through all the materials installed by identifying each library, catalog, and table to be accessed. The bottom section provides a means of searching for a specific material table based on the type of material desired.
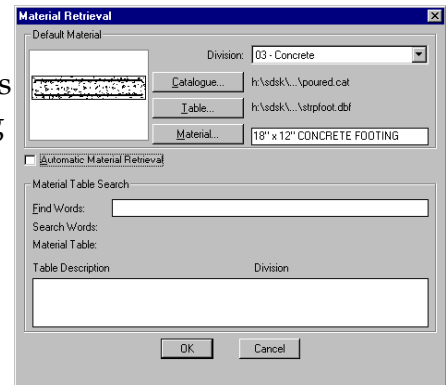
Division: This popup list displays the seventeen material divisions available for selection. When a division is selected for the first time in the current Softdesk project, the Table and Material buttons are grayed out until a specific catalog within the division is selected.

Catalog: Selecting this button displays the Material Catalogs dialog box for the selected division. See Selecting a Catalog below for more information about using this dialog box. After a catalog has been selected, the following Table and Material buttons become active.

Table...: This button displays a dialog of all the material tables contained within the currently selected catalog. See Selecting Material Tables below for more information about using this dialog box

Material...: Selecting this button displays the material's dialog box where a different size or type of material can be selected from the current material table. See Selecting Materials below for more information. The material edit box provides a means of retrieving a material by entering the *material key name*. If the material is found, it is automatically returned and set as the default without opening the dialog box.

A wildcard entry can also be entered in the Edit box to display the material table with only those rows of materials that match the wildcard entry. For example, if a door material table identifies door data with a key column using values such as " 36x68 Door", "36x72Door", and "30x68 Door", where the first element in the value defines the width of the door, making a wild card entry "36*" would only display the Material Table dialog box with 36" wide doors listed. This field accepts any wildcard input which conforms to the formats available in wildcard matching of AutoLisp.

Automatic Material Retrieval: This check box allows faster access to materials by automatically making the catalog or table selection when it is the only selection available. When this check box is on, any dialog box that contains one catalog or table will not be shown. It is necessary to turn automatic material retrieval off in order to add or edit to libraries or catalogs which contain only one entry.

Each time material data is retrieved, it becomes the default for that division and is displayed until replaced with another material. The bottom section, Material Table Search, is used to retrieve material by identifying a material tables base on an associate search phrase or words as described below.

Find Words: This field is used to search all the installed material tables based on patterns of words which are compared to the search words associated with the material tables. This field accepts any wildcard input which conforms to the formats available in wildcard matching of AutoLisp. Entering "door*wood" would display all the tables that have the words "door" and "wood" contained in the search words. For example, the search word shown previously could return tables with search words "Garage doors, wood" or "Exterior doors with all wood construction." Searches are not case sensitive.
☞ Note: Do not use the " (quote) marks when actually entering the search words

☞ Note: Entering an asterisk (*) returns a listing of every table in the Softdesk material database system and provides a quick means to see the scope of materials installed and freely browse and select from them.

Search Words: This field displays the search words that have been defined for the highlighted table in the Table Description area below. The search words are assigned to tables through the Table Add or Edit dialog box accessed through the Material Tables dialog box.

Material Table: This field shows the path and filename of the highlighted table in the Table Description area. This field is for information only and cannot be edited.

Table Description: This box displays the results of a Find Words search. The listing displays the table's description and division in which it was found. If a table is not found which is still referenced by the material database system, the division column will show file not found. *Double-clicking* on a highlighted table displays the Material dialog box for that table and allows a row of data to be selected and retrieved. Material data retrieved through this process is displayed in the Default Material section and becomes the new default for that division.
☞ Note: A single click recovers the current material. You *must* double click!

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Material Catalogues

Material catalog files are designed to group common material tables or a manufacturer's material tables together. There is no limit to the number of material catalog files that can be managed by AEC Tools. Material catalog files are used to identify material table files, and associate the program that is used by Softdesk to use the material. Each material table must have an entry in a material catalog file to be accessed.

Material catalog files conform to industry standard .dbf files but have .cat extension. In order for Softdesk programs to recognize the file as a material catalog, it must be defined with the following structure:

Column Name	Data Type	Width	Decimals
FILE_NAME	CHARACTER	8	0
DESCRIPT	CHARACTER	40	0
R_SLIDE	CHARACTER	18	0
R_PROGRAM	CHARACTER	12	0

☞ Note: The file extension for material catalogs has changed from .dbf to .cat. Previous versions of material catalogs are automatically converted to a file with the new extension.

Material Catalogues are found in the *DATA* folder of the *zm* folder.

Catalogs: This scrolling list shows the names of the material catalogs available for the division. Use this list to select a catalog. These names correspond to the .cat files used to hold the catalog information.
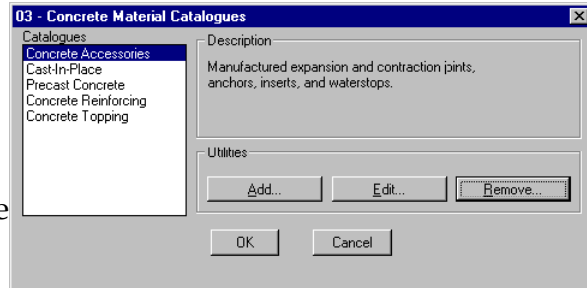
Description: This field shows a brief description of the catalog highlighted in the Catalogs scrolling list. This field can be edited by clicking *Edit....*
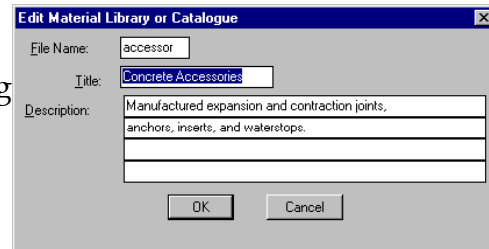
Add...: The Add... button is used to add catalogs to the list of files. When selected, this button displays the Add Material Library or Catalog dialog box.
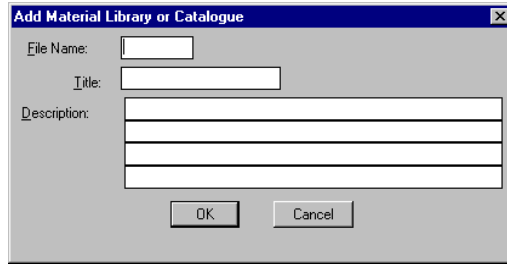
Filename: Enter the name of the .cat or .lib file to add in this edit box. Do not supply an extension with the name. This may be an existing file with the extension *.cat* or *.lib* in the data directory of the directory corresponding to the material division. However, if there is not an existing file with the correct name, the program assumes a new catalog is to be created and displays a dialog box to create it.

☞ Note: New **libraries** cannot be created with this release. A sub-library file with an extension *.lib* must exist before it can be entered into the material database system. *Material library files* manage a division's material catalog files. The catalog files manage material table files which in turn organize materials into logical groups. Each material division's \data folder must have at least the material library file that is known to the program using it.

Material Library Files
The following list provides all the required material library file for AEC Tools and the material keywords used for custom programming to identify material divisions.

Name	File	Keyword
01-GENERAL	general.lib	ZM_GENRL
02-SITEWORK	sitework.lib	ZM_SITE
03-CONCRETE	concrete.lib	ZM_CONC
04-MASONRY	masonry.lib	ZM_MASON
05-METAL	metal.lib and shapes.lib	ZM_METAL
06-WOOD	wood.lib	ZM_WOOD
07-THERMAL	thermal.lib	ZM_THERM
08-DOORS AND WINDOWS	doorwin.lib	ZM_DOOR
09-FINISHES	finish.lib	ZM_FINSH
10-SPECIALS	special.lib	ZM_SPEC
11-EQUIPMENT	equipmnt.lib	ZM_EQUIP
12-FURNISHINGS	furnish.lib	ZM_FURN
13-SPECIAL CONSTRUCTION	construc.lib	ZM_SPEC
14-CONVEYORS	conveyor.lib	ZM_CONVY
15-MECHANICAL	mechanic.lib	ZM_MECH
16-ELECTRICAL	electric.lib	ZM_ELEC
00-FASTENERS	fastners.lib	ZM_FAST

If the files listed above are deleted or renamed, AEC Tools will not be able to access any of the materials within that division.

The material directories are also accessed by other Softdesk programs which may have their own unique material libraries. Additional material libraries may also exist to increase the logical organization of materials. These can be added when necessary as more materials are added. Material libraries used to further organize materials can only be accessed through the material libraries recognized by the current Softdesk program.

Material library files conform to industry standard .dbf files but use a .lib extension. For Softdesk programs to recognize the file as a material library, it must be defined with the structure shown in the following table:

Column Name	Data Type	Width	Decimals
FILE_NAME	CHARACTER	8	0
TITLE	CHARACTER	20	0
DEFAULT	LOGICAL	1	0
DESC_1	CHARACTER	50	0
DESC_2	CHARACTER	50	0
DESC_3	CHARACTER	50	0
DESC_4	CHARACTER	50	0

Areas of customization include, adding additional material catalogs to the material libraries, modification of title and descriptions of the material catalogs, and the addition of sub-libraries to the system.

☞ Note: Two new columns have been added to the material libraries for this release and the file extension has changed from .dbf to .lib. Previous versions of material libraries are automatically converted to the new structure.
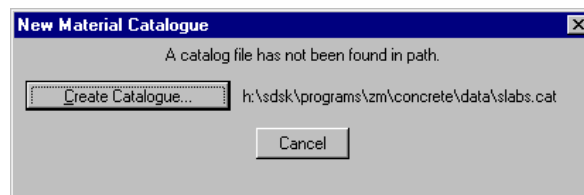
The New Material   Catalogue Dialog box

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Clicking Create Catalog... creates the file and directory, while clicking Cancel returns to the previous dialog box without adding a new catalog file. The *Add Material Table* dialog box appears next.
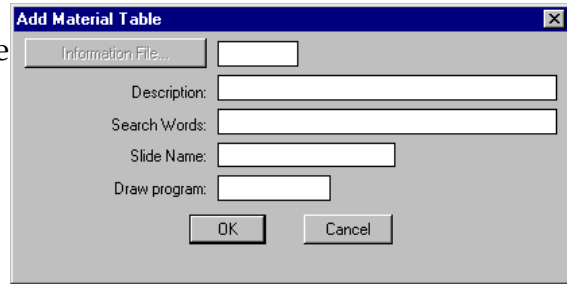
☞ Note: This dialog box is also accessed from the Add... button in the Material Table dialog box

☞ Note: This process does not create the files associated with a new catalog, it merely adds the name to the list of available tables. Make sure the files needed to access materials from the new table are in the correct directories before selecting creating a new catalog

Adding a material table to the Softdesk material database system requires **three** files for AEC Tools to be able to use it. The first file is a **database table** with a *.dbf* extension. This file must be located in the material divisions *data* folder. The second file is the **tables information file** which has the same name as the material table but has an *.ifm* extension. The information file is an ASCII text file and is located in the material divisions *lng* folder. The last file required is a **program file** which is to be used by the Details draw command to create the material object in AutoCAD. The program file must be located in the *material division* folder.

Information File...: Use this edit box to enter the name of the material .ifm file. The .ifm filename is used to identify the material table to add to the current catalog. Do not supply a path or extension with the filename.

Description: Enter a title for the new material table in this edit box. This description is displayed as the table title in the Material Table Viewer dialog box.

Search Words: Enter the words used by the *Find Words* field in the *Material Table Search* section of the *Material Retrieval* dialog box

Slide Name: Enter the name (excluding path and extension) of a slide or reference from a slide library to be displayed for the material table in the Catalog of Material Tables dialog box in this field. The slide name is optional. If this edit box is left blank, a slide is not displayed in the Catalog of Material Tables dialog box. If a slide library reference is used, it must be entered in the format "library (slide)" where "library" is the name of the slide library and "slide" is the name of the slide. The slide file (.sld) or slide library (.slb) must be located in the *data* sub-folder of the folder associated with the current material division.

Draw Program: Enter the name (including extension) of the LISP or ADS program that is associated with the material. This program is used to generate the material. The program file must exist in the folder associated with the material division. This program can be one of the draw programs supplied with the Softdesk material database system, a program supplied from another source such as a manufacturer of the material, or a program created by you.

Click OK to save the table information and exit the Add Material Table dialog box.

Title: Enter a title for the new material library or catalog. This is used for displaying the name in the Catalog list box in the Material Catalogs dialog box.
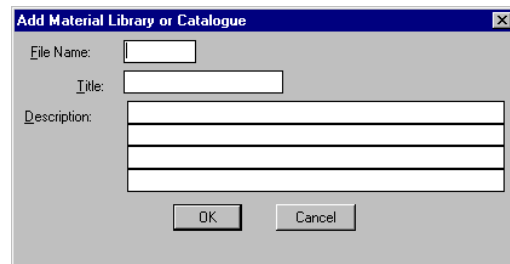
Description: Enter a description for the new catalog in this series of edit boxes. This description is displayed in the Description field of the Material Catalogs dialog box.

Click OK to save the catalog information and exit the Add Material Library or Catalog dialog box. The selected material's prompt sequence for placing it into the drawing starts after you exit the dialog box.

Edit...: The Edit... button is used to edit the currently highlighted catalog. When selected, this button displays the Edit Material Library or Catalog dialog box. All of the fields displayed in this dialog box are identical to the corresponding fields in the Add Material Catalog dialog box
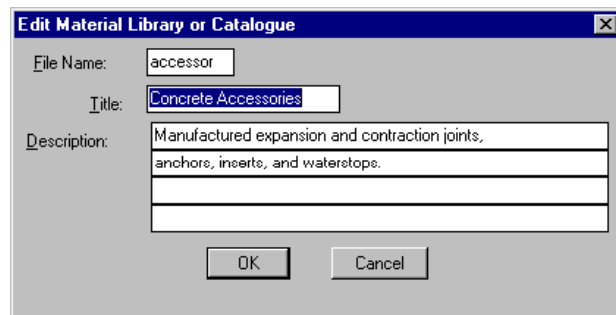
Remove...: The Remove... button is used to remove catalog names from the list of available catalogs in the Material Catalogs dialog box. When this button is selected, the command removes the highlighted catalog from the list of available catalogs. When selected, this option displays a confirmation dialog box before removing the catalog.

☞ Note: This option does not affect any of the files or directories on the hard drive. It simply removes the filename from the scrolling list presented in the Material Catalogs dialog box.

