
Cats 8

AutoCAD / MicroStation

Translator

User Guide



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Introduction

CATS gives you full-featured translation between AutoCAD and MicroStation. CATS gives you the ability to convert 2D or 3D files to and from MicroStation DGN format and your choice of AutoCAD DWG or DXF formats. CATS supports AutoCAD releases 14 through 2009 and MicroStation releases V7 (3, 4, 5, 95, SE, J) and V8.

Features

CATS redefines CAD translation technology. Using our rule-based syntax, you can customize your translations to meet strict project requirements.

Translation options are defined based on the characteristics of each entity as it is translated. Options can be selected based on cell or block name, color, class, entity type, font, graphic group, text justification, level, layer, linetype, and weight — singly or in combination.

All numeric selections can use lists and/or ranges, i.e., (1,4,6-20).

All string selections can use lists and/or DOS wildcards, i.e., (bdr,s01,hvac*,z?c).

CATS also includes extensive options for handling:

- Entities
- Text
- Blocks
- Cells
- Reference files
- XRefs
- Layers
- AutoCAD header variables

Network License

CATS is now provided with a network license. That means you can use it on any workstation on your network by one user at a time.

To set up additional workstations to use CATS, simply create a shortcut to the program file CATS8.EXE on the computer where CATS was installed. That's all there is to it! All network users must have write access to the Cats 8 directory.

Unless you have purchased additional seats, the installation is configured for 1 simultaneous user. Additional user licenses can be purchased for \$1200 each. Discounts are available for adding larger numbers of users. Just give our ales Dept a call for a custom quotation.

Using Cats



Start CATS by double clicking the CATS linked hands icon.

CATS uses a project as the basis for all translations. The project defines all of the specifics of the translation job, the names of files to be translated, the type of input, the type of output, and any parameters used to customize the translation.

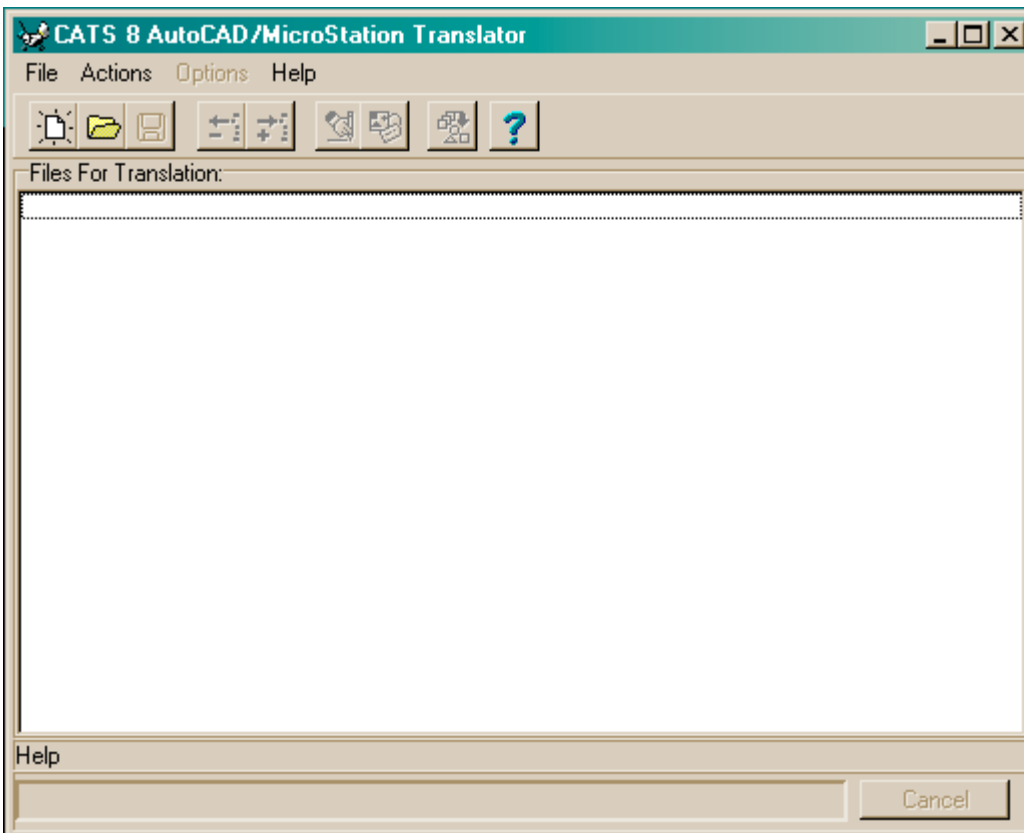
CATS translations can be customized using **Control Files**. Control files contain commands and rules for setting levels / layers, colors, linetypes, font / text styles, weights / widths, and other parameters. See Customizing CATS for details in setting up and using control files.

To create a new project and translate files -

1. Pick the File | New Project menu command. The Options dialog is displayed.
2. Pick the Input Tab .
3. Select the type of file you are translating.
4. Pick the Output Tab .
5. Select the type of file you wish to create.
6. Click the OK button to close the Options dialog.
7. Pick the Action | Add File menu command.
8. Select the files to translate from the Select Files to Add dialog. You can select one or more files.
9. Click the Open button to accept your selections. The files name will be displayed in the Files For Translation list.
10. Pick the File | Save Project menu command to save the project.
11. To translate your files, select the files in the Files For Translation list with the mouse. You can select one or more files.
12. Pick the Actions | Translate menu command to start the translation.
13. Check the Translation Status at the bottom of the CATS Project Window until the translation is finished.

The CATS Project Window

Start CATS by clicking the CATS linked hands icon. You will see the following CATS project window:



The window defines a translation project. A project consists of a list of files to translate and the associated options defining the input, output, and project parameters for the translation.

Commands can be accessed by the toolbars, menus, or keyboard shortcuts:

- **toolbars** - clicking on a toolbar button selects the command. Holding the mouse pointer over a toolbar command will display the name of the command.
- **menus** - pull-down menus give access to all commands.
- **keyboard shortcuts** - the <Alt> key can be used with the underlined letter on the menus to reach the desired menu command.

File Menu

The **File Menu** commands are used to control CATS project files. CATS uses projects to define the input, output, parameters, and files for translation. The File Menu also includes recently used projects. Double click on a recent project name to open it.

The File Menu commands are:

- New Project - create a new CATS project.
- Open Project - open an existing project file.
- Save Project - save changes to a project file.
- Save Project As - save the current project with a new name.
- Exit - exit the CATS program.

File Menu Commands

Accessing the **File Menu** displays the following command list:

New Project	Ctrl+N
Open Project...	Ctrl+O
Save Project	Ctrl+S
Save Project As...	
<hr/>	
License Manager...	Ctrl+L
<hr/>	
Exit	

The following toolbar buttons also provide access to the same file commands:



New Project



The **New Project** command creates a new translation project.

Open Project



The **Open Project** command opens an existing translation project. When a project is opened, the name of the project appears in the window title bar at the top of the CATS window. This is the active project.

Save Project



The **Save Project** command saves the active project parameters to the currently active project. If a new project, which has not been saved before, is active, you will be asked to give a project file name.

Save Project As

The **Save Project As** command saves the active project with a new project name. You will be asked to give the new project name. This new project then becomes the active project and its name is shown in the CATS window title bar.

Exit

The **Exit** command will end the CATS program. If you have made changes to the active project, you will be given a chance to save these changes before the program exits.

Actions Menu

Actions Menu commands are used to control the names in the Files for Translation List and Translate files in CATS project.

The Actions Menu commands are:

- Add File - add files to the Files for Translation List
- Remove File - removes selected files from the Files for Translation List.
- Create Combined Control File - creates a preliminary control file for a group of files.
- Create Separate Control Files - creates a set of individual preliminary control files for each file in a group of files.
- Select All - select all of the files in the Files for Translation List.
- Deselect All - deselect all of the select files in the File for Translation List.
- Translate - translates all of the selected files in the Files for Translation List.

Actions Menu Commands

Accessing the **Actions** menu displays the following command list:

Actions	Options	Help
A <u>dd</u> File		Ctrl+I
R <u>emove</u> File		Ctrl+R
Cr <u>eate</u> C <u>ombined</u> Control File		
Cr <u>eate</u> S <u>eparate</u> Control Files		
S <u>elect</u> All		Ctrl+A
D <u>eselect</u> All		Ctrl+D
T <u>ranslate</u>		Ctrl+T

The following toolbar buttons also provide access to the same file commands:



Add File



The **Add File** command adds file names to the Files For Translation list. A file dialog menu is opened to select one or more files.

Remove File



The **Remove File** command removes the selected file names from the Files For Translation list. File names in the list are selected with the **Select All** command or with the mouse by clicking on the file name. Multiple files can be selected with the <Ctrl> and <Shift> keys.

Create Combined Control File



The **Create Combined Control File** command creates a single working control file using the selected file names in the Files For Translation list. The control file will contain default rules for the layers, levels, colors, linetypes, weights, text styles, fonts actually found in the selected files.

Create Separate Control Files

The **Create Separate Control Files** command creates a separate control file for each file selected in the Files For Translation list. The control files are named *filename.con*. The control files will be created in the same directory as the selected file or the output directory, if specified. The command also creates a control list file for the selected files. The user is prompted for the name of the control list file being created.

Select All

The **Select All** command selects all of the file names in the Files For Translation list.

Deselect All

The **Deselect All** command deselects all of the files names in the Files For Translation list.

Translate



The **Translate** command translates the selected files in the Files For Translation list.

Options Menu

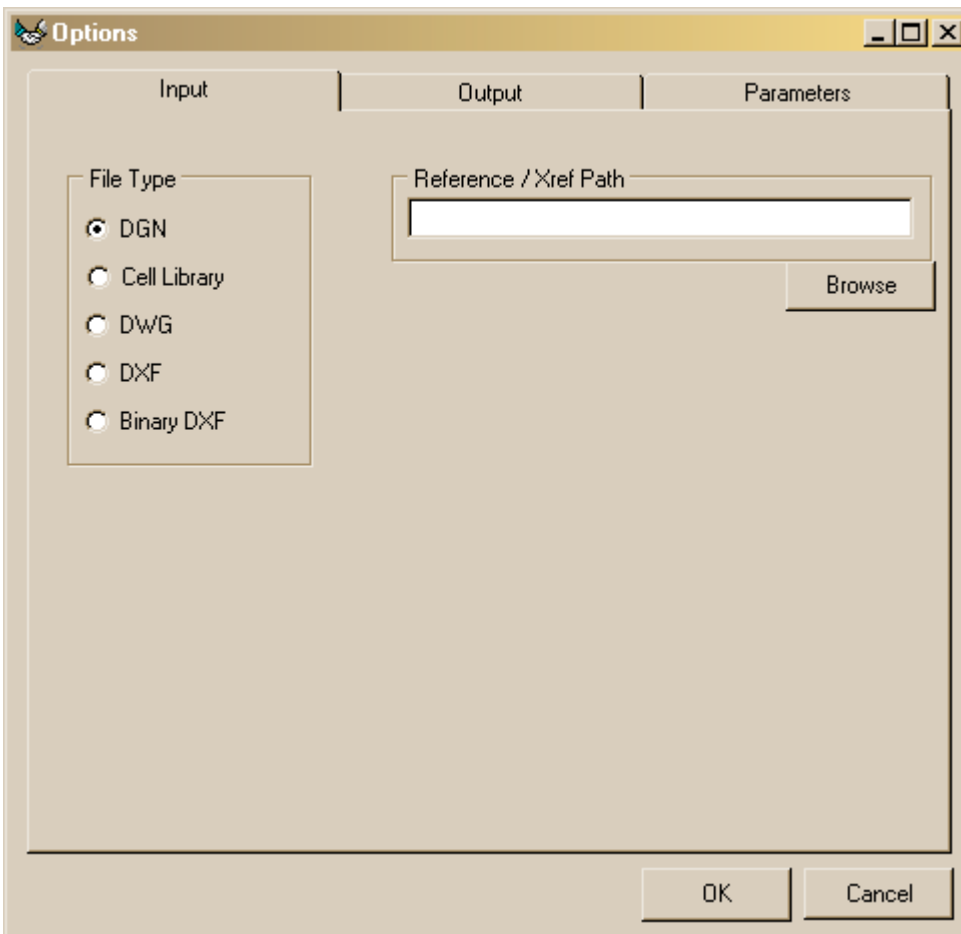


The **Options** menu command or Options command button will open the **Options** window. The **Options** window is used to define the project options and parameters. The **Options** window is automatically displayed when a new project is created.