Material Tables

Material Tables group similar types of a material into a single database file. This dialog box presents a list of material tables (.dbf files) available from the current catalog as well as an optional slide showing a general representation of the materials available within a table. All of the options available from this dialog box are described below.
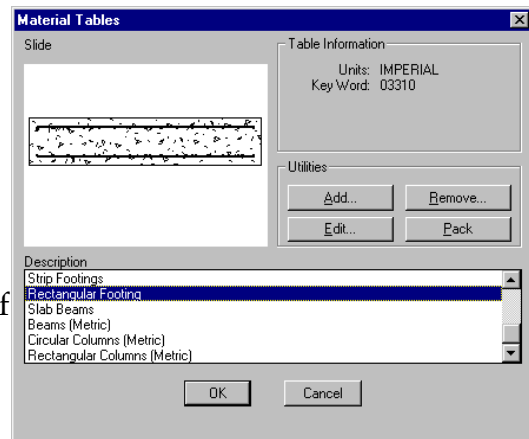
Table Information: This area shows information about the current material table. This information identifies the units that the table's linear values are stored at and the keyword assigned to the table. Units and keywords are retrieved from the material table's *information file*.

Add...: The Add... button is used to add tables to the list of files in the current catalog. See page 17 for more information.

Edit...: The Edit... button is used to edit the currently highlighted table. See page 18 for more information.

Remove...: Click Remove... to remove table na mes from the list of available catalogs in the Catalog of Material Tables dialog box. When this button is selected, the command removes the highlighted table from the list of available tables. When selected, this option displays a confirmation dialog box before removing the table. Click OK to remove the table or click Cancel to exit this option without removing any information.
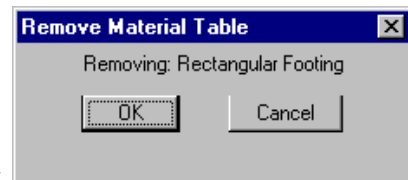
☞ Note: This option does not affect any of the files or directories on the hard drive. It simply removes the filename from the scrolling list presented in the Material Tables dialog box.

Pack: Click Pack to remove all deleted rows of information from the currently highlighted table. A message is displayed at the bottom of the dialog box until the packing is complete. Pack a table when Deleted entries are displayed in the Material Table Viewer dialog box.

💣※ WARNING! If the table you intend to pack is being referenced by other Softdesk applications for use with Landscape or others programs, packing may corrupt the attachment of a material to an AutoCAD entity. If that occurs, the attachment will have to be made again.

Material Dialog Box

This dialog box displays all of the materials available from the current material table. None of the information can be edited in this dialog box. To edit information from a material table, select the Edit... button.
☞ Note: The Edit... button will be grayed out the first time you access this dialog box. Pick OK, then re-enter this material to enable this button
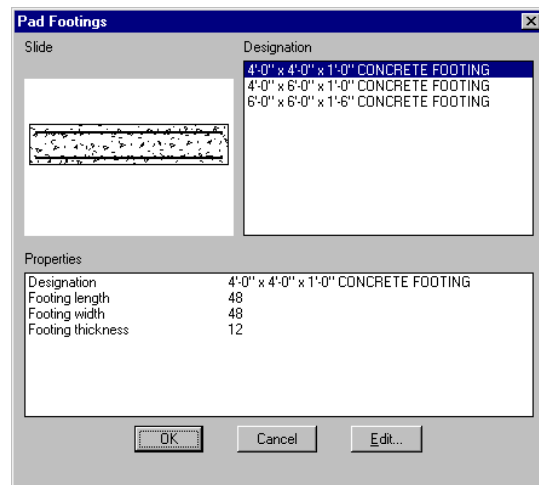
Description: This scrolling list displays all of the materials available from the current table. The title for this list comes from the *.ifm* file and varies depending on the material table selected. Select the material to use from this scrolling list.

☞ Note: This list is built from the first column of the material table.

Slide: This image tile displays the designated slide (if any) for the currently highlighted material. This can either be an independent slide or part of a slide library. This slide representation of the material is entirely optional. The slide is determined by the name found in the R_SLIDE column of the material table. This provides the ability to display a unique slide for each row of information. If no slide is defined for that material, you can either display a slide that is a default for the entire material table as defined in the *.ifm* file or leave this image tile blank.

Size: This area displays information pertaining to the currently highlighted material. The title for this area comes from the *.ifm* file and varies depending on the material table selected. This area can display any information about selected materials such as manufacturer's names, model codes, sizes, or cost information. The descriptive prompt and visibility of each row can be controlled by elements of the *.ifm* file.

Edit... When selected, this button displays the Edit Table dialog box with the current material table.

☞ Note: Deleting rows of material does not remove the data until a pack is run on the file. Pack can be performed from the Material Tables dialog box which references the table.

The Edit Table dialog box displays the contents of the table file one row at a time. The rows display individually and each column occupies a separate line. The user can scroll through the columns of a particular row or through the available rows.
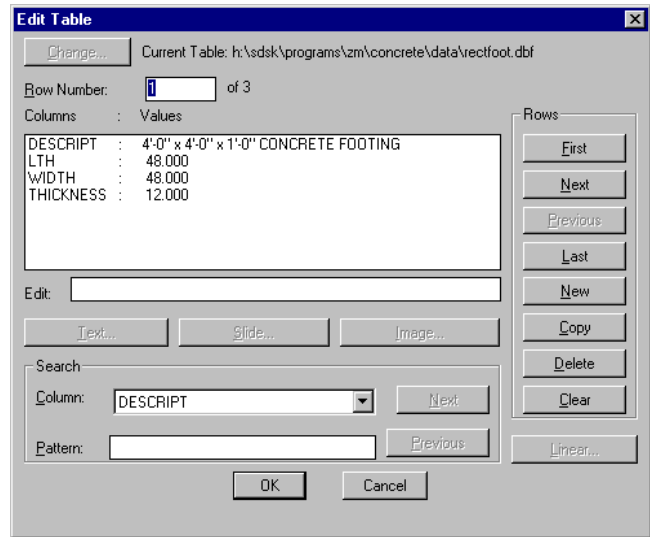
Row Number {X} of {X}: This edit box displays the number of the current row and the total number of rows in the table. When the number of the desired row is entered, the specified row data displays automatically.

Row Contents: The list box near the top of the Edit Table dialog box displays the contents of the current row. Highlighting a column places the value in the Edit box and can be edited as desired.

Edit: This edit box displays the contents of the currently highlighted column. Editing each column is different depending on the data type of the column. Each data type is described in the following table.

DATA TYPE	DESCRIPTION
CHARACTER	This is an editable scrolling string. Depending on the size of the column the entire string may or may not be visible.
DATE	Enter dates in the following form: YYYYMMDD.
NUMBER	For integer numbers only digits are allowed. To enter values for the decimal portion of a real number enter a decimal point (.).
LOGICAL	Press Spacebar to toggle between true (T) and false (F).

Press ESC to quit any edits without saving them. Pressing ENTER saves the edits and updates the row accordingly.

Rows: This section contains buttons to move through the existing rows and create or delete rows.

☞ Note: Deleted rows are not shown.

First: This button moves to the first row in the table.

Next: This button moves to the next row in the table.

Previous: This button moves to the previous row in the table.

Last: This button moves to the last row in the table.

New: This button creates a new, blank row in the table. The row is either appended to the end of the table or inserted after the current row location, depending on the setting of Insert Rows.

Copy: This button creates a new row by copying the current row. The row is either appended to the end of the table or inserted at the current row location, depending on the setting of Insert Rows.

☞ Note: AEC Tools does not directly provide a way to adjust the *Insert Rows* setting. This is controlled in the *database/query* program, which is no longer "supported" by Autodesk, but can still be used in AEC Tools

Delete: This button deletes the current row from the table.

Clear: This button clears all the values in the currently displayed row.

Text: This button displays a text file associated with the row information, provided it is properly associated with the **R_TEXTFILE** reserved column name. The value in the column is the name of the text file or the AutoCAD command reference for a text file that is in a text file library.

☞ Note: Make sure a word processor has been selected for Text editor. This setting is configured through the External Programs option of the Project Settings... command. If no word processor is specified in this field, an Error dialog box is displayed.

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Slide: This button displays a slide associated with the row information, provided it is properly associated with a reserved column name. Slide files are attached when the reserved column "**R_SLIDE**" is present and there is a value for it in the current row. The value in the column is the name of the slide file or the AutoCAD command reference for a slide file that is in a slide library.
Image: This button displays a raster image associated with the row information, provided it is properly associated with a reserved column name. Raster files are attached when the reserved column "**R_RASTER**" is present and there is a value for it in the current row. The value in the column is the name of the raster file.
Linear: This button is disabled in Details. It is available in **Database/Query**

☞ Note: The following items are disabled in Details. They work in the **Database/Query** program, and their descriptions are provided for information purposes only.

Search: The settings in this section search the specified column for a specified pattern. Each time a match is found the row contents are displayed in the row listing area.
Column: This list box displays all the valid column names that can be used for pattern searches. The selected column is used for the pattern search.
Pattern: Enter the value to search for, then press ENTER. An index file is generated for the specified column and the first row in the table that contains the value is found and displayed in the Select Row dialog box. If the specified value is not found, the value that is the closest to it displays.

Each time a new column is searched, a unique index file is created provided an index file for that column does not already exist. An index file is valid only for the current Select Row dialog box. A new index is generated each time the dialog box displays.

Next: This option searches for the next row in the table that contains the value specified with the Pattern setting or is the next closest. If a pattern has not been entered, this button is inactive and, as a result, is grayed out.
Previous: This option searches for the previous row in the table that contains the value specified with the Pattern setting or is the next closest. If a search specified has not been entered, this button is inactive and, as a result, is grayed out.

The .IFM Information File

Information files provide Softdesk programs with critical information about how data table files should be treated, displayed, and processed. These information files are in standard ASCII text format and have an *.ifm* file extension. They can be edited with any standard text editor.
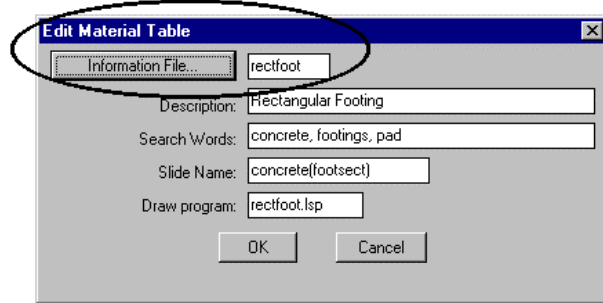
The information file has a very complex structure and feature set. Each Softdesk application will only require or support certain elements of the information file. The program provides a dialog which can be used to edit or create an information file for material tables.

☞ It is recommended that this interface be used since it isolates only those features of the information file that are supported by AEC Tools and makes changes easier to understand than directly editing or creating an information file with an editor.

Each material table file used by AEC Tools must have an accompanying information file (filename.ifm) located in the material division's \lng folder. The Material Table Information File dialog can be accessed through the Information File... button located on the Add or Edit Material Table dialog box.
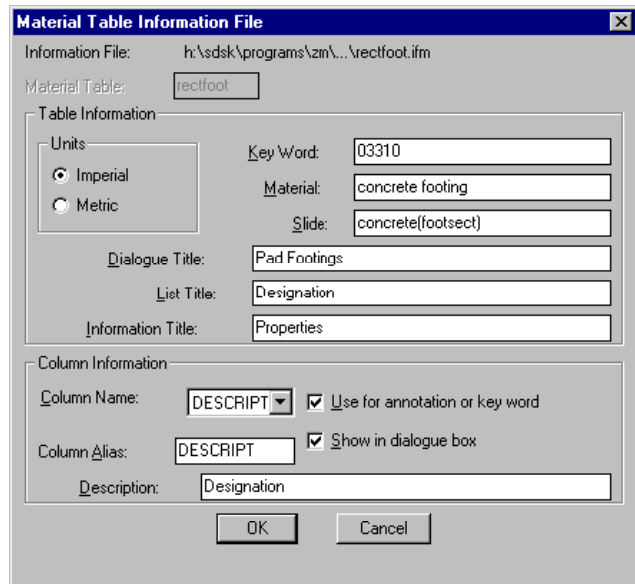
The Material Table Information File dialog box provides input to all features of information files currently supported by AEC Tools. The dialog box provides three areas of information. The top two lines provide information about the filename currently displayed. The center section defines information that applies to the entire table. The bottom section defines information for the individual columns defined in the material table.

The first line in the dialog box shows the complete drive, path and filename of the information file currently displayed. The second line is an edit box with the name of the material table file the information file is associated with. This box is not enabled since this version of AEC Tools requires the material table name to be identical to the information filename.

Units: These radio buttons are used to identify if linear values in the material table are metric or imperial. If imperial units are set, it is assumed all linear values are in decimal inch format. Metric tables assume that all linear units are in millimeters. If any material table does not represent those units correctly, the draw programs will not be able to draw them accurately or may even fail. This restriction only applies to those columns of information that are used to draw the material.

Key Word: This edit box is used to identify a key word that applies to the entire table. The key word links the material table to a specific entry in the keynotes file. The keynotes file associates the key word to a paragraph of text. This link is used to create annotation, charts, and external files for reports and specification reference. Softdesk provides a keynotes file based on the CSI Broad Scope or Medium Scope number for the materials. This can be replaced with any method you wish to implement.
This key word will be associated with all drawings created from any row of data in the material table. A table column can also be identified for use in keynoting.

Material: This edit box is used by Softdesk programs to identify the material defined in the table and insure the selected table is correct for the program intended to use it. This is an English descriptor and must remain unchanged. This applies to materials tables supplied by Softdesk. If you are creating or adding new material tables which do not use Softdesk programs you can ignore this value. If the material tables are used by Softdesk programs, look up the material descriptor supplied with the Softdesk tables used by that program and use it in the new table.

💣 Note: Any changes to the material value causes programs to reject the data as invalid.

Slide: This edit box identifies the name of the slides to display by default in the Material Table Viewer dialog box. The slide name can be a reference to an individual slide (.sld) or a slide from a slide library (.slb). The slide or slide library must reside in the \data folder with the material table. This slide is associated to all of data in the table, a table column can also be identified for use in getting the slide name.

Titles: The next three edit boxes provide the text which is displayed as titles in the Material Table Viewer dialog box beginning with the dialog title. The *Dialog Title* edit box contains the text which is displayed in the title area of the Material Table Viewer dialog box. This should describe the type of material defined in the table.

The *List Title* edit box contains the text which is displayed as the title above the scrolling list of values in the Material Table Viewer dialog box. The scrolling list in the Material Table Viewer is constructed from the first column of the material table being shown. This title should describe the value of the data found in the first column of the material table, such as Model Number or Item.

The *Information Title* edit box contains the text which is displayed as the title above the scrolling list of descriptions and values in the Material Table Viewer dialog box. The scrolling list in the Material Table Viewer is constructed from the columns of data associated to the currently highlighted item listing box. This title should describe the contents of the data being displayed, such as Dimensions and Design Properties or Model Specification.

The last section, **Column Information**, provides support for identifying the use of each column in the material table.

Column Name: This list contains a list of all the columns in the current material table. Highlighting each column name updates the adjacent features in this section to show the current settings associated with the selected column. The settings supported for each column are described below.

Column Alias: Softdesk programs use an alias name instead of the actual table column name when retrieving a row of data from a material table. The Column Alias edit box provides a means for assigning aliases to the columns in a material table. Aliases provide two features. It allows any column name to be used in the material table and still be recognized by Softdesk programs. The second feature is used to identify columns for special use.

The alias is an English descriptor and **cannot be changed**. All materials tables supplied by Softdesk use the same name for both the column name and the alias. If you are creating or adding new material tables which do not use Softdesk programs, you can ignore this value. The alias will match the column name by default. If the material tables are to be used by Softdesk programs, look up the material aliases supplied with the Softdesk tables and apply it to the appropriate column in the new table.

Currently AEC Tools uses two special aliases to identify a material table column for slide filenames and AutoCAD drawing names for block insertion. The **R_SLIDE** alias is used to identify a column with slide names which are to be displayed in the Material Table Viewer for the selected row of data. This provides a means of displaying a unique slide for every row in the material table rather than using the default slide defined in the table information section. The slide name can be a reference to an individual slide (.sld) or a slide from a slide library (.slb). The slide or slide library must reside in the \data directory with the material table. The **R_BLOCK** alias is used to identify a column of AutoCAD drawing names which are to be inserted by the drawing program. The column identified for blocks provides the ability to associate an AutoCAD drawing with each row of data in a material table. The column must contain a valid drawing name without a path or extension. The drawing must be stored in the drawing folder (\dwgs) of the material's division.

☞ Note: The column identified by the *R_BLOCK* alias only works with programs written to retrieve the block name and insert them.

Description: This edit box provides a means of adding a brief description to each column. The description along with the column value from the highlighted data is displayed in the information scrolling list in the Material Table Viewer dialog box. The description should be limited to twenty-eight characters to avoid truncation in the Material Table Viewer dialog box.

Use for annotation or key word: This check box identifies a column for use as the key word linking each specific row in the material table to a specific entry in the keynotes file. The first column is used by default if another column has been specified. Only one column can be toggled on for this support. If a new column is selected, the previous column is automatically deselected. This feature is most commonly used when the leader keynotes option are set to use the data row description column for annotation Show in dialog box: This check box applies to each column and simply controls the visibility of the description and value shown in the information scrolling list of the Material Table Viewer dialog box. This provides a means of suppressing information that is not a factor in making a selection for drawing.

When a new material table is added to the system, an information file is automatically created if one is not found for it. The new information file displays question marks in all the edit boxes in the table information section. The column information section uses the column name for both the alias and description values. The first column in the table is designated as the column for use as the key word or annotation, and all columns are set to being visible in the Material Table Viewer dialog box.

Softdesk material draw programs will validate that the selected data from the material table has the required aliases defined in the tables information file. If any errors are detected, the Missing Alias Identification dialog box is displayed.

Material Palettes

When using the Details program, you may find that you use a group of materials repetitively. While the Material Retrieval dialog box provides an excellent method for selecting rarely used materials, it will become "tiresome" to use with commonly used materials. The Material Palette is the solution to this problem. The Material Palette command provides a dialog box for accessing often-used materials quickly. It should be used to store groups of materials that are used together to produce details.

Any number of material palettes can be made an
d stored, only one palette is active at a time. This command can be accessed from the AEC Details toolbar, or the AEC>Detail Utilities>Material Palette pulldown menu.

When the command is selected for the first time in a project, the dialog box is empty. Click New to name a new material palette or Open to select an existing palette.
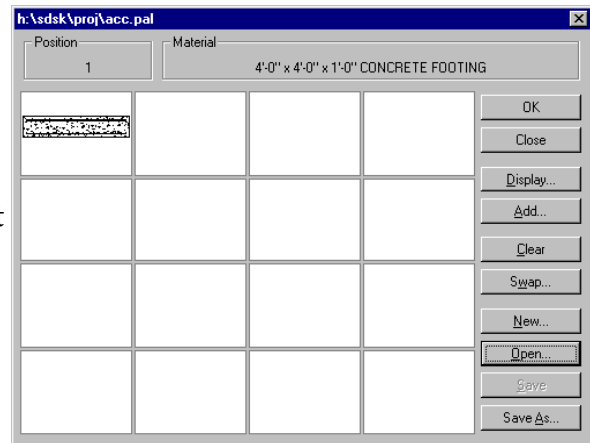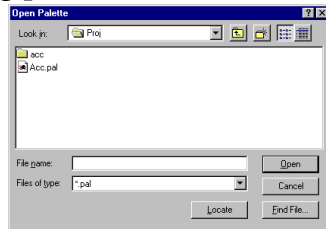
The following are descriptions of the various options within this dialog box.

Palette File: This informational field shows the location and filename of the current palette.

Material: This informational field shows the material number and description of the highlighted material in the Material Palette dialog box. Each material is numbered from left to right and top to bottom. This number can be used to access the material at the command prompt.

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OK: Click OK to accept the current material, exit the dialog box and begin the Draw command. If any changes were made to the palette, a dialog box is displayed to allow the changes to be saved or discarded before closing the dialog box.

Close: Click Close to discard any changes made in the palette, close the dialog box and return to the command prompt.

Add: Click Add to add a new material to the palette. When selected, the *Material Retrieval* dialog box displays. When a material and OK have been selected, the material is placed in the currently selected area of the Material Palette dialog box. If the current area is already occupied with a material, a confirmation dialog box is displayed before the new material replaces it.
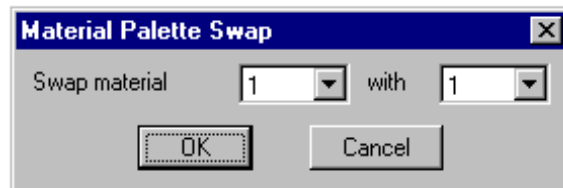
Clear: Click Clear to clear the currently selected material area of the dialog box.

Swap: Click Swap to change the location of two materials in the palette by swapping them. The following dialog box is displayed for choosing the two materials to swap. Use the dialog box to indicate the material area numbers to swap.

New...: Click New to display the AutoCAD New Palette File dialog box for starting a new palette file. There is no restriction to the location or number of palettes created. If a palette is not current when selected, the *project folder* is used as the default folder. If a current palette does exist, its file path is used as the default for the dialog box.

Open...: Click Open to display the AutoCAD Open Palette File dialog box for opening an existing palette file. If a palette is not current when selected, the project directory is used as the default directory. If a current palette does exist, its file path is used as the default for the dialog box. See the AutoCAD documentation for more information about this dialog box.

Save: Click Save to save the current palette configuration.

Save As..: This button saves the palette under a different file name and makes that file the current one. The AutoCAD Save Palette File dialog box opens to create or select a new palette file. If a palette is not current when selected, the *project* folder is used as the default folder. If a current palette does exist, its file path is used as the default for the dialog.

Pick Material Object: The Pick Material Object command is the fastest way to select and draw another piece of a material in the current drawing. The command sets the current material division, catalog, table, and material by selecting an entity from the drawing, and starts drawing that material. When selected, the Pick Material Object command displays the following prompt.

       Pick material object:

Select an entity that has a material table row attachment. The attachment is used to set the current material. If an entity without an attachment is selected, the following prompt displays.

       Object is not identified as material.

If more than one object is picked up by the selection set, the command highlights each entity in turn and displays the following prompt.

       Select this object (Yes/No) <Yes>:

If the highlighted object is the one to query, accept the default of Yes. If it is not, enter N for No in response to this prompt. The command continues to highlight objects until the desired one is selected or until there are no more objects in the selection set. Once the correct object has been selected, that material's division library, catalog, table, and row are set as current and the Draw command is started.

This command can also be accessed from the AEC>Detail Utilities>Pick Material Object pulldown menu.
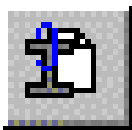
Attach Current Material: You do not have to draw all aspects of your details using database files. You can draw simple AutoCAD entities and use this command to add "intelligence" to them later on by attaching rows of material tables to selected entities. This command replaces all entities selected with a reference to the current material even if the selected entities already reference a material.

The Attach Current Material command uses the row of the current material for the attachment. When selected, this command prompts for selecting the objects for attachment.
☞ Note: To properly set your current material, use the Pick Material Object command before this command

Query Material Object:
The Query Material Object
command displays the
attached row information for selected materials, using the Material Table Viewer dialog
box. This command will only work with entities that have material row information
attached to them using either the Draw or Attach command from any materials menu.
When selected, the Query Material Object command displays the following prompt.

> Pick a material object:

Select the material to query. The material can be a part of an existing detail inserted as a

block or xref. If the object selected is not attached to
a row of a material table, the command displays the
following prompt.

> Object is not identified as material.

If more than one object is picked up by the selection
set, the command highlights each entity in turn and
displays the following prompt.

> Select this object (Yes/No) <Yes>:
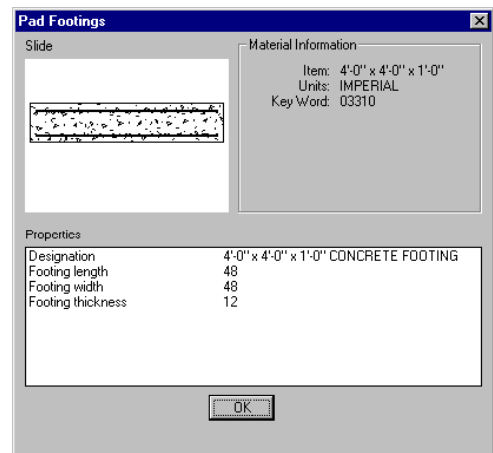
If the highlighted object is the one to query, accept the default of Yes. If it is not, enter N
for No in response to this prompt. The command continues to highlight objects until the
desired one is selected or until there are no more objects in the selection set. Once the
correct object has been found, the command displays the material's dialog box with the
information pertaining to the selected object.

Click OK to return to the drawing. The command continues to prompt for objects to
query until ENTER is pressed as the response to the Pick a material object: prompt. This
command can also be accessed from the AEC>Detail Utilities>Query Material Object
pulldown menu item.

Remove Material Attachment: This command removes attachments between
entities in the drawing and material table row information. When selected,
this command prompts for the objects from which to remove the attachments.
The command then goes through and removes the attachment from the
selected entities. This command can also be accessed from the AEC>Detail
Utilities>Remove Material Attachment pulldown menu item.

Material Edit: This command is accessed from the AEC>Detail Material Edit pulldown menu item. It accesses the Edit Table dialog box for the current material.

☞ Note: To properly set your current material, use the Pick Material Object command before this command.

See pages 21 through 23 for a description of the Edit Table dialog box. The only button not active before is the *Change...* button at the top of the dialog box. This allows you to switch to a different *material table* within the current *catalogue* for editing.
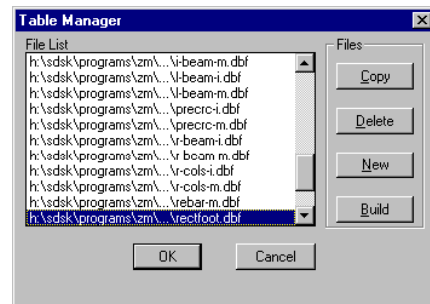
Layer Set: This command is accessed from the AEC>Layer Set pulldown menu item. It accesses the Layer Set dialog box. AEC Tools' Details program uses five pre-defined layers when creating material for each division. There are four layers defined to provide consistent plotted line weight and one layer used exclusively for hatching.

This command is provided for setting correct layers while editing or drawing new objects which represent materials.
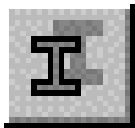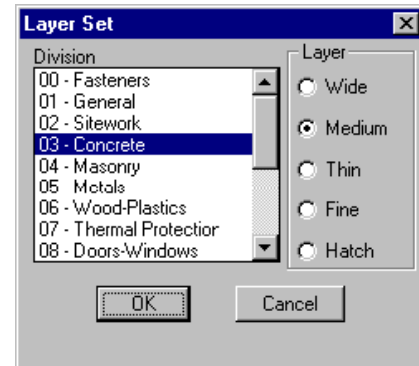
Adjust Material: The Adjust Material command moves or changes a material once it has been placed in the drawing, using the same placement options as when the material was drawn.

☞ Note: Materials do not have to be edited using this command, standard AutoCAD editing commands will work just as well.
After selecting the desired materials, or entities, the command allows for the copying, moving, flipping, and rotating of the selected materials. This command copies, flips, moves, and rotates existing entities. The following prompts display.

      Select objects:
      Base point:
      New base point (Copy/Xflip/Yflip/Move/Rotate/Quit) <Quit>:

This command can also be accessed from the AEC>Detail Utilities>Adjust Material Object pulldown menu item.

Explode Material: The Explode Material command explodes selected materials drawn with AEC Tools. When selected, the command prompts for a material to explode.

As the material is exploded, the new individual entities are attached to the same row of the material table as the original material. This ensures that all material information is retained by the exploded material components. All commands from the AEC Tools program, including the Query Material Object and Pick Material Object commands and the annotation tools, continue to recognize these new material elements. This command can also be accessed from the AEC>Detail Utilities>Explode Material pulldown menu item.

☞ Note: Using the standard AutoCAD EXPLODE command will destroy the material information attached by AEC Tools. If this has happened, used the Attach command to reattach the lost information.
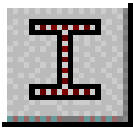
Hatch Material: The Hatch Material command copies hatch patterns while maintaining the material information associated with the hatch pattern. There is also a feature used to create spaces within a hatch pattern. The shape to cut out must be completely bounded by the hatch pattern for this to work properly.

> Pick hatch to edit (or Copy):

☞ Note: To Copy a material, use the *C* option

> Pick entities to avoid...
> Select objects:

☀ Warning: This command occasionally produces unpredictable results. If it does not work as expected, use alternate hatch commands then attach the correct material to the hatch entity. If the boundary for hatching is erased in the process, use the OOPS command (**OP** keyboard macro) to restore it.