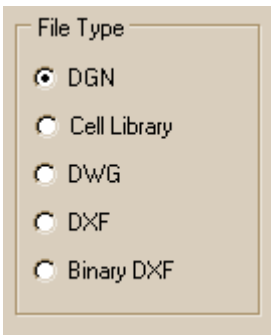
The **Options** window is divided into three tabs: Input , Output , and Parameters.

Input Options

The **Input** tab defines the type and characteristics of the input files, i.e., the file(s) to be translated.

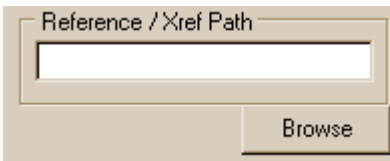


File Type



The **File Type** selection list gives the kind of file to be translated.

Reference / Xref Path



The **Reference / Xref Path** prompt is used to specify a list of directories to search for locating reference files or Xrefs attached to the DGN or DWG file(s) being translated. The directories are entered separated by semicolons, e.g.,

```
c:\temp\lfdgn\sub\refs\;h:\mydir\
```

The Browse button can be selected multiple times to add additional paths.

CATS searches for reference files in the following order:

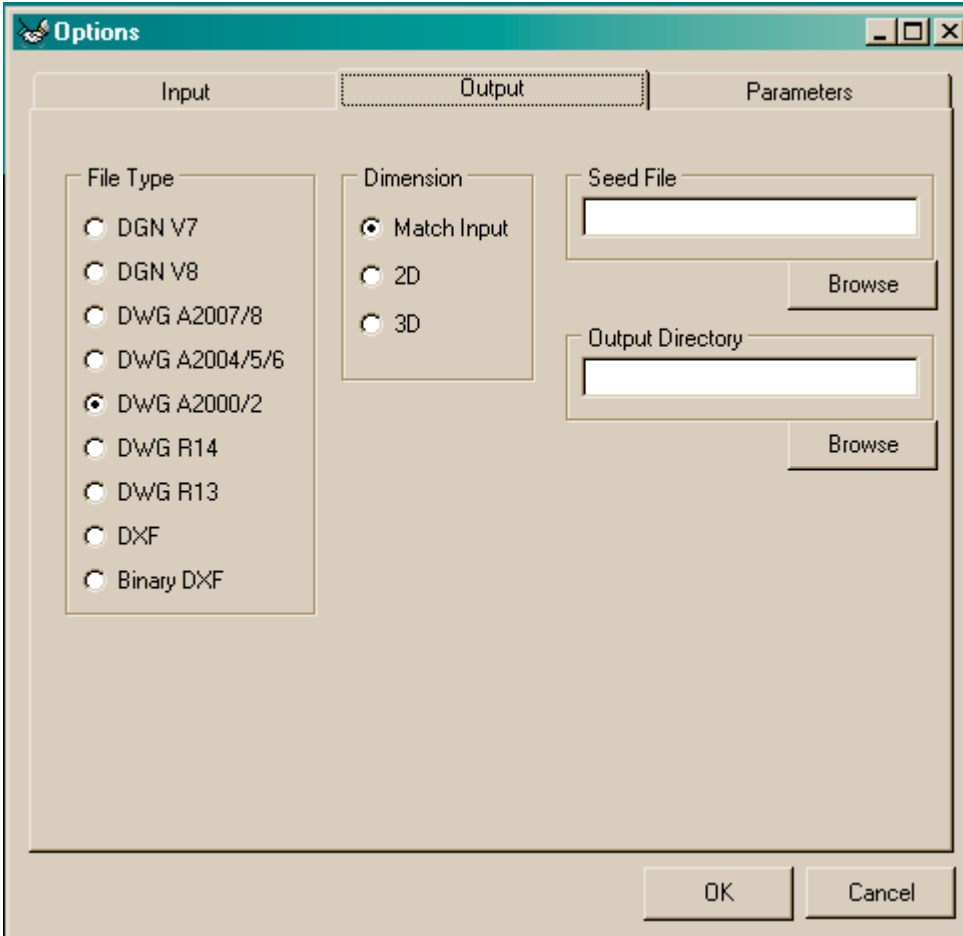
1. the directory of the DGN or DWG being translated
2. the directory given in the attachment
3. the directories given in the **Reference File Path** response

Note:

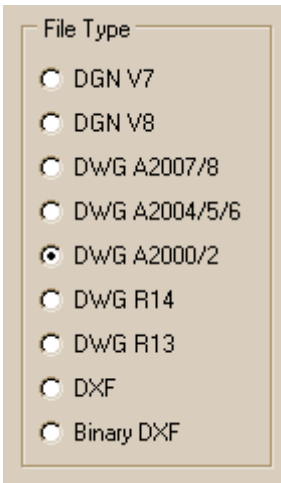
The Use Saved Reference / Xref Path option in the Parameters Tab forces the directory given in the attachment to be checked before the directory of the main DGN or DWG.

Output Options

The **Output** tab defines the type and characteristics of the output files to be generated by the translator.



File Type

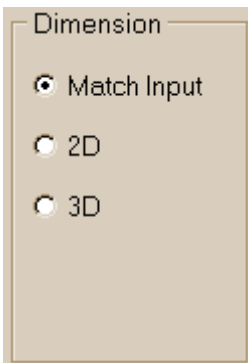


A dialog box titled "File Type" containing a list of radio button options for file formats and versions.

- DGN V7
- DGN V8
- DWG A2007/8
- DWG A2004/5/6
- DWG A2000/2
- DWG R14
- DWG R13
- DXF
- Binary DXF

The **File Type** selection list provides you the capability to choose the file format and version to be generated.

Dimension

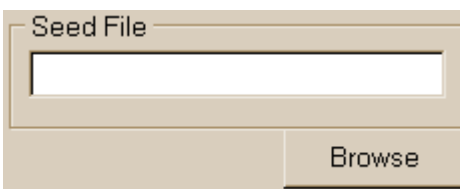


A dialog box titled "Dimension" containing three radio button options for output dimensionality.

- Match Input
- 2D
- 3D

The **Dimension** selection list provides the ability to choose the desired dimension of the output file(s) to be generated. This option is only available when converting to DGN format. DWG files store both 2D and 3D entities in the same file.

Seed File



A dialog box titled "Seed File" with a text input field and a "Browse" button.

The **Seed File** prompt is used to specify the seed (MicroStation) or prototype (AutoCAD) file to be used when creating the output files. The seed file only needs to be specified when a project seed file different from the default seed file is required. The **Browse** button is used to search for the seed file. When no seed file is specified, CATS uses the default seed files located in the **CATS** directory.

The default seed and prototype files supplied with CATS are:

- AcadSeed.DWG - AutoCAD DWG prototype
- SEEDARC2.DGN - MicroStation 2D seed with architectural units
- SEEDARC3.DGN - MicroStation 3D seed with architectural units
- SEEDMAP2.DGN - MicroStation 2D seed with mapping units
- SEEDMAP3.DGN - MicroStation 3D seed with mapping units
- SEEDME2.DGN - MicroStation 2D seed with mechanical units
- SEEDME3.DGN - MicroStation 3D seed with mechanical units
- 2DSEED.DGN - MicroStation 2D seed with standard default units
- 3DSEED.DGN - MicroStation 3D seed standard default units
- V8SeedArch2.DGN – MicroStation V8 2D seed with architectural units
- V8SeedArch3.DGN – MicroStation V8 3D seed with architectural units
- V8SeedMetric2.DGN – MicroStation V8 2D seed with metric units
- V8SeedMetric3.DGN – MicroStation V8 3D seed with metric units

Output Directory

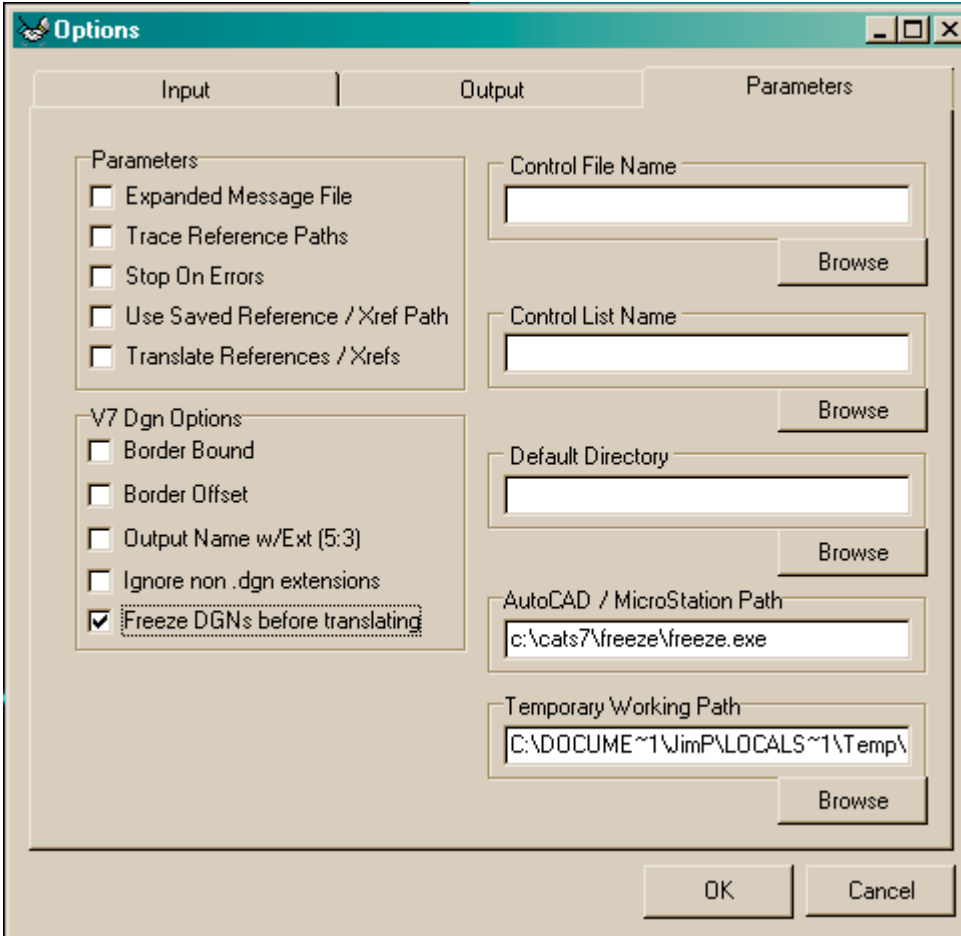
The **Output Directory** prompt allows you to define an alternate directory to write the output files. By default, the output files are written in the same directory as the input files.