This command can also be accessed from the AEC>Detail Utilities>Hatch Material pulldown menu item.
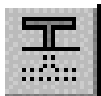
Hide Material: The Hide Material command places entities, selected by signifying a cropping box, onto a hidden line layer. The following prompts display.

> Select hiding edge(s)
> Select objects:
> Select objects to hide:

If a polyline is selected, the following prompt displays.

Do you want to explode selected POLYLINE <Yes>:

The hidden entities are placed on their material division's THIN layer and their linetype is changed, by entity, to HIDDEN.

☞ Note: The results of the change may not be apparent if you are drawing a detail on a drawing with a large scale factor because the hidden line segments are too long to display the spaces. However, when the detail is complete and reinserted into the drawing for plotting, the hidden lines will display correctly.

This command can also be accessed from the AEC>Detail Utilities>Hide Material pulldown menu item.
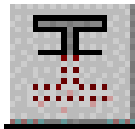
Trim Material: The Trim Material command trims selected entities by selecting a cutting edge and the entities to trim. This command is different from the standard AutoCAD TRIM command in three ways: First, the material information attached to entities is transferred to any entities that are created by the Trim Material command. Second, the AutoCAD block can be trimmed. To do this, the program uses the Explode Material program to explode the block and reattach the material information to the new entities, then the Trim Material command continues on individual entities. Third, AutoCAD Release 12 hatch patterns can be trimmed. To do this, the program explodes the hatch pattern, and finds the point in the pattern furthest from the point picked from trimming. It then attempts to use the BPOLY program to find a boundary for the hatch, if one is found, the hatch will be remade in it. The following prompts display:

Select cutting edge(s)
Select objects:
Select material to trim:

☞ Note: Using the standard AutoCAD TRIM command may destroy the material information attached by AEC Tools. If this has happened, use the Attach command to reattach the lost information.

This command can also be accessed from the AEC>Detail Utilities>Trim Material pulldown menu item.

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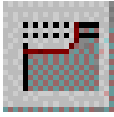
Cut/Cope Steel: The Cut/Cope Steel command is used to draw cuts (copes or blocks) in steel members. It adds the cut, and trims material lines that intersect the cut. This command also provides the option of leaving entities uncut. The following illustrations show an example of a cut.
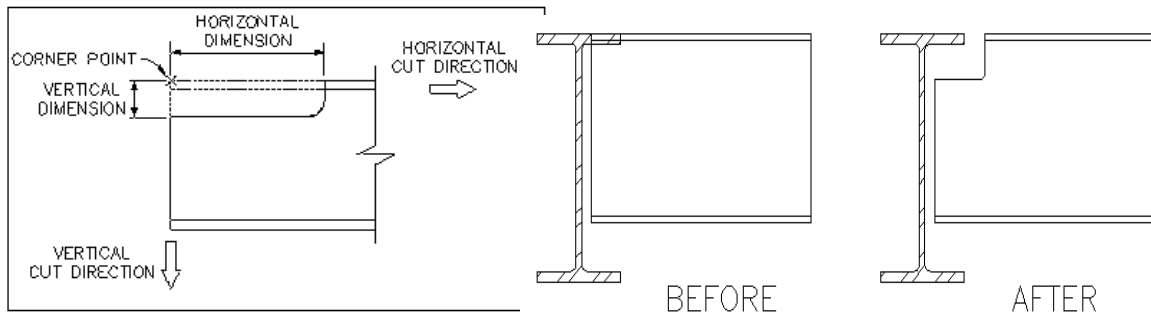
When selected, this command first prompts for points defining the location and orientation of the cut.

> Corner point:
> Vertical cut direction:
> Horizontal cut direction:

The corner point is the angle corner for the cut. The vertical and horizontal cut directions define the angle and direction of the cut for both of the legs. The command then prompts for the length of each leg.

> Vertical dimension <{Length}>:
> Horizontal dimension <{Length}>:

The default values are based on the distances selected when the vertical and horizontal cut directions were specified. The vertical and horizontal dimensions are the lengths of the legs whose orientation was defined by the vertical and horizontal cut directions. A polyline is then drawn on the layer specified for Metal medium with a filleted inside corner of 1/2" . The command then prompts for the removal of entities to leave uncut.

> Remove objects that don't get cut.
> Select objects:

Once all desired materials have been selected, the command trims the remaining intersecting materials around the cut.
This command can also be accessed from the AEC>Detail Utilities>Cut/Cope Steel pulldown menu item.
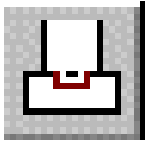
Concrete Keyways: The Concrete Keyways command inserts keyways into existing lines and polylines.

When selected, the command first prompts for the selection of the line or polyline in which the keyway is to be inserted.

Pick concrete at center of key:

The selected object must be a line or polyline. The command places the keyway at the point on the line at which it was selected. The command then prompts for the width and depth of the keyway.

Key width <3 1/2">:
Key depth <1 1/2">:

Either enter values or select points representing the values in response to these prompts. The command then prompts for the direction for the keyway.

Direction of key:

This is usually a direction perpendicular to the selected line or polyline. The command then inserts the keyway at the point selected on the line and prompts for another object. Press ENTER to exit this command.

☞ Note: If there is a hatch pattern, you can edit it with either the *Trim Material* or *Hatch Material* command.

This command can also be accessed from the AEC>Detail Utilities>Cut/Concrete Keyways pulldown menu item.
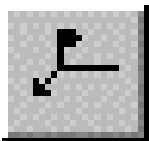
Welding Symbols: The Welding Symbols command creates standard symbols for welding, brazing, and nondestructive examination. The welding symbols are based on the American National Standards Institute (ANSI) and American Welding Society (AWS) standards for both imperial and metric units. The notation conventions used in the welding interface come from ANSI/AWS A2.4-86. All dimensions in this section are given in decimal inches or millimeters, but will be drawn in the drawings current units.

The welding symbols are drawn on the layer for Welding Symbols and the text in the symbols are placed on the layer for Welding Text. The text style used with welding symbols is defined by Weld Symbol Style found in the Text Styles section of the Softdesk Details Project Settings. All welding symbols are made of blocks stored in the zw\dwg folder.

Create Welding Symbols

When the Welding Symbols command is selected, the following prompt is displayed.

Welding symbol (New/Previous/Default/Leader/Multiple/Edit) <New>:

New: When you select this option, the Welding Symbols dialog box displays with the last welding symbol that was drawn. After setting the options for the weld, click OK to draw the welding symbol at the center of the screen. Once completed, you can move the welding symbol to another location. After it is placed, the Leader command is automatically run to add leaders to the end of the welding line.

Previous: This option draws the last welding symbol used in the project without displaying the Welding Symbols dialog box.

Default: This option draws the welding symbol that is saved as the Default welding symbol in the project file. See the section on saving welding symbols for more information.

Leader: This option prompts you to pick a welding symbol to add additional leaders to. It will then find the correct end of the weld line for the leader, and start the Leader command.

Multiple: This option is used to add multiple reference lines to a welding symbol to indicate a sequence of operations. Pick the welding line to add an additional reference line to. The Welding Symbols dialog box is displayed with the last welding symbol drawn displayed. Compose the welding symbol for the next operation, and then click OK. The new reference line and welding symbols will be drawn 1/2" or 12.5 mm from the line selected, and in line with the first segment of the last leader is drawn from the original weld line. Move the new weld line to a new location if necessary, or press ENTER to accept the default location. A line is drawn to connect the two weld lines together.

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Edit: This option prompts you to select a welding symbol to edit. When a welding symbol is selected, it is displayed in the Welding Symbols dialog box. After making the necessary changes, click OK to erase the old welding symbol, and then to draw a new one in its place. If parts of the welding symbol have been moved off the weld line, the program may not find them for editing.

☞ Note: The edit option only works on the individual reference lines of a multiple line welding symbol.

Welding Symbols Dialog Box

The Welding Symbols dialog box allows for the quick and easy creation and editing of welding symbols. As elements are selected, a sketch of the proposed welding symbol is created in the upper right-hand corner of the dialog box. The letters that display on the sketch are place holders for the actual values entered in the dimension and note fields of the dialog box, and may be left blank in the final drawing of the welding symbol. To use the welding symbols interface, start by selecting a direction and a length for the weld line that you are adding. Then select the weld symbols that will be added to the weld line, including any preparation and/or finish symbols necessary for the weld. Up to five welding symbols, and a default welding symbol can be saved for later use.
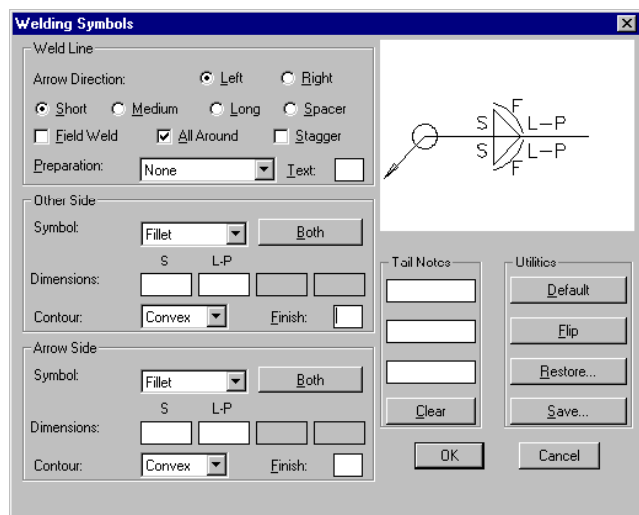
Weld Line
The options in this section of the dialog box control the length and direction of the weld line in the drawing, and is used to add field weld and all around symbols, as well as adding symbols for weld preparation that are added to the weld line.

Left - Right: These buttons determine the end of the weld line that the leader will be drawn from. If a tail is specified, the tail is located at the opposite end of the weld line.

Short: This radio button places the weld symbols on a 1" or 25mm weld line but does not change the sketch in the dialog box.
Medium: This radio button places the weld symbols on a 0.8" or 20mm weld line but does not change the sketch in the dialog box.

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Long: This radio button places the weld symbols on a 1.2" or 30mm weld line but does not change the sketch in the dialog box.

Spacer: This radio button places groove welds on a 1.2" or 30mm weld line with a spacer symbol inserted in the weld line. When using this option, only the V, Bevel, U, and J groove welds can be specified. If any other weld symbol is selected, the weld line will automatically reset to Long.

Field Weld: This adds a field weld flag at the head the weld line pointing towards the tail.

☞ Note: The field weld flag is drawn 0.10" or 2.5mm taller than the AWI specifications to improve the composition of welding symbols that use it.

All Around: This adds an all around symbol to the head of the weld line.

Stagger: This causes the Arrow Side symbol to be drawn 0.10" or 2.5mm to the right of the symbol on the Other Side on the weld line. This does not change the sketch in the Welding Symbols dialog box.

Preparation: This popup box lists all the weld preparation symbols available to be drawn on the weld line. This box is only active if there is at least one side of the weld line without a weld symbol. If both sides of the weld line are empty, then the weld preparation symbol is placed on the Arrow Side. If a weld symbol is later added to the Arrow Side, the weld preparation automatically flips to the Other Side. If a weld symbol with only a weld preparation symbol on the Other Side is needed, place the preparation on the Arrow Side, and select Flip to flip the symbol to the Other Side. If a weld preparation has been selected, and weld symbols are placed on both sides of the weld line, the weld preparation is turned off. In this case, a multiple reference line welding symbol may be needed.

Text: This edit box is used to add the optional notes for Back Welds and Backers symbols.

<u>Other/Arrow Side</u>
The Arrow Side is the bottom of the weld line and the Other Side is the top of the weld line. Otherwise, the following fields function the same way in these sections of the dialog box.

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Symbol: This popup box lists all the valid weld symbols available to be drawn on the weld line. As different symbols are selected, the dimension labels are modified to reflect the dimensions available for the particular weld. Text, for nondestructive examination symbols, can be placed on the line using the weld symbol, Text.

Both: This button copies the weld symbol and all values from the current side of the weld line to the other. This button is disabled for stud welds and surfacing welds that can only be placed on the Arrow Side of the weld line.

Dimensions: These boxes are used to fill in the dimensional information for the welding symbol. The program automatically converts fractions such as 1/4 to the Softdesk small fraction format ~4 if that option is selected and imperial units are being used. The program also adds the degree symbol to angular dimensions. All other annotation must be entered by you, such as adding the for the number of spot welds, or the plus sign (+) symbol for flange symbols. The notation above the edit boxes corresponds to the text positions in the sketch.

Contour: This is used to select the finish contour symbol that will be added to the weld symbol. The placement and orientation of the contour symbol is automatically based on the weld symbol.

Finish Text: This edit box is used to enter the method of obtaining the desired contour for the weld (i.e. type G for Grind).

Tail Notes
The options in this section of the dialog box are used to add tail notes to the welding symbol.

Tail Notes: The edit boxes in the tail note section are used for adding tail notes to the welding symbol. Adding text to the boxes adds a tail to the sketch. The number of notes in the tail (up to three) is determined by the number of lines of text that have been entered in this section. When entering tail notes, fill the edit boxes from top to bottom.

Clear: This button removes the tail from a weld symbol, and clears all text in the Tail Notes edit boxes.

Utilities
The options in this section of the dialog box are used to save and restore welding symbols and to flip welding symbols from one side of the weld line to the other.

Default: This option clears all values from the Weld Symbol dialog box and loads the welding symbol saved as the Default Weld.

Flip: This option flips weld symbols from the Other Side of the weld line to the Arrow Side and visa versa. The button is disabled when stud and surfacing welds that are on the Arrow Side are selected.

Restore...: This option is used to restore a saved welding symbol. See the Restoring Welding Symbols section.

Save...: This option is used to save the current welding symbol to the project file. See the Saving Welding Symbols section.

OK: This option saves the current welding to be used as the starting point for the next use of the Welding Symbols dialog box, or as the Previous option from the command line. After the welding symbol is saved, it is drawn.

Saving Welding Symbols

Each of the radio buttons correspond to a welding symbol number to save. As the buttons are selected, the sketch of the welding symbol currently stored at that position is displayed. If no weld symbol has not been stored, an empty weld line is displayed.
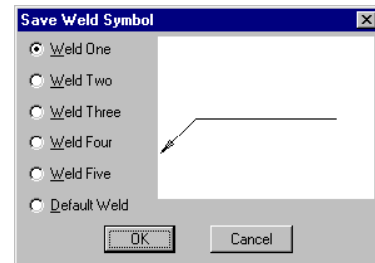
The welding symbols are saved in the file \zw\weld.dfm stored in the current project folder.