Use Existing Cell Library

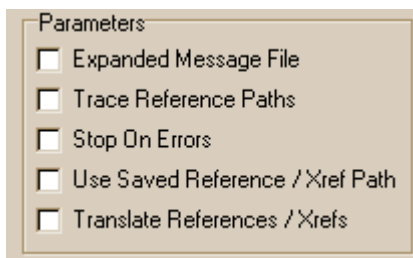
The **Use Existing Cell Library** prompt allows you to define an existing cell library to use for V7 DGN output files. This cell library can be used to define cells to be used in place of the blocks from the input DWG file. If a cell with the same name as the block being translated is found in the existing cell library it will be used in place of the block from the DWG, otherwise a new cell will be created in the cell library. The DWG block name can be changed to a different cell name with Cell Command in the control file. The placement point and scale will be taken from the original DWG block.

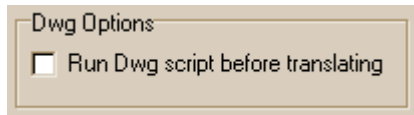
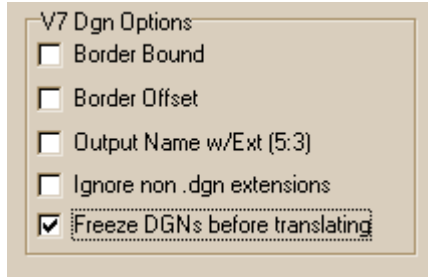
Parameter Options

The **Parameters** tab defines the general translation parameters.



Parameters





The **Parameters** list is used to select a variety of processing options. You can choose any number, all, or none of the following options:

Expanded Message File

This option lists the contents of the control files and error messages in the cats8.log file. This is useful for debugging control files. The control statement causing the error will be listed right before the error message in the log file.

Trace Reference Paths

This option will list all attempts to locate DGN reference or DWG Xref files using the various search paths and options in the **cats8.log** file. This is useful if CATS is having trouble finding reference or Xref files.

Stop On Errors

This option is used when translating multiple files. Normally, when CATS encounters an error translating a file, it will attempt to proceed to the next file in the list. If errors are encountered and **Stop On Errors** has been checked, you will be given the option to continue or stop translating. Errors will be listed in the **cats8.log** file.

Use Saved Reference / Xref Path

This option is used to force the DGN reference file or Dwg Xref file path saved in the main DGN or DWG file to be used instead of the normal CATS search scheme for finding references or Xrefs.

Translate References / Xrefs

This option is used to automatically translate all attached DGN reference files or DWG Xrefs along with the main DGN or DWG file. This option is normally used when the FileToRef or FileToXref command is used in the control file. The normal mode for CATS only translates the selected files. References or Xrefs must be translated separately.

Border Bound

This option is used in conjunction with special border cells when the input file type is V7 DGN. In order for this option to take effect you must have a reference file attached with the logical name **BDR**. The reference file must have a cell named **LFTBTM** placed at the left bottom corner of the drawing and a cell named **RHTTOP** placed at the right top corner of the drawing. You can then check this option to use border cells **LFTBTM** and **RHTTOP** to

define the left bottom and right top corners of the clipping boundary, respectively. All design file elements outside this clipping boundary will be excluded from the output file. **Border Bound** can be used with or without the use of **Border Offset**. Valid only for V7 DGN input files.

Border Offset

This option is used in conjunction with special border cells when the input file type is V7 DGN. In order for this option to take effect you must have a reference file attached with the logical name **BDR**. The reference file must have a cell named **LFTBTM** placed at the left bottom corner of the drawing and a cell named **RHTTOP** placed at the right top corner of the drawing. You can then check this option to use border cell **LFTBTM** for the drawing offset location. **Border Offset** can be used with or without the use of **Border Bound**. Valid only for V7 DGN input files.

Output Name w/Ext (5:3)

This option is used to generate output file names using the first five characters of the V7 DGN file name followed by the three-character input file extension. The new extension will always be .DWG. This is useful when you have a group of DGN files with the same name and different extensions. Since AutoCAD only allows .DWG as the file extension, this option provides a means of generating file names that avoid duplication. It also limits file names to the 8.3 file format. Note that to properly use this feature your file names must be unique within the first 5 characters of the file name. Valid only for V7 DGN input files.

Ignore non .dgn extensions

This option is used to generate output file names using only the filename with the .dwg extension. Normally CATS will append non .dgn extension to filename and then add the .dwg extension to make sure the resulting DWG filenames are unique. For example, main.s01 will normally be translated to main_s01.dwg. With this option selected, main.s01 will be translated to main.dwg. This option should only be used when you are sure the resulting filenames will be unique. Valid only for V7 DGN input files.

Freeze DGNs before Translating

This option will activate the freeze option prior to translating to AutoCAD. Freezing is necessary to allow translation of MicroStation associative dimensions, multilines, and associative patterns. This option requires that MicroStation be available from the translation system. The freeze MDL components must be copied to the appropriate MicroStation directories. See **freeze.txt** for details. Freeze will create a copy of the file, then freeze and translate the copy. Valid only for V7 DGN input files.

Run Dwg script before Translating

This option will activate an AutoCAD script prior to translating to MicroStation. With this option, you can run any AutoCAD script to preprocess the DWGs before translating. This option requires that AutoCAD be available from the translation system. Two predefined scripts are included with CATS – R12.scr to convert to R12 format and R3.scr to convert to R13 format. Valid only for DWG input files.

Control File Name

The **Control File Name** prompt is used to select the project control file to be used for the translation. The **Browse** button allows you to search for the control file. The control file commands are described in detail in Customizing CATS. If not specified, CATS uses the system defaults.

Control List Name

The **Control List Name** prompt is used to select the control list file to be used for the translation. The **Browse** button is used to search for the control list file. The control list file is used for projects needing multiple different control files based on the type of file being processed. The control list file is described in detail in Customizing CATS.

Default Directory

The **Default Directory** prompt is used to specify a list of folders to be searched to locate control files, control list files, seed files, and other configuration files. Semicolons separate the folders, e.g.,

```
c:\temp\;f:\dgn\sub\refa\;h:\mydir\
```

The CATS folder is always included as a default directory.

AutoCAD / MicroStation Path

These prompts are used when the **Freeze DGNs before translating** or **Run DWG script before translating** options have been selected. The AutoCAD / MicroStation path gives the location of the AutoCAD or MicroStation

executable. The temporary working path gives the location where the DGN or DWG files are copied for the preprocessing application.

Translating Files

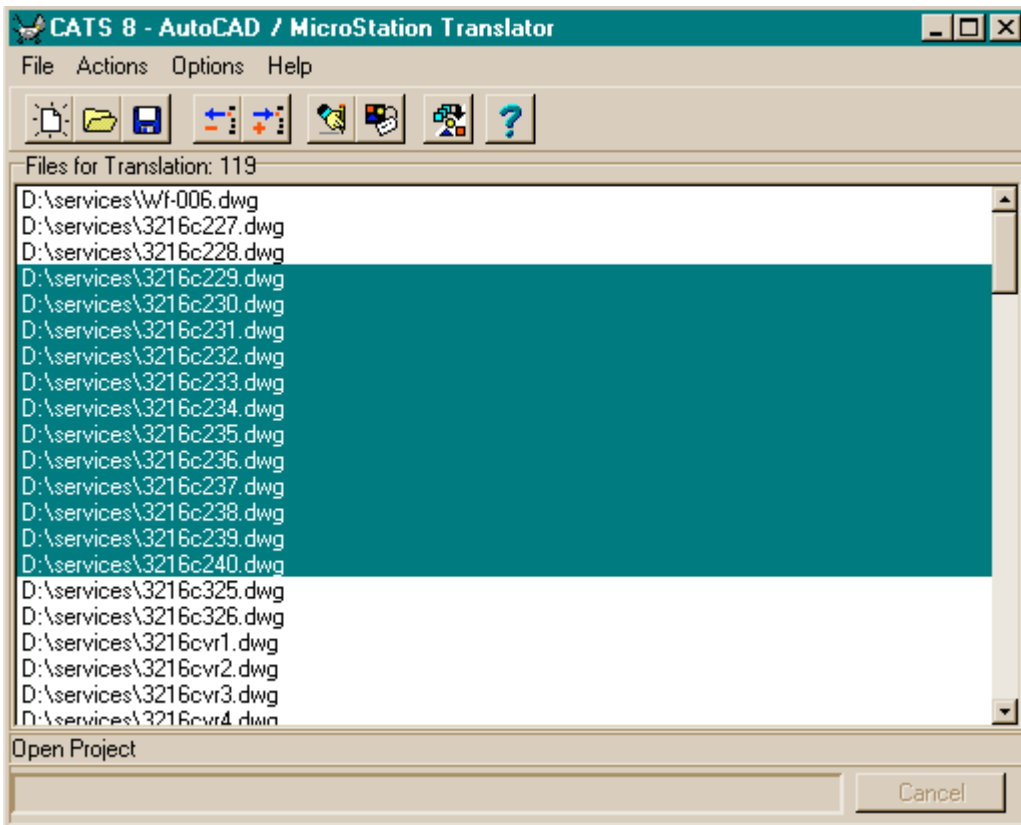
To translate files with CATS, you must first create or open a project. The project defines the input, output, parameters, and files to translate.

To translate files in a project:

- Select the files to translate.
- Pick the Translate command from the Actions Menu.
- Check the progress until complete.

Selecting Files for Translation

The **Files For Translation** list contains the file names of the project's files to be translated.



Files can be selected with the mouse and the <Ctrl> or <Shift> keys. To select a single file, click on the file name with the left mouse button. To select a range of files, click on the first file name with the left mouse button, then press and hold the <Shift> key and click on the last file name with the left mouse button. All of the files between

the two clicks will be selected. To select individual files click on each file name with the left mouse button while pressing the <Ctrl> key. These two methods can be combined to select the files to be translated. The **Select All** menu command will select all files.

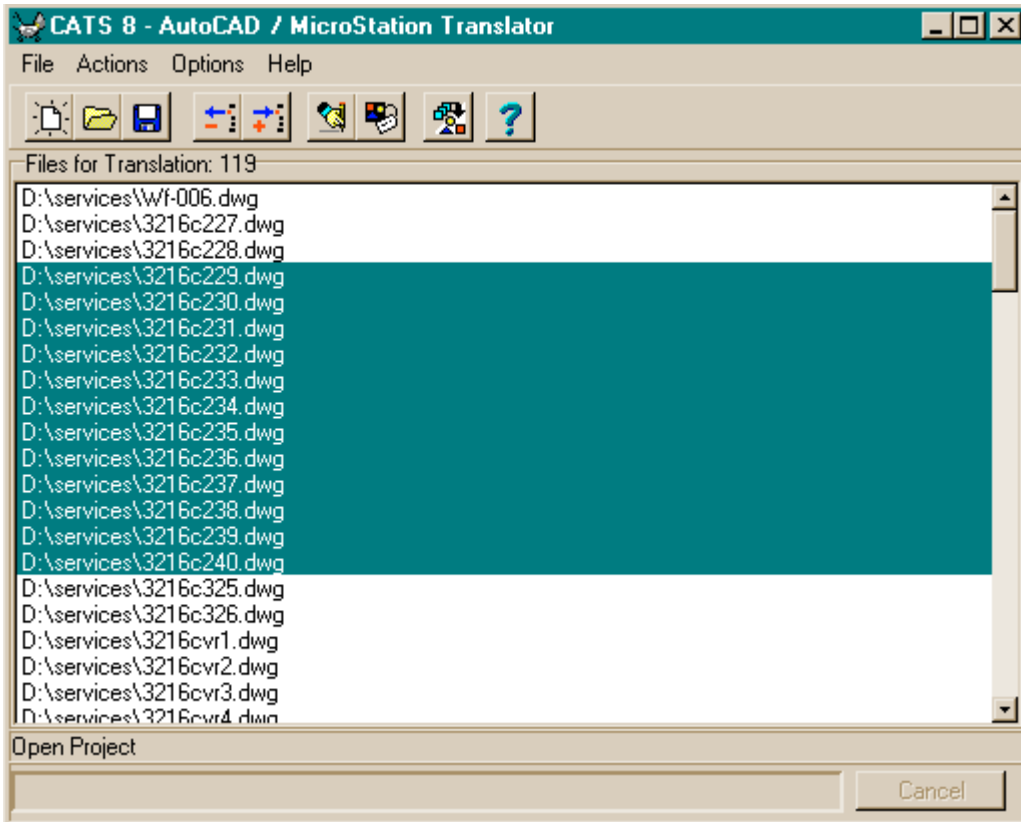
Select All

<u>A</u> ctions	<u>O</u> ptions	<u>H</u> elp
<u>A</u> dd File		Ctrl+I
<u>R</u> emove File		Ctrl+R
Cr <u>e</u> ate <u>C</u> ombined Control File		
Cr <u>e</u> ate <u>S</u> eparate Control Files		
<u>S</u> elect All		Ctrl+A
<u>D</u> eselect All		Ctrl+D
<u>T</u> ranslate		Ctrl+T

The **Actions** menu item **Select All** can be used to select all of the files in the **Files For Translation** window.