Default weld: When the current welding symbol is saved in this position, it becomes the Default welding symbol for the project.

OK: This saves the current welding symbol to the position selected and returns to the Welding Symbols dialog box.

Cancel: This returns to the Welding Symbols default file, and does not save the current welding to the project file.

Restoring Weld Symbols

Each of the radio buttons correspond to a stored
welding symbol. As the buttons are selected, the sketch
of the welding symbol is displayed.

OK: This button loads the currently displayed weld into
the Welding Symbols dialog box.

Cancel: This button returns to the Welding Symbols dialog box with the values
unchanged.

Miscellaneous Compose Detail Commands

Within the AEC>Compose Details> cascading menu are five commands for creating
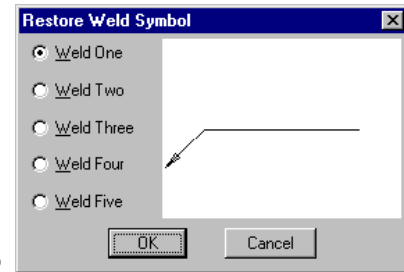miscellaneous steel details. These are covered in the following section.

The **Steel Base Plates Top** command inserts a top view of a base plate. The following
illustration shows an example of this view.
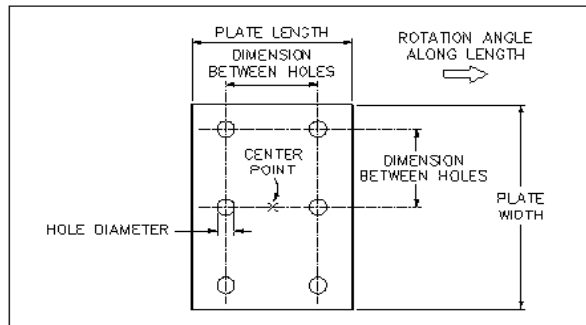
When selected, this command first
prompts for a width and length for the
base plate.

              Plate width <1'
{300}>:

              Plate length <1'-4" {400}>:
Enter values for the width and length, or select points demonstrating these dimensions.
The command then displays the following prompts.

              Plate thickness <1" {30}>:
              Hole diameter <1 1/16" {32}>:
Either enter values or select points demonstrating these dimensions. The command then
prompts for the number of holes along the width of the base plate.

              Number of holes along the width <2>:
This value must be positive and non zero. If this value is two or greater, the command
then prompts for the distance between the holes.

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Dimension between holes:

Either enter a distance or select two points demonstrating the distance between holes. The command then repeats this set of prompts for the holes along the length of the base plate.

Number of holes along the length <2>:
Dimension between holes:

The command then prompts for an insertion point for the base plate and a rotation angle along the length of the plate.

☞ Note: Verify that Osnaps are off before hole placement. (You can use the **F3** key to turn them of or verify that any are set)

Center point of baseplate:
Rotation angle along length <0>:

Once these have been selected, the command inserts the base plate symbol with the appropriate hole symbols. The following prompt displays to place another base plate.

Copy objects:

After a baseplate has been placed or copied, press ENTER. Additional options are presented which allow you to manipulate the basepoint or exit the command.

New base point (Copy/Xflip/Yflip/Move/Rotate/Quit) <Quit>:

The **Steel Base Plates Width and Length** commands insert base plates as seen along their width or length, depending on which command you choose. The following illustration shows an example of both width and length views.
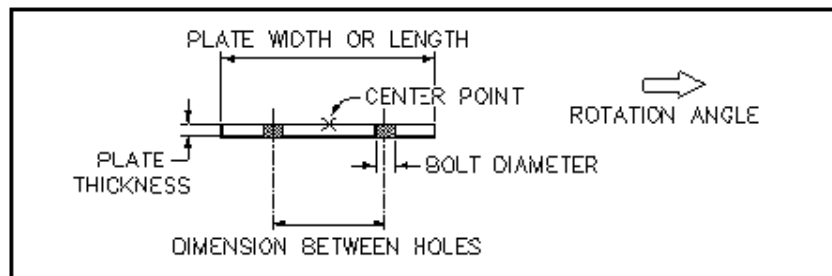
Remaining prompts for this command are similar to the Steel Base Plates - Top command.

Steel Shear Plates: The Steel Shear Plates command is used to add shear plates to a detail. The following prompt displays.

View (Face/Edge/Top) <Face>:

Each of the prompt options are explained in the following sections.

Face: The Face option inserts a face view of a shear plate. This includes any bolts used for the plate. The following illustration shows an example of this view.

When selected, the command first prompts for the number and type of holes (bolts).

Number of bolts <{#}>:
Bolt <{Default}> (Select/Table):

☞ Note: If no bolt has been previously used, you will be prompted to select on from a Material Table

The command then prompts for a number of rows of holes.

Number of rows (1/2) <1>:

This number can either be one (1) or two (2). If two rows of holes are specified, the command then prompts for the gage distance for the holes.

Gage dimension <3" {75}>:

If one row of holes is specified and the number of holes specified is two or greater, the command prompts for the distance between the holes.
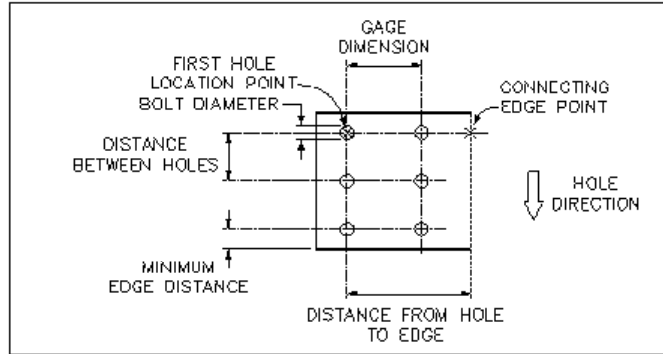
Dimension between holes <3" {75}>:

If more than one hole is to be inserted, the command then prompts for the minimum distance between the holes.

Minimum edge distance <1 1/4" {40}>:

After the information for the holes has been entered, the command prompts for the thickness of the shear plate.

Plate thickness <1/2" {13}>:

The command then prompts for an insertion point for the shear plate. This is the location of the first hole (closest to the edge) in the plate.

First hole location point:

The command then prompts for a connecting edge point.

Connecting edge point:

The command then prompts for the distance from the first hole to the edge of the plate.

Distance from hole to edge <{Distance}>:

The default distance is based on the connecting edge point selected. If one row of holes and more than one hole is being placed, the command prompts for a direction for hole placement.

Show direction of holes <270>:

Select a point or enter an angle for the direction the holes are to be placed. The command then inserts the shear plate with the appropriate number and placement of bolt heads.

Edge: This option inserts an edge view of a shear plate. This includes any holes used for the plate. The following illustration shows an example of this view.
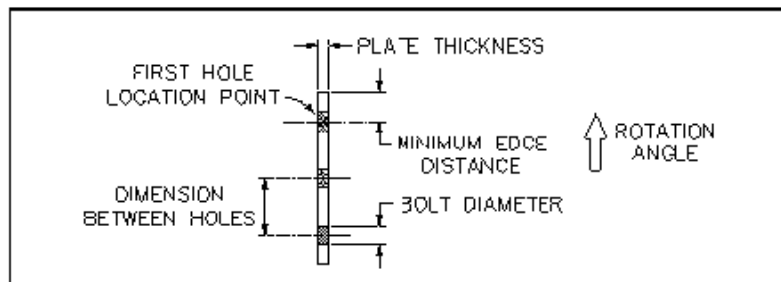
When selected, the command first prompts for whether or not the plate is skewed.

Is plate skewed <No>:

The command then prompts for the diameter of the holes to be placed and the number of holes.

Hole diameter <13/16" {22}>:

Number of holes <2>:

Either enter values for these prompts. The value entered for the number of holes must be positive and non zero. If the plate is skewed, the command then prompts for a number of rows of holes.

Number of rows (1/2) <1>:

This number can either be one (1) or two (2). If two rows of holes are specified, the command then prompts for the gage distance for the holes.

Gage between holes <3" {75}>:

If one row of holes is specified or if the plate is not skewed and the number of holes specified is two or greater, the command prompts for the distance between the holes.

Dimension between holes <3" {75}>:

If more than one hole is to be inserted, the command then prompts for the minimum distance between the holes.

Minimum edge distance <1 1/4" {40}>:

After the information for the holes has been entered, the command prompts for the thickness of the shear plate.

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Plate thickness <1/2" {13}>:
If the plate is not skewed, the command then displays the following prompts.
Plate width <6" {150}>:
First hole location point:
Rotation angle <90>:
Is plate hidden <No>:
The first hole location point is the location of the first hole (closest to the edge) in the plate. If "Y" for Yes is entered in response to the "Is plate hidden" prompt, the command inserts the plate using a hidden linetype. If the default of "No" is accepted, the command places the plate using the linetype designated for the appropriate layer. After all information has been entered, the command inserts the shear plate with the appropriate number and placement of holes.

If the plate is skewed, the command prompts for an insertion point that locates the edge of the plate and the location of the first hole (closest to the edge) in the plate.
Edge and first hole location point:
The command then prompts for the direction and angle of the skew for the plate.
Direction of skew <{#}>:
Angle of skew <{#}>:
Use the pointing device to show the direction and angle of the skew graphically. Alternatively, enter coordinates for the direction and a value for the angle of the skew. The following illustration shows an example of a skewed shear plate.
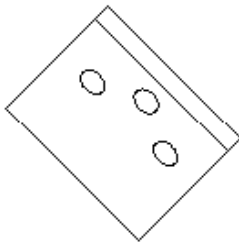
The command then prompts for the distance from the first hole to the edge of the plate.
Distance from hole to edge <{Distance}>:
The default distance is based on the connecting edge point selected. If one row of holes and more than one hole is being placed, the command prompts for a direction for hole placement. Additional options are presented which allow you to manipulate the basepoint or exit the command.
New base point (Copy/Xflip/Yflip/Move/Rotate/Quit) <Quit>:

Top: This option inserts a top view of a shear plate. This includes any holes used for the plate. The following illustration shows an example of this view.
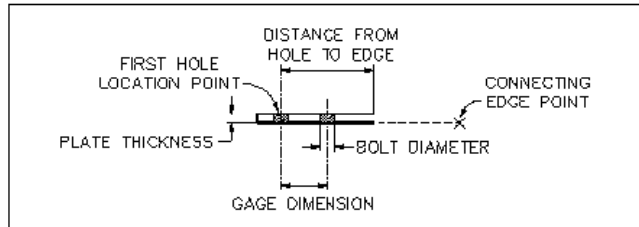
When selected, the command first
prompts for the diameter of the holes to be placed and the number of holes.

Hole diameter <13/16" {22}>:
Number of holes <2>:

Either enter values for these prompts. The value entered for the number of holes must be positive and non zero. The command then prompts for a number of rows of holes.

Number of rows (1/2) <1>:

This number can either be one (1) or two (2). If two rows of holes are specified, the command then prompts for the gage distance for the holes.

Gage dimension <3" {75}>:

If one row of holes is specified and the number of holes specified is two or greater, the command prompts for the distance between the holes.

Dimension between holes <3" {75}>:

If more than one hole is to be inserted, the command then prompts for the minimum distance between the holes.

Minimum edge distance <1 1/4" {40}>:

After the information for the holes has been entered, the command prompts for the thickness and length of the shear plate.

Plate thickness <1/2" {13}>:
Plate length <11 1/2" {305}>:

The command then prompts for an insertion point for the shear plate. This is the location of the first hole (closest to the edge) in the plate.

First hole location point:

The command then prompts for a connecting edge point.

Connecting edge point:

The command then prompts for the distance from the first hole to the edge of the plate.

Distance from hole to edge <{Distance}>:

The default distance is based on the connecting edge point selected. The command then inserts the shear plate with the appropriate number and placement of holes. Additional options are presented which allow you to manipulate the basepoint or exit the command.

New base point (Copy/Xflip/Yflip/Move/Rotate/Quit) <Quit>:

Steel Connection Angles: The Steel Connection Angles command inserts steel angles. When selected, the command first prompts for the view to draw.

> Angle view (Section/Elevation) <Section>:

The command then determines if the current material is an angle shape. If it is not, the command determines if any angles have been drawn previously in the drawing session. If angles have been drawn, the command defaults to the last angle type drawn. If no angles have been drawn, the command displays the following informational prompts.

> Angle data not found.
> Enter to select.

Press ENTER to select an angle material. The program is looking for the materials from the Steel Shapes Material Catalog. If the current material is a valid angle, the command displays the following prompt.

> Angle <{Material Name}> (Select/Table):

Press ENTER to use the current material, enter "S" to access the Material Tables dialog box to select a new one, or "T" to access the Material Catalog dialog box to select a new one.

Section: If "S" for Section is entered in response to the "Angle view (Section/Elevation)" prompt, the command next prompts for an angle length.

> Angle length <8 1/2">:

Enter the length for the entire angle or pick two points demonstrating the length. The command then prompts for the number of rows and gage for each leg of the angle.

> Number of rows in {Size} leg (0/1/2) <1>:

There can be zero (0), one (1), or two (2) rows per leg. If zero rows is chosen, the command prompts for the next leg. If one row is chosen, the command prompts for the gage dimension of the first row.

> Gage 1 dimension <{#}>:

If two rows are chosen, the command also prompts for the gage dimension of the second row.

> Gage 2 dimension <{#}>:

The command then displays the same set of prompts for the second leg.

> Number of rows in {Size} leg (0/1/2) <1>: 2
> Gage 1 dimension <{#}>:
> Gage 2 dimension <{#}>:

The command then prompts for the diameter of the hole at the center of the angle.

> Hole diameter <13/16">:

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The following illustration shows the dimensional parameters for angles drawn in section view.
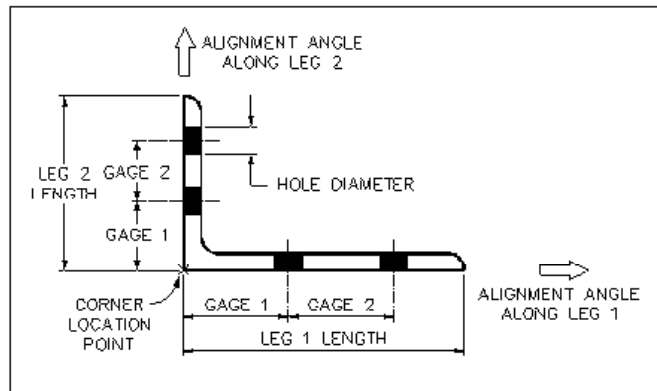
Once the hole diameter has been entered, the command prompts for a location point for the angle.