Deselecting Files

The **Files For Translation** list contains the file names of the project's files to be translated.



The mouse and <Ctrl> key can be used to deselect files in the **Files For Translation** window. To deselect specific files, click on the file name with the left mouse button while pressing the <Ctrl> key. The **Deselect All** menu command will deselect all files in the list.

Deselect All

Actions	Options	Help
Add File		Ctrl+I
Remove File		Ctrl+R
Create Combined Control File		
Create Separate Control Files		
Select All		Ctrl+A
Deselect All		Ctrl+D
Translate		Ctrl+T

The **Actions** menu item **Deselect All** can be used to deselect all of the files in the **Files For Translation** window.

Translate Command

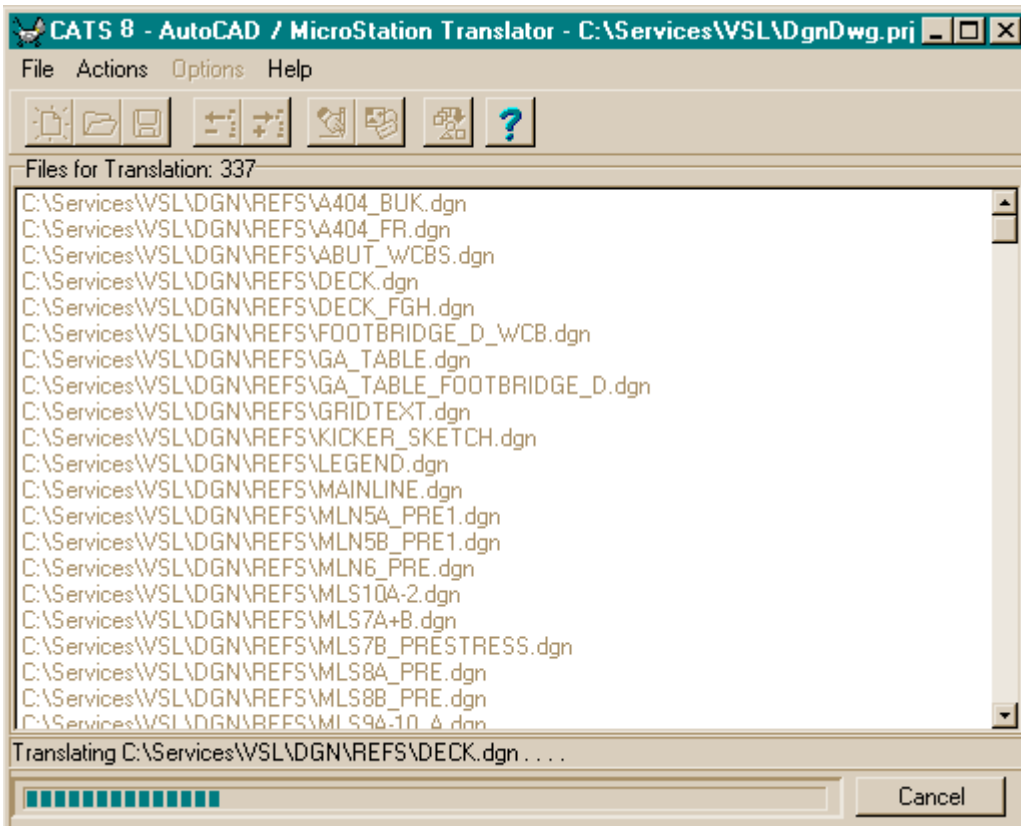
Once you have selected a file or list of files for translation, start the translation by clicking the **Translate** button



or by selecting the **Actions** menu item **Translate**.

Actions	Options	Help
Add File		Ctrl+I
Remove File		Ctrl+R
Create Combined Control File		
Create Separate Control Files		
Select All		Ctrl+A
Deselect All		Ctrl+D
Translate		Ctrl+T

Translation Progress



The files are translated one at a time. As each file is translating the status bar shows the current file name and the progress bar is updated as the translation proceeds.

Canceling a Translation



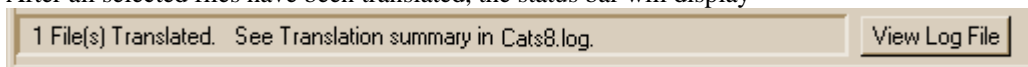
Clicking on the **Cancel** button will terminate the translation processing.

The status bar will then display:



Checking the Results

After all selected files have been translated, the status bar will display



showing the total number of files translated.

CATS creates the message file **cats8.log** for each execution. The message file contains a list of the files translated, including reference files and control files used. Click the **View Log File** button to view the log file. If the ExpandedMessageFile option was selected, the contents of control files are also listed. The message file also lists error messages for any errors encountered. If an error was encountered in a control file, processing is terminated at that point. With the ExpandedMessageFile option checked, the line containing the error will be the last line printed. A sample message file follows:

```
CATS Version 8
Copyright (c) Decision Graphics, Inc. 1994 - 2009. All Rights Reserved.
Main Control file: c:\cats8\sample\main.con
**** Processing input file: c:\cats8\sample\ref1.dgn Using ref.con
           Output file: c:\cats7\sample\ref1.dwg
Main Control file: c:\cats8\sample\main.con
**** Processing input file: c:\cats8\sample\main.dgn
Output file: c:\cats8\sample\main.dwg
Processing reference: refx.dgn
Main Control file: c:\cats8\sample\main.con
**** Processing input file: c:\cats\sample\main.s01 Using main.con
Output file: c:\cats8\sample\main.dwg
Processing reference: ref2.dgn89.refx.dgn
Processing reference: ref1.dgn Using ref.con
```

Customizing Cats

You can customize CATS translations by using **Control Files** and **Control List Files**. Commands are entered in text files, called control files, normally with the .con extension. A new control file is normally defined for each project. If a project has several different types of files, a separate control file is used for each different file type. The control list file, normally with the .lst extension, is then used to select the control file to be used for each file type.

A **Control File** is a text file (with a .con extension) which contains commands and rules for customizing a translation.

A **Control List File** is a text file (with a .lst extension) which contains a list of files and the associated control file to use.

Note:

- If you want to handle all of the files in the translation project the same, use a single control file and no control list file.
- If you have files that fall into several different classes each needing specific customization, create a control file for each class and use a control list file to associate the files to the desired control file.

Control Files (.con)

Control files are ASCII text files containing commands and rules to customize your translations. A control file can be created manually with your favorite text editor or by allowing CATS to scan the file(s) to be translated and generating a default control file or files.

Control files are free form text files containing comments, commands, and rules. One or more spaces, tabs, or commas separate each command or rule. Commands or rules can be contained on a single line or continued on several lines for clarity. Blanks lines can be used for additional clarity. Commands and rules can be entered in upper or lower case or a mixture.

;Comments

A comment can be any text to describe or document your control file. It can appear anywhere in the file. A comment must begin with a semicolon ; in the first position of a line.

Example

```
; This is a comment
```

Commands

Commands are single words or phrases that give an action to take - like assign a layer or bind xrefs. Commands can be entered on separate lines or on the same line separated by spaces, tabs, or commas.

Examples

```
Layer=walls
BindXref
color=3 weight=4, level=34
```

Rules

Rules are qualifiers that limit the commands to specific part of the file being translated. Rules have two parts - the selection criteria and the commands to perform. The commands are performed only for the entities that match the selection criteria. The commands are ignored for all other entities. Rules always are of the form:

If selections Then commands Endif

The selections are a list of criteria (level/layer, color, textstyle, etc) which select the entities to apply the commands to. Selections may also contain wildcards. The commands are a list of actions to modify the selected entities.

Examples

```
If Level=34 Then Layer=Walls Endif
If Level=23 Then Layer=Floor Color=3 Width=0.5 Endif
If Layer=Arch, Color=5 Then Level=23 Endif
If TextStyle=Romans Layer=Text Then
    Font=3
    TextWidthFactor=0.95
Endif
```

Rules are free from and can spaced out across multiple lines or within the same line to improve readability. The rule must begin with **If**, followed by the selections. **Then** is used to separate the selections from the commands. The commands are next. **Endif** is used to end the rule. Spaces, tabs, or commas can separate the individual selections and commands.

Processing Commands and Rules

Control files usually have many commands and rules. Each entity in the input file is examined to see if the commands and rules apply. The commands and rules are applied one by one from the top down. Every command and rule is used for every entity. Several rules may apply to a particular entity and the final result will be the combination of all commands applied. Thus it is a good idea put your more general rules first followed more specific rules.

Example

```
If Layer=Wall Then Level=2 Endif
If Layer=Wall LineType=Dashed Then Level 3 Endif
```

In this example, a line on layer Wall and linetype Continuous would be assigned to level 2. A line on layer Wall and linetype Dashed would be assigned to level 3. If the rules were reversed -

```
If Layer=Wall LineType=Dashed Then Level 3 Endif
```

```
If Layer=Wall Then Level=2 Endif
```

both lines would match the second rule and be assigned to level 2.

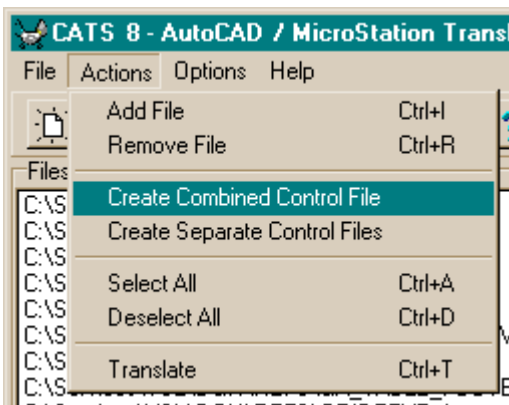
Selections always check the characteristics of the original entity. If you have a rule which changes the color of an entity. A following rule such as:

```
If Color=3 Then Color=4 Endif
```

will check the original color, not the color as changed by a previous rule.

Create Combined Control File

You can use the **Create Combined Control File** menu option to scan a list of files and create a default control file based on the characteristics of your files.



Select **Create Combined Control File** when you need all files being translated to use the same translation commands and rules. A single control file named project.con will be generated by scanning all files highlighted in the Files For Translation list. **Project.con** will contain default assignments for all cells, layers, linetypes, etc used in any one or all of the highlighted files.