Corner location point:

Select the point for the outside corner of the angle. The command then displays the following prompts.

Alignment angle along {Size} leg <0>:

Alignment angle along {Size} leg <90>:

Enter angles or select points that demonstrate the angle and orientation of each leg of the angle. The command then inserts the section view of the angle.

Copy objects:

New base point (Copy/Xflip/Yflip/Move/Rotate/Quit) <Quit>

Elevation: If "E" for Elevation is entered in response to the "Angle view (Section/Elevation)" prompt, the following prompt is displayed.

Angle <{Angle material}> (Catalog/Tables/Select/Palette):

Selecting the Catalog option displays the Material Catalog associated with the menu. Selecting the Tables option displays the Material Table associated with the menu. The Select option shows the material dialog box associated with the material. The Palette option displays the current Palette dialog box for material selection. After a material has been selected from any of these methods, the command displays the Connection Angle Parameters dialog box.

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Connection Angle Parameters Dialog Box

The following describes the options within this dialog box.

Construction Settings

The Construction Settings control how the angle is drawn.

Facing Leg: These radio buttons control if the facing leg of the angle is the long or short leg of the angle, for angles with un-equal legs.

Drawn: These radio buttons control how the facing angle is drawn, Normal: with a continuous line, Hidden: with hidden lines, or Hide Leg: where the outstanding leg of the angle is drawn with hidden lines.

Overall Length of Angle: This is a calculated field, taking the worst case of sum of the minimum edge distances top and bottom, and the distance between all the holes for the facing or outstanding leg. When connectors are not added to the angle, this is an editable field.

Construction Point Length: This field is not used in the AEC Tools program.

Construction Point Offset: This field is not used in the AEC Tools program.
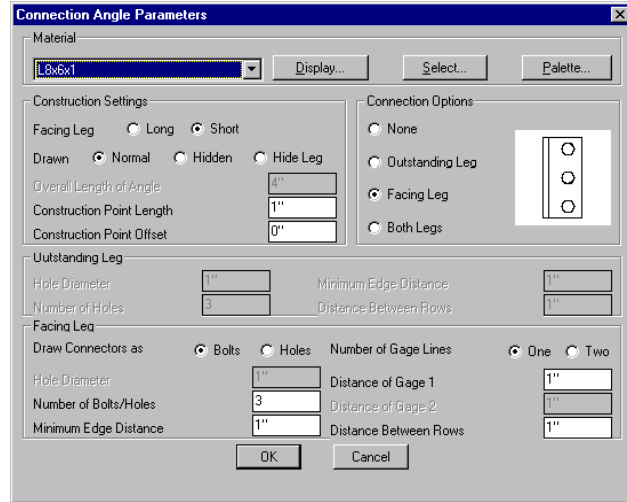
Connection Options

These radio buttons control the type of connection that the angle creates.

None: Bolts or holes are not added to the angle.

Outstanding Leg: Holes are added to the outstanding leg of the angle.

Facing Leg: Bolts or holes are added to the facing leg of the angle.

Both Legs: Bolts or holes are added to the facing leg, and holes are added to the outstanding leg of the angle.

Outstanding Leg

When holes are added to the outstanding leg of the angle in the Connection Options section, these additional parameters are available.

Hole Diameter: Diameter of hole placed in outstanding leg.

Number of Holes: Number of holes drawn in outstanding leg. The holes are drawn as AutoCAD Solids.

Minimum Edge Distance: The distance from the top of the outstanding leg to the center of the first hole.

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Distance Between Rows: Distance between the centers of the rows of bolt holes.

Facing Leg
When connectors are added to the facing leg of the angle, these additional parameters are used.

Draw Connectors as: These radio buttons control whether the connectors drawn on the facing leg are holes or bolts.
Hole Diameter: If the connectors on the facing leg are drawn as holes, this controls the diameter of the holes.
Number of Bolts/Holes: This controls the number of bolts or holes drawn on the facing leg.
Minimum Edge Distance: This is the vertical distance from the edge of the angle to the center of the first and last hole.
Number of Gage Lines: These radio buttons control if the holes are drawn in one vertical row or two.
Distance of Gage 1: This is the distance from the face of the angle to the centers of the first gage line's holes.
Distance of Gage 2: This is the distance from the center of the first gage line's holes to the center of the second gage line's holes.
Distance Between Rows: This is the vertical distance between the centers of a gage line's holes.

After selecting the type of angle to draw in the dialog box and selecting OK, the following prompt displays for the selection of a bolt type, if one has not been selected yet.

> Bolt data not found.
> Enter to select.

Press ENTER to select the bolt data if the Draw connectors as option is set to Bolts. The program is looking for the materials from the Fasteners Material Catalog.

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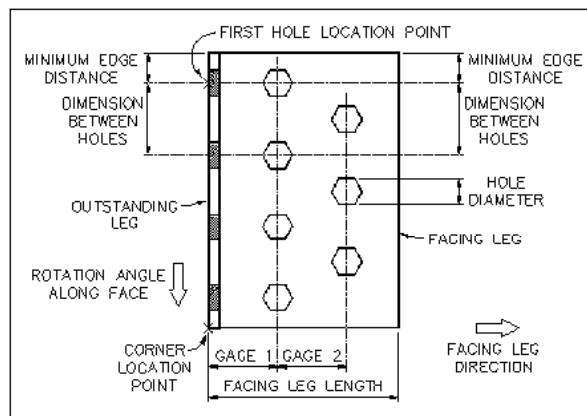
After all of the hole information has been entered, the command prompts for the
location of the first hole point. This point not only locates the first hole, but also the
entire angle in relation to the rest of the detail.

<div align="center">First hole location point:</div>

The command then prompts for information needed to position the angle.

Once the hole sizes and positions for the legs of the angle are defined, the command
prompts for the rotation angle along the face of the angle. This determines how the
angle is oriented in the drawing.

<div align="center">Rotation angle along face <270>:</div>

The command then prompts for the direction of the facing leg in relation to the angle
along the face of the angle.

<div align="center">Direction of facing leg:</div>

After the direction of the facing leg has been entered, the command inserts the
elevational view of the angle.

Holes
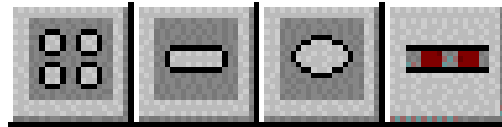
The Holes commands allow you to add
symbols representing holes or slots to the detail. These commands are accessed from
either the Detail Utilities toolbar, or the AEC>Detail Utilities>Holes> cascading menu.
There are four types op holes: round; slots; ellipses; and edge view.

Round Holes: Select the Round Hole command to display the following prompt.

<div align="center">Hole diameter <13/16">:</div>

Enter a diameter or select two points displaying the diameter. The following illustration
shows the parameters used for drawing top views of holes.
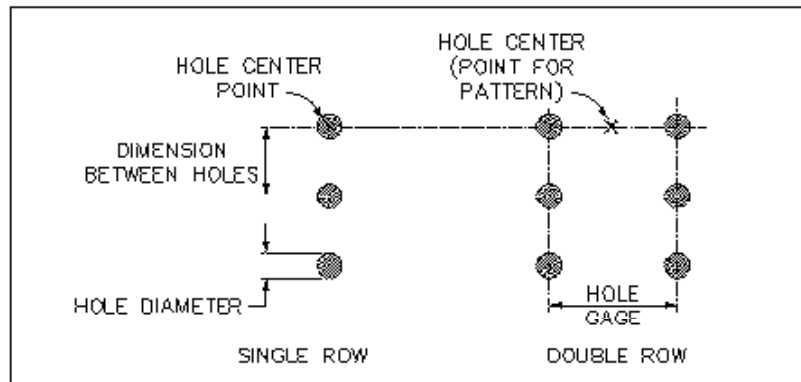
The command then prompts for an insertion point for the hole.

<div align="center">Hole center point (or Pattern):</div>

Either pick a center point for the hole or enter P for Pattern to place an array of holes. If an insertion point is picked, several holes of the same size can be placed by picking several insertion points in response to the following prompt.

> Copy objects:

Press ENTER in response to the Copy objects prompt to exit the command.

If P for Pattern is entered in response to the Hole center point (or Pattern) prompt, the following prompts display.

> Number of holes <{#}>:
> Number of rows <{#}>:
> Hole gage <{#}>:
> Dimension between holes <{#}>:
> Start location point:
> Direction angle along row <{#}>:

Slots: Select the Slot command to prompt for the length and width of the slot.

> Slot width <13/16" {22}>:
> Slot length <1 5/8" {44}>:

Enter a width and length, or select points displaying the length and width. The command then prompts for an insertion point and rotation angle for the slot.

> Slot center point:
> <Rotation angle>/Reference:

The following illustration shows the parameters used to draw top views of slots.
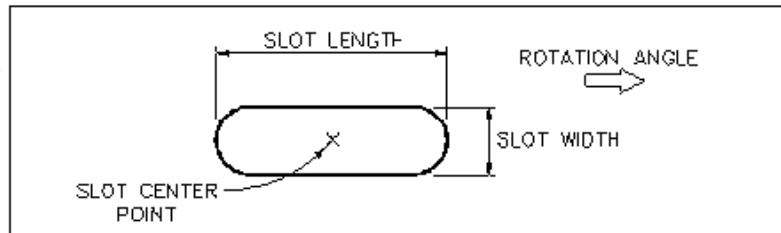
After these values are selected or entered, several slots of the same size can be placed by picking several insertion points in response to the following prompt.

> Copy objects:

Press ENTER in response to the Copy objects prompt to exit the command.

Ellipse: The Ellipse command creates an ellipse on your drawing. The following prompts display.

> Horizontal hole length <8">:
> Vertical hole length <1'-4">
> Ellipse center point:
> Rotation angle<0>:
> Copy objects:

Press ENTER at the Copy objects prompt to exit the command.

Edge View Hole: The Edge View of Hole command draws a symbol representing a hole viewed from an edge. The following prompts display.

> Hole diameter <1/2">:
> First material edge:
> Second material edge (Enter for projection):
> Copy objects:

Press ENTER at the Copy objects prompt to exit the command.

☞ Note: The Material you select must be drawn as a line or polyline

Keynoting

To facilitate entering of annotation and to provide for consistent annotation, AEC Tools includes a standard keynote table file, *keynotes.dbf*. This file contains keywords and keynotes based on the CSI (Construction Specifications Institute) and CSC (Construction Specifications Canada) Masterformat. This file is by no means complete and is intended to be an outline for the creation of a complete project keynote file.

☞ Note: The keynotes file can be replaced with any form of keynoting that associates a key word with a paragraph of text. To create a new keynotes file simply delete the keynotes file, keynotes.dbf, in the projects **zm** folder. When any keynote command is accessed, a new empty file will be automatically created to replace the deleted one.

When you are adding annotations, the AEC Tools program accesses this keynotes file to expand key words into the notes for annotation. For example, bricks have a key word, 04210, and a note, CLAY UNIT MASONRY. With the AEC Tools keynoting system, if you type 04210 in response to the Text prompt of the Keynote Leaders, Notes, or the Softdesk AEC Tools Text Leader command, then the system expands 04210 to read CLAY UNIT MASONRY.

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If you type a double question mark (??) as a separate line of text at the Text prompt, then AEC Tools accesses the Keynotes Manager dialog box and opens the keynote.dbf file to the first record.

If you type a question mark followed by part of a keyword as a separate line of text (for example, ?04), the Keynote Leaders command accesses the Keynotes Manager and sets the current keyword to the first one that matches the part of the keyword entered, in this case, 04100 MORTAR AND MASONRY GROUT. Leader text and notes can be made of multiple references to the keynotes.dbf file by typing additional keys at the Text prompt.

Whenever Details draws a material, it adds information about the material as extended entity data to the graphic object. Using this information, drawings can be annotated automatically by picking the materials to annotate. Details uses one of two sources for determining a material's key word: the description of the material specified in the material's **table file** or the keyword in the material's .**ifm file** (Information file) to link to the *keynotes.dbf* file.

The type of method used for determining the key word is set in the Keynote Leader Setting dialog box. Once a keyword is retrieved for the selected material, the keynotes file is automatically searched for a note associated with it. The action that is performed after the keynotes file has been accessed varies depending on the command that is accessing the file and the project settings made to define its use.
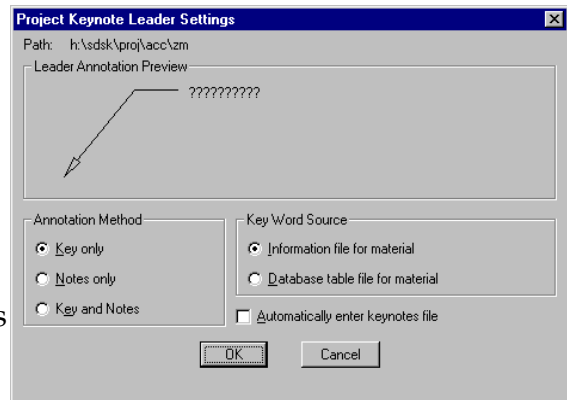
The intelligent annotation commands work on either individual material objects or on completed detail drawings. The completed details can either be AutoCAD blocks or external reference files. To allow flexibility in the way details are annotated from project to project, details can be saved without annotation, inserted into a detail sheet and then annotated using the intelligent annotation commands to meet the project requirements.

There are six commands associated with keynoting. All of them can be found in the AEC>Keynoting cascading menu, while four can be accessed in the Keynotes toolbar.

Keynote Leaders: The Keynote Leaders command annotates existing materials with text leaders. Depending on the settings in the Keynotes Leader Settings from the Utility menu, this command can work in a number of different ways.

When selected, the command first prompts for the material to annotate.

Pick a material object:

Select any material to annotate. The command displays the selected material and prompts for a new arrow point.

Material: {Material}
New object point (Skip/Default) <Default>:

Select a new point for the arrow point of the leader, or press ENTER to use the point at which the material was selected. This is the starting point for the leader at which the arrow head is drawn. The skip option allows the selected material to be skipped and a new material selected without canceling the command.

☞ Note: The settings from the AEC menu determine the style of the leader including the shape and size of the arrow head. A cascading Annotation menu is available from the Draw menu.

After you have defined a starting point for the leader, the command prompts for additional leader points.

To point:

The command continues to prompt for additional points. When all leader points have been selected, press ENTER at this prompt. The command draws the leader.