Control List Files (.lst)

Customizing commands are entered in text files, called control files, normally with the .con extension. If a project has several different types of files, a separate control file can be used for each different file type. The control list file, normally with the .lst extension, is then used to select the control file to be used for each file type. You can use your favorite text editor to create and maintain your control files and control list files.

The control list file contains a list of drawing files and corresponding control file names to be used for each drawing as it is processed. This option is used when a single control file is not sufficient to handle the translation commands for a project. A control file list can be used with the standard control file or by itself. If used with the standard

control file, the standard control file commands will be evaluated first followed by the commands from the appropriate control file as given in the control list file. The format of the control list file is described below.

If all the rules and definitions you set up in the standard control file apply to all of your data, you do not need to set up a control list file. However, if you need different parameters for different types of drawing files, the control list file lets you specify a separate control file for each drawing file.

The control list file is an ASCII file containing a list of drawing file/control file pairs. CATS will select the appropriate control file to be used based on the current file name being processed. If a standard control file was also selected, its commands will be applied first followed by the commands from the control file selected from the control list file.

The format of the control list file is:

```
drawing_file spec          control_file
drawing_file spec          control_file
```

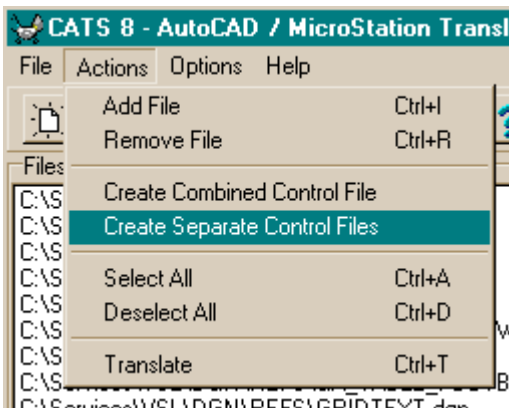
DOS wildcards can be used with the drawing file names. Lines beginning with a semi-colon (;) will be treated as comments. For example:

```
;
; Control List File for ABC Project
;main.s01          main.con
ref*. *           ref.con
*.bdr             border.con
c:\abc\*. *       allabc.con
```

Rules in **main.con** will be applied when translating **main.s01**. **Ref.con** will be applied to all files beginning with **ref**; rules in **border.con** will be used for all files with the extension **.bdr**; and rules in **allabc.con** will be used for all files on drive **c:** in directory **\abc**.

Create Separate Control Files

You can use the Create Separate Control Files menu option to scan a list of files and create a control list file and individual default control files for each of the files in the list.



Select **Create Separate Control Files** when you need each file being translated to use different translation commands and rules. A control list file will be generated matching each file to its corresponding control file. Also, a control file will be generated for each file highlighted in the Files For Translation list. The file name will be the same as the drawing name and will use **.con** for the extension. The *drawingname.con* files will contain default assignments for all cells, layers, linetypes, etc contained in that file only.

Commands

Commands are single words or phrases that instruct CATS to modify the translation of an entity - like assign a layer or bind xrefs. Commands can be entered on separate lines or on the same line separated by spaces, tabs, or commas. Commands are also used with rules to restrict the commands to selected entities.

Commands

ArcStrokes	DeleteASE-LPN	Level	TextHeightFactor
AddToModelspace	EditText	Linetype	TextObliqueAngle
AddToPaperspace	EnterData	LowerLeftText	TextStyle
AttachWithSavedPath	ExcludeElement	LwDefault	TextWidthFactor
ArrtributeTag	ExitRules	MakeAnonymousBlock	UseExistingEnterdata
AttributeToTag	ExplodeCell	MakeLinkage	UseGlobalOriginForInsbases
AttDefToText	FileBlockName	NoGlobalOriginAdjustment	Weight
BCurveStrokes	FileToBlock	NoXclip	Width
BindXref	FileToRef	Offset	3DElevation
Block	FileToXRef	PlaceEndCaps	
Cell	FittedText	PlineToShape	AttributeName
Color	Font	PointCell	ForceType2Cells
ConversionFactor	FontToShape	Polyline	ForceAllLayersOn
CurveStrokes	FontToSymbol	ProcessLeftOverTags	MakeUserBlock
CustomLinetype	ForceCellLevel	ReferenceLevelSymbology	TextToLine
CustomLinetypeScale	ForceMuTag	ScaleCell	
DefineHeader	Force3DPolylines	Set	
DefineLayers	FractionStack	SkipFile	
DefineStyles	IncludeElement	SkipPoint	
DefineDxfHeader	KeepAllReferences	SkipReferences	
DefineDxfLayer	Layer	SymbolFont	

ArcStrokes Command

Dgn > Dwg

Purpose

Use **ArcStrokes** to define the number of degrees per segment to be used when stroking a MicroStation elliptical arc for AutoCAD R12 polylines.

Format

ArcStrokes = *strokeno*

Arguments

strokeno = a decimal value representing the number of degrees per stroked segment desired.

Usage

ArcStrokes is only valid when translating from MicroStation to AutoCAD.

Example

ArcStrokes = 10

will stroke R12 elliptical arcs in **10** degree segments.

Remarks

By default, the value used will be **2** degrees per segment. Increasing this value will decrease the number of vertices generated and the AutoCAD file size accordingly.

AddToModelspace

Dgn > Dwg

Purpose

Use **AddToModelspace** to specify entities to convert AutoCAD modelspace.

Format

AddToModelspace

Usage

AddToModelspace is only valid for MicroStation to AutoCAD translations.

Example

AddToModelspace

will translate all entities to AutoCAD modelspace.

If Level=1 Then AddToModelSpace

will translate all level 1 entities to AutoCAD modelspace.

AttachWithSavedPath**Dgn <> Dwg**

Purpose

Use **AttachWithSavedPath** to use the full saved path when attaching xrefs or reference files..

Format

AttachWithSavedPath

Usage

AttachWithSavedPath is valid when translating both directions. It is used with the FileToRef or FileToXref command.

Example

FileToXref AttachWithSavedPath

will translate reference files to xref using the path saved in the Dgn file.

AddToPaperspace

Dgn > Dwg

Purpose

Use **AddToPaperspace** to specify entities to convert AutoCAD paperspace.

Format

AddToPaperspace

Usage

AddToPaperspace is only valid for MicroStation to AutoCAD translations.

Example**AddToPaperspace**

will translate all entities to AutoCAD paperspace.

If Level=1 Then AddToPaperspace

will translate all level 1 entities to AutoCAD paperspace.

AttributeName

Dgn > Dwg

Purpose

Use **AttributeName** to assign the attribute name for attribute created from Dgn tags.

Format

AttributeName = newname

Arguments

newname = AutoCAD attribute name.

Usage

AttributeName is only valid when translating from MicroStation to AutoCAD.

Example

```
If TagName=Valve Then  
    AttributeName=AValve  
Endif
```

will assign the attribute name Avalue to the attribute created from the tag named Valve.

AttributeTag

Dgn > Dwg

Purpose

Use **AttributeTag** attribute tag names to attributes created from enterdata text. Each successive enterdata field text in a cell will be assigned the specified tag name.

Format

AttributeTag = tagname

Arguments

tagname = AutoCAD tag name.

Usage

AttributeTag is only valid when translating from MicroStation to AutoCAD.

Example

If Cell=Valve Then

AttributeTag=ValveNumber

AttributeTag=ValveSize

AttributeTag=ValveManufacturer

Endif

will assign attribute tags are assigned to each successive enterdata field as encountered for the selected cell.

AttributeToTag Command

Dwg > Dgn

Purpose

Use **AttributeToTag** to convert AutoCAD attributes to MicroStation tags instead of enter data fields.

Format

AttributeToTag

Usage

AttributeToTag is only valid for AutoCAD to MicroStation translations.

Example

AttributeToTag

will translate all AutoCAD attributes on all layers to MicroStation tags.

AttDefToText Command

Dwg > Dgn

Purpose

Use **AttDefToText** to convert AutoCAD attribute definitions in a block to regular MicroStation text in the cell definition in the cell library. The Attdéf default value is used for the text string. If no default value is given, then the attdéf tag is used.

Format

AttDefToText

Usage

AttDefToText is only valid for AutoCAD to MicroStation translations.

Example

AttDefToText

will convert attdéfs in block to regular text in the cell library.

BCurveStrokes Command

Dgn > Dwg

Purpose

Use **BCurveStrokes** to adjust the number of points generated by the BSpline curve stroking process.

Format

BCurveStrokes = *nn*

Arguments

nn = the number of points to add between each pair of original vertices while stroking the element. The larger the number of points, the smoother the resultant curve.

Usage

BCurveStrokes is only valid when translating from MicroStation to AutoCAD.

Example

BCurveStrokes = 10

will stroke all **BSplines** with **10** points generated between original vertices.

Remarks

By default, the value used is **3**.

BindXref Command

Dwg > Dgn

Purpose

Use **BindXref** to bind an xref in a Dwg into the output Dgn file. The model space entities of the Xref on displayed layers will be copied into the main Dgn file.

Format

BindXref

Usage

BindXref is only valid when translating from AutoCAD to MicroStation.

Example

BindXref

will bind all Xrefs into the output Dgn file.

Remarks

By default, Xrefs are ignored.

Block Command

Dgn > Dwg

Purpose

Use **Block** to assign the block name to be used.

Format

Block = blockname

Arguments

blockname = AutoCAD block name.

Usage

Block is only valid when translating from MicroStation to AutoCAD.

Example

```
If Cell=arrow Then
  Block = NewArrow
Endif
```

will assign the block name NewArrow when the cell Arrow is used.

Cell Command

Dwg > Dgn

Purpose

Use **Cell** to assign the cell name to be used.

Format

Cell = *cellname*

Arguments

cellname = MicroStation cell name 1 to 6 characters in length.

Usage

Cell is only valid when translating from AutoCAD to MicroStation.

Example

```
If Block = LONGBLOCKNAME Then  
  Cell = ABC  
Endif
```

will assign the name **ABC** to the cell created from the AutoCAD block **LONGBLOCKNAME**.

Remarks

By default, the AutoCAD block name will be used as the cell name. If the AutoCAD name is longer than 6 characters, a new name of the form **zznnnn** will be assigned.

Color Command

Dgn <> Dwg

Purpose

Use **Color** to assign the output color to elements.

Format

Color = *colorno*

or:

Color = BYLAYER

or:

Color = BYBLOCK

Arguments

colorno = integer color number to be assigned. User variables may be used to assign color number.

BYLAYER indicates the color will be set per the layer table. BYLAYER is only valid when generating an AutoCAD file.

BYBLOCK indicates the color will be assigned at block placement time. BYBLOCK is only valid during MicroStation cell to AutoCAD block translation.

The default color table follows:

<u>AutoCAD</u>	<u>MicroStation</u>	<u>Color</u>
1	3	red
2	4	yellow
3	2	green
4	7	cyan
5	1	blue
6	5	magenta
7	0	white
8	9	gray
30	6	orange

Usage

Color is valid for translating both directions.

Example

```
If Block=* Then
  Color = BYLAYER
Endif
```

will select all blocks and assign the color BYLAYER basis.

ConversionFactor Command

Dgn <> Dwg

Purpose

Use **ConversionFactor** to set a multiplier to be applied to all coordinates, scales, and text sizes..

Format

ConversionFactor = *factor*

Arguments

factor = the decimal multiplier to be applied to all coordinates during translation.

Usage

ConversionFactor is valid for translating both directions.

Example

conversionfactor = 2

will double the size of all entities in the drawing being translated.

Remarks

By default, conversion factor will be set to **1** resulting in no additional scale being applied during translation.

CurveStrokes Command

Dgn > Dwg

Purpose

Use **CurveStrokes** to adjust the number of points generated by the curve stroking process.

Format

CurveStrokes = *nn*

Arguments

nn = the number of points to add between each pair of original vertices while stroking the element. The larger the number of points, the smoother the resultant curve.

Usage

CurveStrokes is only valid when translating from MicroStation to AutoCAD.

Example

CurveStrokes = 8

will stroke curves with **8** points generated between original vertices.

Remarks

By default, the value used is **3**.

CustomLinetype Command

Dwg > Dgn

Purpose

Use the **CustomLinetype** to set MicroStation custom linetypes.

Format

CustomLinetype = *mscustom*

Arguments

mscustom = the name of the MicroStation custom linetype

Usage

The **CustomLinetype** action is only valid for AutoCAD to MicroStation translations.

Example

If Linetype = DASHED Then CustomLinetype = BOUNDARY Endif

will select all AutoCAD entities with the linetype **DASHED** and translate to MicroStation referencing the **BOUNDARY** custom linetype.

Remarks

If the custom linetype name contains parentheses () or blanks, you must enclose the name in single quotes, e.g.,

If Linetype = BORDER then CustomLinetype = '(BORDER)' Endif

Note that the actual linetype definitions are not translated by CATS. This command provides a means for you to assign the appropriate MicroStation custom linetype based on the AutoCAD linetype name. The custom linetype definition must have been loaded into the MicroStation seed file before translation.

CustomLinetypeScale Command

Dwg > Dgn

Purpose

Use **CustomLinetypeScale** to specify the scale for custom linetypes.

Format

CustomLinetypeScale = *scale*

Arguments

scale = custom linetype scale to be written to the entity definition.

Usage

The **CustomLinetypeScale** action is only valid for AutoCAD to MicroStation translations. It must be used with the **CustomLinetype** command.

Example

CustomLinetype = MyLine, CustomLinetypeScale = 5
will set the custom linetype to **MyLine** and its scale to **5**.

Define Dwg Header Command

Dgn > Dwg

Purpose

Use the **Define Header** sequence to define any AutoCAD header variables that need to be set up for your project.

Format

```
Define Header:  
  HeaderVariable = value  
  ...  
  ...  
  ...  
  HeaderVariable=value  
EndHeader
```

Arguments

HeaderVariable = the AutoCAD header variable name. Refer to your AutoCAD documentation in the section describing the DXF format for a complete list of header variables and valid values for each variable. It is required.

value = the value you wish to assign to the variable. Refer to your AutoCAD documentation for a complete list of header variables and valid values for each variable. It is required.

Usage

Define Header is only valid when translating from MicroStation to AutoCAD. It cannot be used as part of a rule.

Example

```
Define Header:  
  DIMSTYLE=ROMANS  
  MENU= ALLEBG  
  TEXTSTYLE=ROMANS  
EndHeader
```

Remarks

You may assign as many header variables as you wish in the **Define Header** sequence. Each individual variable may only be assigned once per file.

Header variables can also be defined conditionally. Refer to the DefineDXFHeader command.

Define Layer Command

Dgn > Dwg

Purpose

Use the **Define Layer** sequence to set up AutoCAD layer table definitions.

Format

```

Define Layer:
  Name = layername
  Color = layercolor
  Linetype = layerlinetype

  Weight = layerweight
  Output layeroption
  ...
  ...
  ...
EndLayer

```

Arguments

layername = the AutoCAD layer name 1 to 31 characters in length. It is required.

layercolor = the AutoCAD color number assigned to the layer. It is optional. If not specified, *layercolor* defaults to 7 (white).

layerlinetype = the AutoCAD linetype name 1 to 10 characters in length assigned to the layer. It is optional. If not specified, *layerlinetype* defaults to **CONTINUOUS**.

layerweight = the AutoCAD 2000++ lineweight assigned to the layer or Default. It is optional. If not specified, *layerweight* defaults to **DEFAULT**. The weight value is a integer which represents hundredths of mm, e.g. the value 5 represents .05 mm width.

layeroption = **always** or **if_used**. If **always** is selected, the layer definition will be written to the AutoCAD file whether any entities are placed on the layer or not. If **if_used** is selected, the layer definition will only be written if entities have been placed on the layer. It is optional. If not specified, *layeroption* defaults to **always**.

Usage

Define Layer is only valid when translating from MicroStation to AutoCAD. Multiple layers can be defined within a **Define Layer...EndLayer** sequence. The keyword **Name =** begins a new layer table entry. It cannot be used as part of a rule.

Example

```

Define Layer:
  Name = FLOORPLAN1
  Color = 2
  Linetype = DASHED

  Weight = 5
  Output If_used
  Name = ELECTRICAL Color = 7 Output Always
EndLayer

```

creates a layer table entry named **FLOORPLAN1**. The layer color is set to **2** and layer linetype is set to **DASHED**. This layer will not be written to the AutoCAD file if, at the end of translation, it contains no entities. This example also creates a layer table entry named **ELECTRICAL**. The layer color is set to **7**. The layer linetype is not specified and will default to **CONTINUOUS**. The **ELECTRICAL** entry will always be written in the layer table whether any entities reference it or not.

Remarks

Layers can also be defined conditionally. Refer to the [DefineDxfLayer](#) command. **Define Layer** only places a layer definition in the AutoCAD layer table. It does not place any entities on that layer. To assign the layer to an entity, use the Layer command, i.e.,

If Level=2 then Layer=ELECTRICAL Endif

Valid lineweight values for AutoCAD are: 0.00, 0.05, 0.09, 0.13, 0.15, 0.18, 0.20, 0.25, 0.30, 0.35, 0.40, 0.50, 0.53, 0.60, 0.70, 0.80, 0.90, 1.00, 1.06, 1.20, 1.40, 1.58, 2.00, 2.11

Define Style Command

Dgn > Dwg

Purpose

Use the **Define Style** sequence to set up AutoCAD text style table definitions.

Format

Define Style:

```
Name = stylename
Shape = styleshape
Height = styleheight
WidthFactor = stylewidth
ObliqueAngle = styleoblique
Output styleoption
```

```
...
...
...
```

EndStyle

Arguments

stylename = the AutoCAD text style name 1 to 31 characters in length. It is required.

styleshape = the AutoCAD shape file to be referenced. Since this is a DOS file name it must be 1 to 8 characters in length and adhere to DOS naming conventions. It is optional. If not specified, *styleshape* defaults to **TXT**.

styleheight = the decimal character height for text placed in AutoCAD with the style. It is optional. If not specified, *styleheight* defaults to **0.0**.

stylewidth = the decimal character width factor for text placed in AutoCAD with the style. It is optional. If not specified, *stylewidth* defaults to **1.0**.

styleoblique = the decimal obliqueing angle for text placed in AutoCAD with the style. It is optional. If not specified, *styleoblique* defaults to **0.0**.

styleoption = **always** or **if_used**. If **always** is selected, the text style definition will be written to the AutoCAD file whether any text entities reference it or not. If **if_used** is selected, the text style definition will only be written if at least 1 text entity references it. If not specified, *styleoption* defaults to **always**.

Usage

Define Style is only valid when translating from MicroStation to AutoCAD. Multiple text styles can be defined within a **Define Style: ... EndStyle** sequence. The keyword **Name =** begins a new text style table entry. It cannot be used as part of a rule.

Example

Define Style:

```
Name = SPECIAL
Shape = ROMANS
ObliqueAngle = 10
Output Always
```

```
Name = PLAIN Shape = MONOTXT Output Always
EndStyle
```

creates a text style table entry named **SPECIAL**. Text style **SPECIAL** uses font file **ROMANS** and has an obliqueing angle of **10** degrees defined. This style is defined whether it is used or not. This example also created a text style named **PLAIN**. **PLAIN** references shape file **MONOTXT** and is always defined whether it is used in the drawing or not.

Remarks

Define Style only places a style definition in the AutoCAD text style table. It does not assign the style to any text entities. To assign the text style to a text entity, use the `TextStyle` command, i.e,

```
If Font=1 Then TextStyle=PLAIN Endif
```

DeleteASE-LPN**Dgn > Dwg**

Purpose

Use **DeleteASE-LPN** to delete the ASE-LPN table record for R14 file to eliminate ASE errors messages for R14 Xrefs.

Format

DeleteASE-LPN

Usage

DeleteASE-LPN is only valid when translating from MicroStation to AutoCAD.

Example

DeleteASE-LPN

will delete the ASE-LPN table record from the output Dwg file.

Remarks

Only use this command if you have had ASE errors opening R14 Dwg files.

Dynamic DefineHeader Command

Dgn > Dwg

Purpose

Use **DefineHeader** when you only want the AutoCAD header variable assignments made when selection criteria have been met.

Format

If.....Then DefineHeader (*HeaderVariable=value*) Endif

or:

**If.....Then
 DefineHeader (*HeaderVariable=\$ReferenceScaleFactor*)
Endif**

Arguments

HeaderVariable = the AutoCAD header variable \$name. Refer to your AutoCAD documentation for a complete list of header variables and valid values for each variable.

value = the value you wish to assign to the \$variable. Refer to your AutoCAD documentation for a complete list of header variables and valid values for each. User and/or CATS system variables may be used for either a portion of or the full value to assign.

ReferenceScaleFactor = this value will be multiplied by the current reference file scale; the product will be written to the AutoCAD header.

Usage

DefineHeader is only valid when translating from MicroStation to AutoCAD.

Example

```
If FileName = ???????X, FileExt = BDR  
    Then DefineHeader (LTSCALE=0.25)  
Endif
```

will define the **LTSCALE** to **0.25** for files whose name ends with **X** and have the extension **.BDR**.

Dynamic DefineLayer Command

Dgn > Dwg

Purpose

Use **DefineLayer** when you only want the layer table entries created when selection criteria have been met.

Format

```
If.....Then DefineLayer = (layername,color,linetype) Endif
```

Arguments

layername = the AutoCAD layer name. User variables and/or CATS system variables may be used as a portion of or the complete AutoCAD layer name.

color = the AutoCAD color number assigned to the layer.

linetype = the AutoCAD linetype name assigned to the layer.

Usage

DefineLayer is only valid when translating from MicroStation to AutoCAD.

Example

```
If Level = 4 Then  
  DefineLayer = (%prefix%MYLAYER,4,CONTINUOUS)  
Endif
```

performs a conditional layer definition. When level **4** is encountered, the layer name will be composed of the contents of the user variable **%prefix%** followed by **MYLAYER**. The layer color will be **4** with linetype set to **CONTINUOUS**.

Remarks

Layers defined conditionally will always be written to the AutoCAD file's layer table whether used or not.

EditText Command

Dgn <> Dwg

Purpose

Use **EditText** to edit text on a character by character basis.

Format

EditText (matchstring To outputstring)

Arguments

matchstring = the input text string to be matched before the edit will occur.

outputstring = the text string to replace the matchstring in the output text string.

The following special characters are applicable to the matchstring and outputstring:

\nnn is converted to the decimal ASCII character value nnn

\\ is converted to a single \ character

Some special AutoCAD characters are represented by %% prefixes, i.e., %%**p** for plus/minus.

Usage

EditText is valid for translating both directions.

Example

EditText (TX To Texas)

EditText (%%d To \094)

will convert the AutoCAD degree symbol to the ASCII value 94.

Remarks

EditText is most useful for mapping special characters from one CAD system to the other. By default, the following characters are remapped automatically:

Degree Symbol:

AutoCAD %%d

MicroStation ASCII 094

Plus/Minus Symbol:

AutoCAD %%p

MicroStation ASCII 200

EnterData Command

Dgn <> Dwg

Purpose

Use **EnterData** to control handling of MicroStation enter data text.