At this point, the prompt structure and results are different depending on the setting in the Keynote Leader Setting dialog box. The text applied to the leader will be composed from the key word and note from the project *keynotes.dbf* file. The leader can be defined to display the key word only, the note only, or both the key and note.

The Keynotes Manager dialog box automatically displays at the end of the leader creation. By default, the entry that is closest to the key word is highlighted. You can use either the default, you can select a new keynote, or you can create a new keynote. Whichever keynote is highlighted will be applied to the leader after you click OK. If you did not select automatic entry, the text values found in the keynotes file will automatically be applied to the leader.

If you set notes only for annotation and no note is found for the key word, the key word is used and the text prompt is displayed to add more text. Using a keynote text option which includes the key note in the annotation leader text will display the following prompt at the end of the note placement:

Text:

At this point the AutoCAD DTEXT command is active and positioned for a new line of text directly below the note. This provides the ability to add additional lines of text to the note text already applied to the leader. The keynotes file can be accessed to add more text. If you type a double question mark (??) as a separate line of text, the Keynotes Manager dialog box displays to select a note. If you type a question mark followed by part of a keyword as a separate line of text (for example, ?06), the Keynotes Manager dialog box displays and sets the current keyword to the first one that matches the part of the keyword that you typed. If an exact match is found in the keynotes file, the note is automatically applied without displaying the dialog box. This process can provide complex notes that combine notes from the kenotes.dbf file and regular text that is typed at the command line.

Note: To have individual material description from the material table used as keynote leader annotation make the following setting in the Keynote Leader Setting dialog box. Set the annotation method to key only, keyword source to the database table, and turn automatic keynote file entry off.

ConDoc Annotation
To support ConDoc style annotation make the following setting in the Keynote Leader Setting dialog box. Set the annotation method to **key only**, keyword source to the **information file**, and **select automatic keynote file entry**. These settings provide medium scope annotation capability. The keynotes file is set to be automatically entered to create narrow scope annotation by creating a new key word and note for the specific material being annotated.

☞ Note: Narrow scope annotation can be automated by adding a narrow scope key word column to the material table and provide a unique narrow scope key moored to each row in the table. Identifying the column as the key word column in the material tables information file. Set the key word source in the Keynote Leader Setting dialog box to the database table file.

Use the **Chart Keynotes** command to produce the chart that is referenced by the key word annotation leaders.

☞ Note: The AEC Tools program provides a link between the Text Leader command in the Tools menu and the keynoting system. If you type a double question mark (??) as a separate line of text, the Text Leader command accesses the Keynotes Manager dialog box. If you type a question mark, followed by part of a keyword as a separate line of text (for example, ?06), the Text Leader command accesses the Keynotes Manager and sets the current keyword to the first one that matches the part of the keyword entered.

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Notes: The Notes command adds notes to any detail drawing, using a combination of the standard AutoCAD DTEXT command, and a link to the Details program's keynotes file system. When selected, this command first prompts for the justification.

        Note justification...
        Left/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR <Left>:

The command then prompts for the starting point and rotation angle for the text.

        Start point:
        Rotation angle <0>:

The actual location within the text of the start point depends on the justification style selected. The command then prompts for the note text.

        Text:

This section of the Notes command is identical to the AutoCAD DTEXT command.

The Details program provides a link between the Notes command and the keynoting system. If you type a double question mark (??) as a separate line of text, the Notes command accesses the Keynotes Manager dialog box. If a question mark, followed by part of a keyword, is typed as a separate line of text (for example, ?06), the Notes command accesses the Keynotes Manager and sets the current keyword to the first one that matches the part of the keyword entered.

Chart Keynotes: The Chart Keynotes command creates a chart of keyword and keynotes for selected materials. The command first prompts for the selection of the materials to include in the chart.

        Select objects:

Press ENTER in response to this prompt after all materials have been selected. The command then goes through the selection set and finds material keywords and keynote descriptions. These keywords and keynote descriptions are based on the keynotes file *keynotes.dbf* located in the either the system or project folder.

        Number of objects searched: {#}

After all information has been retrieved, the command prompts for the starting point for the chart.

        Chart start point:

The Chart Keynotes command then puts the chart in DRAG mode and allows the selection of a new chart start point.

        Reposition chart:

Either select a new start point or press ENTER to leave the chart in the original location.

Specification File: The Specification File command creates two external ASCII text files for the materials used on a per drawing per project basis. The Specification File command first prompts for the selection of the materials to use in the specification file.

 This selection can include both individual material components as well as completed details on a detail sheet. These details can either be standard inserted blocks or external references (Xref).

Select objects:

Press ENTER in response to this prompt after all materials have been selected. The command then goes through the selection set and extracts the key word and note for each material based on the keynotes file, *keynotes.dbf*, located in the project **zm** folder.

Number of objects searched: {#}

After all information has been retrieved, the command creates a drawing specific file in projects **zm/spec** folder. This file uses the current drawing name with *.spx* extension. The program then uses all of the specification files in that directory to produce two project reports in the project **zm** folder. The *keynotes.txt* file contains information identical to that produced by the Chart Keynotes command, except it includes data from all drawings in the project that have had the specification file command run on them. This provides a project level report on all material key words and notes. The second file, *spec_ref.txt*, is produced in the same way but contains a project level report of all key words and their associated specification file and section reference. As the project reports are created the following prompts are displayed showing the number of lines of information written to each file.

Wrote {#} lines to file: {keynote.txt File}

Wrote {#} lines to file: {spec_ref.txt File}

These specification files can be viewed and edited using any text editor or word processor.

Edit Keynotes File: The Edit Keynotes File... command is used to edit the keynote file. This file is called keynotes.dbf and is stored in the project **zm** folder. When selected, the Edit Keynotes File... command displays the Keynotes Manager dialog box.

This dialog box displays the key words in a scrolling list to the left of the screen and the descriptions in the right section. The specification reference area below the note shows the specification file name and section reference. All of the areas and options available from this dialog box are described in the following sections.
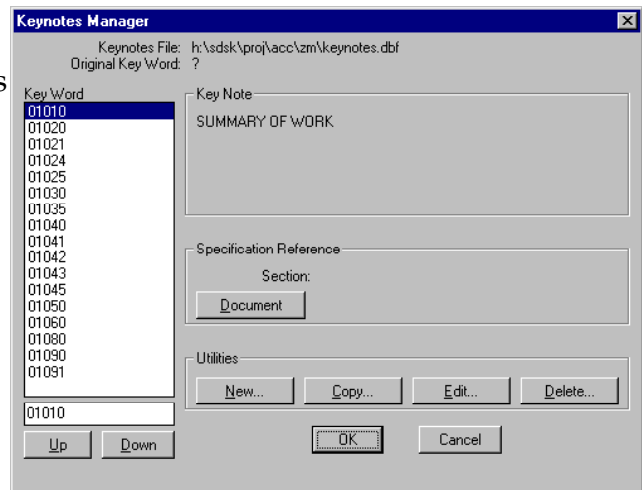
Keynotes File: This area shows the directory path and keynotes file being displayed. This field cannot be edited in this dialog box.

Original Key Word: The original keyword is the keyword that was used to access the keynotes file. A single question mark (?) means no keyword was supplied and the first key word is highlighted.

Key Word: This scrolling list presents all of the key words in the keynotes.dbf file. The list displayed is controlled by the Up and Down button described below. These are the codes corresponding with the keynotes used by annotate leaders, notes, charts, and external reports. A key word can be any character string but is limited to 12 characters and must be unique within the file. The edit box at the bottom of the list is provided to quickly highlight a key word and note. A complete or partial key word can be entered, if an exact match is not found the closest match is highlighted.

Up: Replaces the current list of keywords with the keywords above the currently highlighted key word.

Down: Replaces the current list of keywords with the keywords below the currently highlighted key word.

Key Note: This is the description for the keynoted material corresponding with the highlighted key word. This description is used when keynote leaders, notes, charts, and external reports are used. Key notes consist of one to six lines of text with a limit of 50 characters per line.

Section: This is an optional field which can be used to identify a section within the specification document shown in the document field. This information is output to a project file named spec_ref.txt when the Specification File command is run. The section text can be up to 20 characters long.

Document: This is an optional field which can be used to identify a specification document that relates to the currently highlighted key word. This information is output to a project file named spec_ref.txt when the Specification File command is run. The document file name can be up to 50 characters long.

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New...: This b
utton accesses a dialog box used to enter new keynotes.
All options available in this dialog box are described in
the following section.
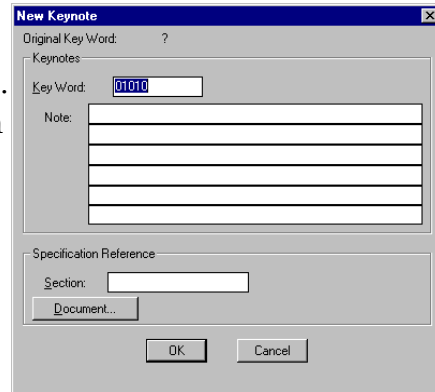
> **Original Key Word**: This is the keyword that
> was used to enter the keynotes table.
>
> **Key Word**: Enter the new key word in this edit
> box. The key word can be up to 12 characters in
> length and can include any alphanumeric characters with one exception. Do not
> type a double question mark (??) as part of any key word. This character set is
> used as a wild card when entering keynote information from other programs.
>
> **Description:** Enter the description for the keynote in this set of edit boxes. The
> description can be up to 6 lines in length and can include any alphanumeric
> character.
>
> **Section:** Enter a reference to a particular section within the specification file
> specified in the document field. This reference can be any character string up to
> 20 characters long. Typically, this field would identify a portion of the
> specification document by a section, title, or page number. This entry is optional.
>
> **Document:** This button displays the AutoCAD File Selection dialog box used to
> identify the name of the specification document to be associated with the key
> word. This entry is optional.

Copy: This button accesses a dialog box used to copy existing keynotes to new ones.
The areas in the Copy Keynote dialog box are identical to those used in the New
Keynote dialog box. See the previous New button description.

Edit: This button accesses a dialog box used to edit existing keynotes. The areas in the
Edit Keynote dialog box are identical to those used in the New Keynote dialog box. See
the previous New button description .

Delete: This button deletes the currently highlighted
key word and accompanying keynote. When
selected, this button first displays a confirmation
dialog box before removing the key word and key
note.
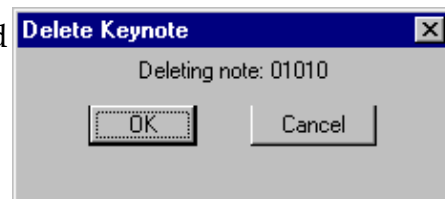
The keynotes.dbf file is an industry standard dBaseIV file that can be edited with external database programs. The column format of the database file is critical to maintain, and must not be changed.

AEC Tools and Details uses the *keynotes.dbf* table file in the project directory for intelligent annotation, keynote charts, and for specification file generation. The file has the format shown in the following table:

Column Name	Data Type	Width	Decimals
KEYWORD	CHARACTER	12	0
KEY_NOTE1	CHARACTER	50	0
KEY_NOTE2	CHARACTER	50	0
KEY_NOTE3	CHARACTER	50	0
KEY_NOTE4	CHARACTER	50	0
KEY_NOTE5	CHARACTER	50	0
KEY_NOTE6	CHARACTER	50	0
SPEC_SECT	CHARACTER	20	0
SPEC_TEXT	CHARACTER	50	0

KEYWORD: The Keyword column contains a text string that is used as the index for the keynotes.dbf file.

KEY_NOTE1-6: The Key_Note columns contains the text strings that are returned Details for use in annotation.

SPEC_SECT: The Spec_Sect column contains a text string that is used to store a reference to a section of the specification text file associated with the keyword.

SPEC_TEXT: The Spec_Text column contains a path to a specification text file associated with the keyword

☞ Note: Two specification columns have been added to the keynotes.dbf file for this release. Previous versions of this file are automatically converted to the new structure.

Detail Management

Creating details is only part of the benefit of the Details program in AEC Tools. Using details over and over again, or combining different scaled details **without** having to use *paperspace* is another very important aspect of this software.

The Detail Manager commands can be accessed from the AEC>Detail Manager cascading menu, or the Detail Manager toolbar.
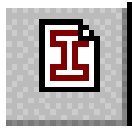
Setup New Detail: The Setup New Detail... command prepares the current drawing for detail creation. This command displays a dialog box containing standard drawing scales as well as an edit box for entering non-standard drawing scales. Select the detail scale factor from the scrolling list in the dialog box or enter a scale factor in the Scale edit box. Click OK to exit this dialog box.
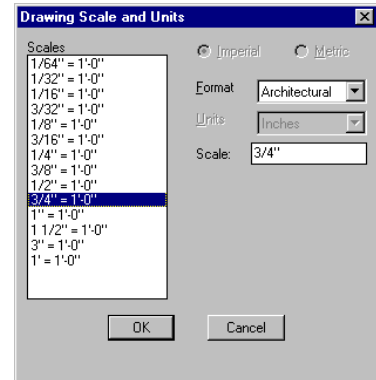
☞ Note: A detail *with* annotation or hatching is drawn for the **final plot scale** you specify at this time. It will not display correctly if you try to plot it at a different scale in the future

After the scale has been selected, the command displays the following informational prompt.

> Draw detail.

Use any command from either AutoCAD, the AEC Tools program, or any other Softdesk program to create the detail. When the detail is finished, use the Create Detail command to save it to a library and insert it at the proper scale in the current drawing. If you do not want to save the detail or if you want to start the detail over with new scale selected, use the Reset command to reset the drawing scales and use the AutoCAD ERASE or UNDO command to remove any unwanted entities from the drawing.

☞ Note: The drawing scale is set for detail creation until the Create Detail or Reset Drawing command is used, which will set the drawing back to the original scale defined when it was created.

Setup Existing Detail: he Setup Existing Detail command puts the current drawing in detail mode by specifying an existing detail block or xref to edit. The Setup Existing Detail command displays the following prompt.

> Select detail block:

The detail scale is calculated from the selected detail automatically. The drawing is placed in detail mode with the detail scale equal to the block scale. The command then prompts for an insertion point for this block. This point should be located where the detail is to be edited.

> Insertion point:

The command then determines the drawing scale based on the selected block. Both the detail name and scale are displayed at the command prompt line.

> Edit existing detail named {Detail Name}.
> Drawing scale: {Scale}

The block is resized to true size and inserted as separate entities at the selected point for editing. Use any command from either AutoCAD or the AEC Tools menu to edit the detail. When the detail is finished, use the Create command to save it to a library and reinsert it at the proper scale in the current drawing.