Format

EnterData *mode*

Arguments

mode = **Enable** or **Disable**.

If **Enable** is selected, MicroStation enterdata text will be converted to/from AutoCAD attributes.

If **Disable** is selected, regular text elements will be generated instead of attributes/enter data text.

Usage

EnterData is valid when translating both directions.

Example

EnterData Enable

will translate attributes/enterdata text to enter data/attributes on the destination system.

Remarks

By default, enter data text is enabled and attributes generated.

ExcludeElement Command

Dgn <> Dwg

Purpose

Use **ExcludeElement** when you want to omit entities from the translation.

Format

ExcludeElement

Usage

ExcludeElement is valid for translating both directions.

Example

```
If TextStyle = ITALIC Then  
  ExcludeElement  
Endif
```

will select all AutoCAD text entities referencing text style **ITALIC** and omit them from the generated MicroStation file.

Remarks

By default, all elements will be translated unless specifically excluded.

ExitRules Command

Dgn <> Dwg

Purpose

Use **ExitRules** when you are satisfied with the status of a particular element based on the commands processed to this point. All following commands will be skipped.

Format

ExitRules

Usage

ExitRules is valid for translating both directions.

Example

```
If Cell = ELTM Then
  Layer = ELEC
  ExitRules
Endif
```

will place cell **ELTM** on the layer named ELEC and then terminate any further processing on this element.

Remarks

By default, each element will be processed for the full list of commands until the end is reached. When you do not want any further actions taken on a particular element, **ExitRules** will conclude processing for that element.

ExplodeCell Command

Dgn > Dwg

Purpose

Use **ExplodeCell** to drop MicroStation cells to their components during translation.

Format

ExplodeCell

Usage

ExplodeCell is only valid when translating from MicroStation to AutoCAD.

Example

```
If Cell = SCL* Then  
  ExplodeCell  
Endif
```

will drop all incoming MicroStation cells beginning with **SCL** to their components during translation.

Remarks

By default, MicroStation cells will be translated. If a cell is being exploded, the header is ignored and the elements within the cell are translated individually. **ExplodeCell** only applies to type 2 MicroStation cells. Shared cells (types 34 and 35) cannot be exploded.

FileBlockName Command

Dgn > Dwg

Purpose

Use **FileBlockName** to assign an AutoCAD block name to the block generated from the contents of a MicroStation reference file with the FileToBlock command. This command can also be used in conjunction with FileToXRef to specify the name of the XRef attachment created for a reference file.

Format

FileBlockName = *blockname*

Arguments

blockname = AutoCAD block name. User variables and/or CATS system variables may be used for a portion of or the complete block name.

Usage

FileBlockName is only valid when translating from MicroStation to AutoCAD.

Example

If FileName = DE* Then FileToBlock FileBlockName = ROOMDE Endif

will select all incoming data in files beginning with **DE** and place the contents in a block definition named **ROOMDE**.

Remarks

By default, the reference file's logical name will be used for the block name. If there is no logical name, the file name is used. FileToBlock must also be active for this to occur.

FileToBlock Command

V7 Dgn > Dwg

Purpose

Use **FileToBlock** to convert the contents of a reference file to an AutoCAD block definition and instance.

Format

FileToBlock

Usage

FileToBlock is only valid when translating from MicroStation to AutoCAD.

Example

FileToBlock

will convert the contents of all reference files to AutoCAD block definitions and instances.

Remarks

The block or XRef will be named per the FileBlockName command. If no FileBlockName was defined, the reference file's logical name will be used. If there is no logical name, the file name will be used. By default, elements in reference files will be merged with elements in the main design file to a single output file.

FileToRef Command

Dwg > Dgn

Purpose

Use **FileToRef** to convert an AutoCAD XRef attachment to a MicroStation reference file attachment.

Format

FileToRef

Usage

FileToRef is only valid when translating from AutoCAD to MicroStation.

Example

FileToRef

will convert the AutoCAD XRef xrefs to a MicroStation reference file attachments.

FileToXRef Command

Dgn > Dwg

Purpose

Use **FiletoXRef** to convert a MicroStation reference file attachment to an AutoCAD XRef attachment.

Format

FileToXRef

Usage

FileToXRef is only valid when translating from MicroStation to AutoCAD.

Example

FileToXRef

will convert MicroStation reference files to AutoCAD XRef attachments.

FittedText Command

Dgn > Dwg

Purpose

Use **FittedText** to force all text to be placed fitted within the original range box.

Format

FittedText *mode*

Arguments

mode = **Enable** or **Disable**.

If **Enable** is selected, text sizes will be adjusted per entity to insure the text fits within the range of the original.

If **Disable** is selected, text will be placed with the same justification as the original text.

Usage

FittedText is only valid when translating from MicroStation to AutoCAD.

Example

FittedText Enable

will convert text and place the resultant AutoCAD text fitted.

Remarks

By default, fitted text is disabled.

Font Command

Dwg > Dgn

Purpose

Use **Font** to define the MicroStation font number to be assigned to text elements.

Format

Font = *fontno*

Arguments

fontno = the MicroStation font number to be assigned.

The default font assignments are:

<u>AutoCAD</u>	<u>MicroStation</u>
TXT	0
MONOTXT	3
ROMANS	1
ROMANC	2
ROMAND	2
ROMANT	2
ITALICC	23
SCRIPTS	30
SCRIPTC	30

V8 default font numbers for Windows fonts are:

<u>V8 Font</u>	<u>Windows Font</u>
1025	Arial
1029	Times New Roman
1053	Arial Narrow

Usage

Font is only valid when translating from AutoCAD to MicroStation.

Example

Font = 23

will select all text and assign font 23 in the MicroStation file.

FontToShape Command

Dgn > Dwg

Purpose

Use **FontToShape** to convert font character symbols to shape instances.

Format

FontToShape (char to shapefilename,shapename)

Arguments

char = the character to be replaced

shapefilename = the name of the AutoCAD shape file containing the desired shape definitions

shapename = the name of the shape to be substituted for the character being replaced

or

\nnn for the index of the shape in the shape file.

Usage

FontToShape is only valid when translating from MicroStation to AutoCAD.

Example

FontToShape (A to ashapes,arrow)

will replace all A's with the AutoCAD shape named **arrow** found in the shape file **ashapes.shx**.

Remarks

The shape file being referenced must already be loaded into the AutoCAD prototype file prior to translation.

FontToSymbol Command

Dgn <> Dwg

Purpose

Use **FontToSymbol** to convert font character symbols to block or cell instances.

Format

FontToSymbol (*nnn* to *name*)

Arguments

nnn = the octal ASCII code (0-377) for the character to be replaced

name = the name of the block or cell to be substituted

Usage

FontToSymbol is valid for translating both directions.

Example

FontToSymbol (065 to arrow)

will replace all A's (octal **065**) with the AutoCAD block or MicroStation cell named **arrow**.

Remarks

The blocks or cells being referenced must already be defined in the AutoCAD seed file or the MicroStation cell library prior to translation.

ForceAllLayersOn Command

Dgn > Dwg

Purpose

Use **ForceAllLayersOn** to display all layers in the Dwg file even though they were not displayed in the Dgn file.

Format

ForceAllLayersOn

Usage

ForceAllLayersOn is only valid when translating from MicroStation to AutoCAD.

Example

ForceAllLayersOn

will display all layers in the Dwg file

ForceCellLevel Command

Dwg > Dgn

Purpose

Use **ForceCellLevel** to set the cell header type 2 element level and all component entity levels to the level of the DWG insert entity.

Format

ForceCellLevel

Usage

ForceCellLevel is only valid when translating from AutoCAD to MicroStation.

Example

ForceCellLevel

will select all blocks being translated and assign the cell level for its header and all components to be set to the level associated with the insert instance for the block.

Remarks

When **ForceCellLevel** is used, any level settings in the block definition are ignored. Each cell instance will be given level assignments based on each individual insert instance.

ForceType2Cells Command

Dwg > V8 Dgn

Purpose

Use the **ForceType2Cells** command to force the creation of type 2 cells in V8 Dgn files. Normally all cells are created as shared cells.

Format

ForceType2Cells

Usage

ForceMuTag is only valid when translating from AutoCAD to V8 MicroStation.

Example

ForceType2Cells

will force the creation of type 2 in the V8 Dgn file instead of shared cells..

ForceMuTag Command

Dwg > Dgn

Purpose

Use **ForceMuTag** with the **AttDefToText** command. In those cases where the tag value is converted to text and the tag value is **UTAG**, it will be changed to **MUTAG**. This is a special command and is not likely to be of general use.

Format

ForceMuTag

Usage

ForceMuTag is only valid when translating from AutoCAD to MicroStation. It can only be used with the **AttDefToText** command.

Example

AttDefToText, ForceMuTag

will convert attdefs with no default value and a tag value of **UTAG** to a text string of **MUTAG**.

Force3DPolylines Command

Dgn > Dwg

Purpose

Use **Force3DPolylines** to force all polylines to be translated to 3D space polylines even if they are planar.

Format

Force3DPolylines

Usage

Force3DPolylines is only valid for MicroStation to AutoCAD translations.

Example

Force3DPolylines

will convert all relevant MicroStation elements on all levels to 3D polylines.

FractionStack Command

Dgn > Dwg

Purpose

Use **FractionStack** to set the mode for translating single-character stacked fractions.

Format

FractionStack *mode*

Arguments

mode = **Enable** or **Disable**.

If **Enable** is selected, stacked fractions will be retained.

If **Disable** is selected, stacked fractions will be unstacked during translation.

Usage

FractionStack is only valid when translating from MicroStation to AutoCAD.

Example

FractionStack Disable

will select all incoming text elements and unstack any single-character fractions that occur in the text strings.

Remarks

By default, stacked fractions will be unstacked to separate numerator, slash, and denominator characters.

KeepAllReferences Command

Dgn > Dwg

Purpose

Use **KeepAllReferences** when you need to include all reference files regardless of view display setting.

Format

KeepAllReferences

Usage

KeepAllReferences is only valid when translating from MicroStation to AutoCAD.

Remarks

By default, only reference files attached with display on in the view being translated will be included in the translation along with the main file. You can force all reference files to be included with the **KeepAllReferences** command.

Layer Command

Dwg > V8 Dgn, Dgn < Dwg

Purpose

Use **Layer** to assign the output AutoCAD or V8 layer for elements.

Format

Layer = layername

or:

Layer = BYBLOCK

Arguments

layername = AutoCAD or V8 layer name. User variables and/or CATS system variables may be used as a portion of or the complete layer name.

BYBLOCK indicates the layer will be assigned at block placement time.

Usage

Layer is only valid when translating from MicroStation to AutoCAD and AutoCAD to V8 MicroStation.

Example

Layer = Elec

will assign the Layer Elec to all elements.

Remarks

By default, entities will be placed on an AutoCAD layer using the MicroStation level number as the name or the level name if named level have been used.

Level Command**Dwg > Dgn**

Purpose

Use **Level** to assign the output MicroStation level for elements.

Format

Level = *nn*

Arguments

nn = MicroStation level number. For versions below V8, from **1** to **63**.

Usage

Level is only valid when translating from AutoCAD to MicroStation.

Example

level = 5

will select all AutoCAD entities and place them on MicroStation level **5**.

Remarks

Level numbers are assigned sequentially to the AutoCAD layer name table. For versions below V8, if more than 63 layers exist, level numbering restarts at **1** for each group of 63 layers.

Linetype Command

Dgn <> Dwg

Purpose

Use **Linetype** to assign the output linetype to elements satisfying the selection criteria.