☞ Note: There is no *automated* process to resave the detail

To abort the creation of a detail, use the Reset command to reset the drawing scales and use the AutoCAD ERASE or UNDO command to remove any unwanted entities from the drawing.

☞ Note: The drawing scale is set for detail creation until the Create or Reset command is used, which will set the drawing back to the original scale defined when it was created.
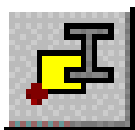
Create Detail: The Create Detail command is used to turn a completed detail into a detail block at the established detail scale. The detail first requires an insertion base point.

> Insertion base point:

Once the insertion point has been specified, the entities that make up the detail must be selected. In most cases a prompt will be displayed to allow automatic selection of all objects drawn since the detail creation was initiated. Standard selection is always available whether automatic selection is used or not. The command displays the following prompts.

> Automatically select objects for detail <Yes>:
> Select objects:

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Use any of the AutoCAD or Softdesk selection methods to select the detail entities. At this point the original drawing scale factors are restored and a prompt is displayed to allow the detail to be saved to a library.

> Original drawing factors restored.
> Drawing scale: {Scale}
> Save detail to library <Yes>:

Responding yes at this prompt displays the Save Detail dialog box explained below. A no response displays the following prompts to place the detail in the current drawing.

> Block name:
> Insertion point:
> Add detail title <Yes>:

If the block name provided already exists in the drawing or is externally referenced, a prompt will display a warning and ask if it should be redefined.

Saving a Detail to a Library

Saving a detail to a library is done through the Save Detail dialog box. The top portion displays information about the detail which is about to be saved. The first field shows the current library in which the detail will be saved. The library can be change through the library at the bottom of the dialog box.
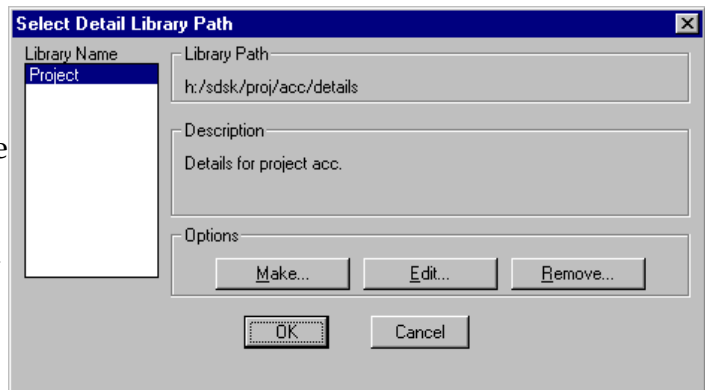
The remaining fields in the detail information box display the drawing scale and its insertion factor and the date the detail was created.

The edit boxes provide input to define the name of the drawing and slide to be created in the library. If the detail drawing and slides already exists in the library, the dialog box will present an option to overwrite the existing detail with the current one being saved.

The Title edit box is used to help identify the detail at insertion time and also is used for the title of the detail in the drawing. The descriptive fields provide the ability to further describe the detail, the descriptive text is only used to help identify the detail at insertion time.

Once the fields are filled in and the OK is selected, the detail selection set is wblocked to the library directory along with a slide of everything currently visible on the screen.

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☞ Note: The detail's reference slide is **automatically created based on what is shown on the screen before the Save Dialog box is entered**. Use the AutoCAD ZOOM commands to make sure the screen shows the detail the way you want it displayed in the Insert Detail dialog box prior to initiating the create sequence.

Once the detail has been saved to a library, the command line presents a prompt to allow optional insertion of the detail as either a regular block or an external reference.

> Insert detail <Yes>:
> Insert as (Xref/Block) <Block>:
> Insertion point:
> Rotation angle <0>:
> Add detail title <Yes>:

If a detail is blocked and inserted into the drawing, either directly or after saved to a library, you are prompted for the addition of a title. The type of tile applied to the detail depends on the settings made in the Border and Title Settings dialog box.

Detail Libraries

Detail Libraries allow you to group and display libraries in a logical way for use in your construction documents. The Select Detail Library Path dialog box is first displayed when saving a new detail. Double click on the Library Name from the scrolling list on the left to select the library. The following dialog box appears.
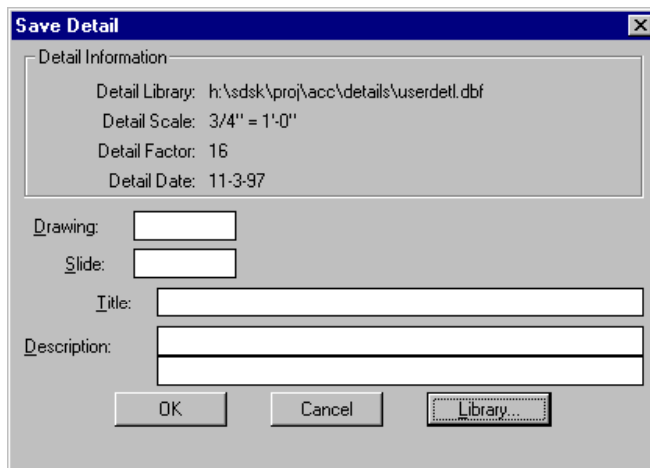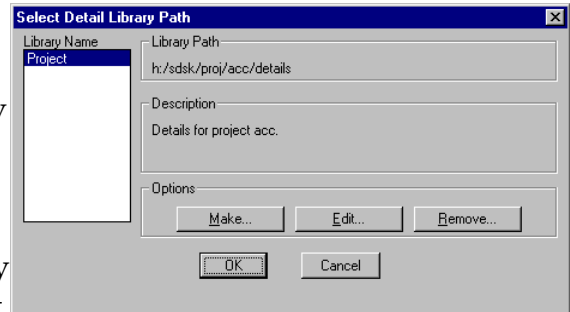
Select a drawing name and slide.

☞ Note: Keep the drawing name to 8 letters, and use the same name for both slide and drawing file.

The Title description line is used by the **Create Title** command. The description field is optional.

The following describes the Options section in the Select Detail Library Path dialog box.

Make...
**:** This button accesses the Make Detail Path dialog box. Specify a new Name for the library. Each Library is contained in a separate database file. The default for the project is the *Userdetl.dbf* **Details** folder of the current Softdesk project. This dialog box allows you to create new folders off of the current Project Root, or off of the Library Root folder.
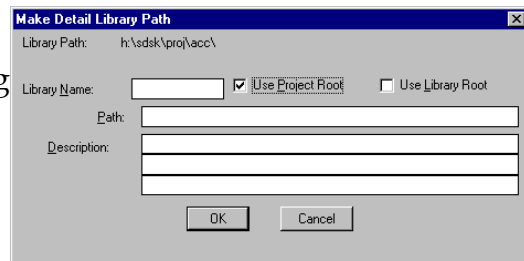
☞ Note: To specify a different drive, edit the DSLIB=(H:\,Details Library Root) line in the *sdsk.dfm* file

Edit: This button accesses the Edit Detail Path dialog box . It is similar to the Make Detail Path dialog box. Edit this as required to accommodate changes to where you want to store your details.
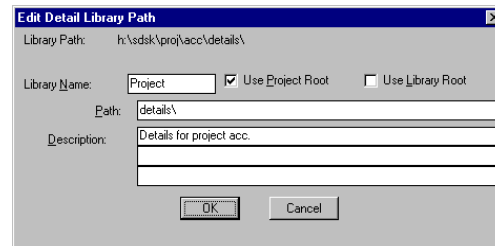
Remove: Use this button to delete a Detail Library.
This does not erase the *userdetl.dbf* file, or related drawing and slide files. It merely removes them from the Library Name list.
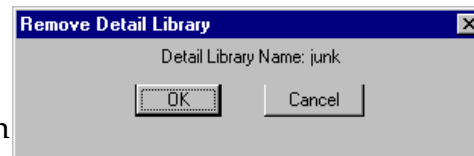
Create Titles: The Create Titles command is used to draw a detail title around details. In addition to accessing this command from the menu, any detail insertion command will also provide this feature automatically. A border may also be drawn with the title depending on the settings made in the Title and Border Setting dialog box.

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The prompt sequence is identical for border type setting beginning with the following prompt.

Detail identification:

Detail identification is used to identify the detail when referenced from section and detail marks. If the border grid is set for use, the detail identification is automatically determined provided the drawing supports border grids. The point used to determine the identification value differs depending on the type of title and border being used. If only the title is set, the center of the title bubble is used to determine the identification value. If a border is included in the title, the lower left hand corner of the border is used to determine the identification value.

Detail title:

The detail title is used to describe the detail. If the title is being created as a continuing process of a detail insertion, the title text is automatically supplied from the detail title assigned to it in the library. No prompt is displayed when title is created as a part of detail insertion.

Detail scale: {Scale}

The detail scale uses the current drawing scale by default unless the title is being created as a continuing process of detail insertion. If detail insertion is active, the detail scale is retrieved from the library and no prompt is displayed.

Detail reference:

Detail reference is designed for listing the drawings which have section or detail marks referencing the detail. This is typically used as a form of cross referencing.

If the detail title includes a border the following prompts are displayed to define the outer boundary points of the border, including the title block area.

First corner point:
Second corner point:

When the border is completed, margin lines will be generated if they were enabled in the Title and Border Setting dialog. Margin lines are intended for reference only therefore are drawn on the NOPLOT layer.

Reset Drawing: The Reset Drawing command resets and restores the original drawing scale and settings. This takes the drawing out of DETAIL mode. Informational prompts reporting the drawing scale are displayed.

Use the Reset Drawing command to exit the creation or editing of a detail. This returns the drawing to the original scale but it does not change any of the entities created or edited with the Setup New Detail... or Setup Existing Detail commands.
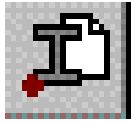
Insert Detail: The Insert Detail... command inserts details selected from a detail library. When this command is selected, it displays the AEC Tools Library File dialog box. This dialog box displays the details found in the current library one at a time. The dialog box title displays the drive, path and filename of the current library. All parts of this dialog box are described in the following section.

Drawing: This scrolling list displays the names of the details saved in the current detail library. Select the name of the detail to insert from this list.

Image: This image tile shows the slide saved (if any) when the detail was created. If a slide was not captured, this image tile is left blank. The field below this image tile displays the title saved with the detail. This title is automatically applied to the detail when it is inserted and add detail title is accepted as part of the insertion process.

Edit: This button displays the Save Detail dialog box, to edit the Slide, Title, or Description information.

Delete: This button displays the Delete Detail Warning dialog box before removing a detail from the library. Click OK to delete the detail or Cancel not to delete it.

Comments: This area displays any comments that were made about the detail. This information cannot be edited.

Information: This area displays information about the highlighted detail. This information shows the scale and date of creation. This information cannot be edited.

Library: This button is used to change the current detail library. When selected, this button accesses the Select Detail Library Path dialog box. This dialog box lists all defined detail libraries and their paths and descriptions. Select the library of details to display from the Library Name scrolling list. If a library other than the current one is selected, that library remains current until a new one is selected.

☞ Note: A library must be established before any detail management function is performed. This dialog box may be displayed as the first action to do until a library has been selected.

Once the detail to be inserted has been highlighted, Click OK to place the detail. The command then prompts for how the detail is to be inserted.

Insert as (Xref/Block) <Block>:

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If X for Xref is entered as the response to this prompt, the command inserts the detail as an external reference. Press ENTER to accept the default of Block to insert the detail as a block. Regardless of how the detail is inserted, the command prompts for an insertion point for the detail.

Insertion point:

Use the pointing device to drag the detail into place or enter coordinates for the insertion point.

☞ Note: If the block was created in the same drawing, this command will not allow you to insert the detail as an XREF.

The prompting continues with the block rotation angle an option to add a detail title.

Rotation angle <0>:

Add detail title <Yes>:

Save Detail: The Save Detail command can be used to save any objects in the current drawing as a detail at the current drawing scale. This command is similar to the Create Detail command, except it simply WBLOCKs the selected objects to the library. No insertion commands follow the save. This command can be used to create detail for several different libraries or to add details taken from other sources to detail libraries for use with the AEC T

ools program.
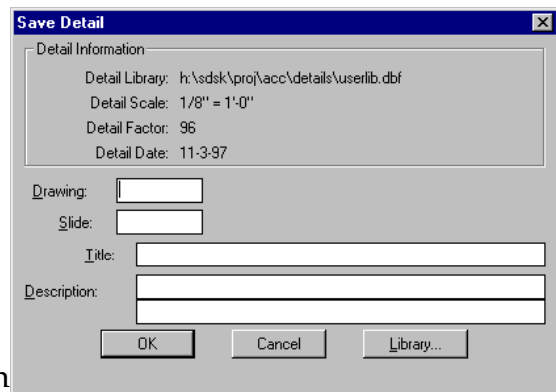
When selected, the Save Detail command prompts for the selection of the objects to save.

Select objects:

Insertion base point:

The command then displays the Save Detail dialog box.

About the Author

Mr. Addis first began working with CAD in 1987 as the office manager of a small architectural practice. In January of 1988, Reid became the CAD system coordinator of Bower Lewis Thrower Architects in Philadelphia. A year later, he formed his own CAD consulting firm, Addis Computer Consultants, which he currently manages while serving as the Architectural Application Specialist at Granary Associates in Philadelphia.

Reid is a Certified Technical Trainer (CTT), and an AutoCAD Certified Instructor (ACI). Beginning in 1993, Reid became a *Softdesk Registered Consultant* (SRC). With Autodesk's acquisition of Softdesk in 1997, the SRC program evolved into the *Autodesk Authorized Independent AEC Consultant program*. Reid has always felt that a consultant's three greatest assets are honesty, integrity, and independence. In order to provide a wider variety of services to his clients, Reid began to support KETIV's ArchT software in 1997, and was a *Certified ArchT Trainer*. Reid also works with IntelliCAD, and 3D Studio Viz. Reid provides training for *AutoCAD's Architectural Desktop,* AutoCAD, AutoCAD LT, 3D Studio Viz, and IntelliCAD. Reid is also co-author for the book "Softdesk Architecture 1 Certified Courseware" from Onword Press. He has worked with AutoCAD since 1987, beginning with Release 2.6, and provides production assistance to many architectural firms.

Reid also provides consulting services for the MicroStation community. He has worked with MicroStation since Release 3.3 and offers a variety of training courses for MicroStation SE/J. He has co-authored two books on MicroStation including "Inside MicroStation 5.0" from New Riders Press, and "The MicroStation Bible" from Onward Press. Before Ideagraphix was purchased by Bentley Systems, Inc, Reid worked with their MicroArchitect and speedikon M products

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