Format

Linetype = *linetype*

or:

Linetype = BYLAYER

or:

Linetype = BYBLOCK

Arguments

linetype = AutoCAD linetype name; or MicroStation linetype number. User variables may be used for setting linetype.

BYLAYER indicates the linetype will be set per the layer table. **BYLAYER** is only valid when generating an AutoCAD file.

BYBLOCK indicates the linetype will be assigned at block placement time. **BYBLOCK** is only valid during MicroStation cell to AutoCAD block translation.

<u>MicroStation</u>	<u>Linetypes</u>
0	solid
1	dotted
2	medium dashed
3	long dashed
4	dash dot
5	short dashed
6	dash dot dot
7	long dash short dash

Usage

Linetype is valid for translating both directions.

Example

Linetype = DOT

will assign the AutoCAD linetype **DOT** to incoming MicroStation elements from level **10**.

LowerLeftText Command

Dgn <> Dwg

Purpose

Use **LowerLeftText** to place text using the stored lower left point instead of the calculated alignment point.

Format

LowerLeftText mode

Arguments

mode = **Enable** or **Disable**.

If **Enable** is selected, text will be placed using the stored lower left point.

If **Disable** is selected, text will be placed using the calculated alignment point.

Usage

LowerLeftText is valid for translating both directions.

Example

LowerLeftText

will force text to be placed with lower left justification.

Remarks

This action is useful when text is misplaced. Sometimes the text alignment point is not accurate. In these cases, the **LowerLeftText** command can result in a better text placement.

LwDefault

Dwg > Dgn

Purpose

Use **LwDefault** to define the AutoCAD lineweight to be used when DEFAULT is encountered when translating from AutoCAD to MicroStation.

Format

LwDefault = nn

Arguments

nn = the lineweight value expressed an integer representing hundredths of mm, e.g., 158 represents a lineweight of 1.58 mm

Valid lineweight values for AutoCAD are: 0.00, 0.05, 0.09, 0.13, 0.15, 0.18, 0.20, 0.25, 0.30, 0.35, 0.40, 0.50, 0.53, 0.60, 0.70, 0.80, 0.90, 1.00, 1.06, 1.20, 1.40, 1.58, 2.00, 2.11

Usage

LwDefault is valid for translating both directions.

Example

LwDefault = 120

will set the default lineweight value to 1.20 mm.

Remarks

AutoCAD does not store the default lineweight value in the Dwg. Use this command to supply a lineweight value to be used with a lineweight of DEFAULT is encountered.

MakeAnonymousBlock Command

Dgn > Dwg

Purpose

Use **MakeAnonymousBlock** to convert each instance of a MicroStation type 2 cell into an anonymous block in AutoCAD.

Format

MakeAnonymousBlock

Usage

MakeAnonymousBlock is only valid when translating from MicroStation to AutoCAD.

Example

MakeAnonymousBlock

will convert all cells into anonymous blocks in AutoCAD.

Remarks

This action is useful when translating files created by MicroStation applications which create new cells for each piece of equipment but do not assign unique names for each different graphics representation. Only type 2 cells can be converted to anonymous blocks; shared cell definitions and instances cannot.

MakeLinkage Command

V7 Dgn > Dwg

Purpose

Use **MakeLinkage** to create a special DGI linkage set in AutoCAD for each MicroStation database linkage encountered.

Format

MakeLinkage

Usage

MakeLinkage is only valid when translating from MicroStation to AutoCAD. The DGI linkage set is added to the entity as extended data with the following format:

<u>Group Code</u>	<u>Description</u>
1001	Application name. This field will always be the string DGIDATA. Other applications will use different names.
1002	Begin descriptor set. This field will always contain the string {.
1000	Data type. This field will always be the string DATABASETYPE= <i>aaa</i> , where <i>aaa</i> is the database type name: Informix, XBase, RIS, Oracle, Ingress, Sybase, or ODBC.
1000	Data type. This field will always be the string DGILINKAGE= <i>nnn</i> , where <i>nnn</i> is the database entity (table) type. Future DGI applications may use different codes.
1000	Data type. This field will always be the string MSLINK= <i>nnn</i> , where <i>nnn</i> is the linkage key. Future DGI applications may use different codes.
1002	End descriptor set. This field will always contain the string }

Additional descriptor sets of the form **{entity,key}** will be present for each additional database linkage attached to the graphic element.

Example

MakeLinkage

will generate linkages for all elements with database linkages in the graphic file.

Example DXF line with linkage -

```

0
LINE
8
PIPELAYER
10
1000.0
20
1000.0
11
2000.0
21
3000.0

```

```
1001
DGIDATA
1002
{
1000
DATABASETYPE=Xbase
1000
DGILINKAGE=120
1000
MSLINK=7756834
1002
}
```

Remarks

This action is useful for converting MicroStation files with database linkages to be used with AutoCAD applications designed to access the DGI linkage set information.

MakeUserBlock Command

Dgn > Dwg

Purpose

Use **MakeUserBlock** to create new block definition and insert for each occurrence of a type 2 cell. The block name is the same as the cell name plus a unique id number.

Format

MakeUserBlock

Usage

MakeUserBlock is only valid when translating from MicroStation to AutoCAD.

Example

MakeUserBlock

Remarks

This command is useful for converting Dgns made with MicroStation application packages such as Architecture which create pseudo cells for doors, etc.

NoGlobalOriginAdjustment Command

Dwg > V7 Dgn

Purpose

Use **NoGlobalOriginAdjustment** to bypass the automatic global origin adjustment feature for Dgn files.

Format

NoGlobalOriginAdjustment

Usage

NoGlobalOriginAdjustment is only valid when translating from AutoCAD to MicroStation.

Example

NoGlobalOriginAdjustment

Remarks

MicroStation's global origin is normally automatically adjusted when the coordinate range of the Dwg will not fit the Dgn. This command is useful for cases with multiple xrefs which don't all fit in a single coordinate range setup.

Offset Command

Dgn <> Dwg

Purpose

Use **Offset** to define a drawing offset to be added to all element coordinates.

Format

Offset = (xoffset,yoffset,zoffset)

Arguments

xoffset = the X coordinate offset in master units.

yoffset = the Y coordinate offset in master units.

zoffset = the Z coordinate offset in master units.

Usage

Offset is valid for translating both directions.

Example

offset = (10000,20000,10000)

will offset all resulting coordinates by **10000** in X, **20000** in Y, and **10000** in Z.

Remarks

MicroStation's global origin will automatically be taken into account during translation. It will be extracted from the DGN file when translating to AutoCAD or from the seed file being used when translating to MicroStation. If offset is defined, it will be used instead of the global origin values.

NoXclip**Dgn > Dwg**

Purpose

Use **NoXclip** to ignore clipping for reference files translated to Xrefs.

Format

NoXclip

Usage

NoXclip is only valid for translating from MicroStation to AutoCAD.

Example

NoXclip

will create xrefs in the output Dwg without xclipping.

PlaceEndCap

V7 Dgn > Dwg

Purpose

Use **PlaceEndCap** to put circular endcaps on line, linestring, or arc entities with width. Used primarily for Printed Circuit Board applications.

Format

PlaceEndCap

Usage

PlaceEndCap is only valid for MicroStation to AutoCAD translations.

Example

PlaceEndCap

will add circular endcaps to linear entities.

PLineToShape Command

Dwg > Dgn

Purpose

Use **PLineToShape** to create filled shapes for polylines with width.

Format

PLineToShape

Usage

PLineToShape is only valid for translating from AutoCAD to MicroStation.

Example

If Layer = * then PLineToShape endif

will create filled shapes for all polylines with width.

PointCell Command

Dwg <> Dgn

Purpose

Use **PointCell** to define cells or blocks point cells regardless of how they were defined in AutoCAD or MicroStation. A point cell is a cell or block which takes the active level/layer, color, and linetype when it is placed.

Format**PointCell****Usage**

PointCell is valid for translating both directions.

Example**PointCell**

will translate all AutoCAD blocks or MicroStation cells as point cells regardless of how they were defined.

Remarks

By default, only those blocks defined on AutoCAD layer 0 with ByBlock layer, color, and linetype will be translated to point cells. In MicroStation, only cells with the point cell type will be translated as point cells.

Polyline Command

Dgn > Dwg

Purpose

Use **Polyline** to control AutoCAD polyline generation.

Format

Polyline mode

Arguments

mode = **Enable** or **Disable**.

If **Enable** is selected, polylines will be generated as appropriate for the incoming MicroStation element.

If **Disable** is selected, no polylines will be generated; strings will be dropped to lines, and polyarcs will be dropped to arcs.

Usage

Polyline is only valid when translating from MicroStation to AutoCAD.

Example**Polyline Disable**

will not allow any polyline entities.

Remarks

By default, AutoCAD polylines will be enabled and generated.

ProcessLeftOverTags

Dgn > Dwg

Purpose

Use **ProcessLeftOverTags** to place unmatched Tags as regular text.

Format

ProcessLeftOverTags

Usage

ProcessLeftOverTags is only valid when translating from MicroStation to AutoCAD.

Example

ProcessLeftOverTags

Remarks

MicroStation Tags are normally associated with specific cells. By default, these tags are converted to AutoCAD attributes definitions (ATTDEF) in the block and attributes in the block insert. Unmatched tags are ignored. **ProcessLeftOverTags** is used to place any unmatched tags as regular text in the AutoCAD Dwg.

ReferenceLevelSymbology

Dwg > V7 Dgn

Purpose

Use **ReferenceLevelSymbology** to allow xrefs to be converted to reference file using level symbology.

Format

ReferenceLevelSymbology

Usage

ReferenceLevelSymbology is only valid when translating from AutoCAD to V7 MicroStation. It is used with the FileToRef command.

Example

ReferenceLevelSymbology

ScaleCell Command

Dgn <> Dwg

Purpose

Use **ScaleCell** to specify a scale factor for Dgn cells and Dwg blocks and hatches. The scale value will be multiplied by the original scale factor to compute the new scale.

Format

ScaleCell = value

or:

ScaleCell =\$

Arguments

value = the scale factor to be applied to the cell or block insert.

\$ = the scale factor is the scale factor of the current reference file.

Usage

ScaleCell is valid for translating both directions.

Example

ScaleCell=10

will scale all cells up by a factor of 10.

Set Command

Dgn <> Dwg

Purpose

Use **Set** to define variables to use in string values such as layer names, block names, etc.

Format

Set (variable = value)

Arguments

variable = user-defined variable name of your choosing. When later referenced in another command, you must specify the variable name between percent signs, i.e. *%name%*.

value = the character string you wish to assign to variable for later use.

Usage

Set is valid for translating both directions.

Example

Set (prefix=abc)

will set the variable **prefix** to **abc** for later use.

SkipFile Command

Dgn <> Dwg

Purpose

Use **SkipFile** when you want to omit from translation the remaining elements in the current file.

Format

SkipFile

Usage

SkipFile is valid for translating both directions.

Remarks

The currently open drawing file is closed and the next file in the list is processed.

SkipPoint Command

Dgn <> Dwg

Purpose

Use **SkipPoint** to control generation of point (zero-length line) entities.

Format

SkipPoint *mode*

Arguments

mode = **Enable** or **Disable**.

If **Enable** is selected, no point elements will be generated.

If **Disable** is selected, points will be generated in the output file.

Usage

SkipPoint is valid for translating both directions.

Example

SkipPoint Enable

will disallow generation of point or zero-length line elements.

Remarks

By default, point entities will be generated in the output file.

SymbolFont Command

Dgn <> Dwg

Purpose

Use **SymbolFont** to control symbol character placement.

Format

SymbolFont *mode*

Arguments

mode = **Enable** or **Disable**.

If **Enable** is selected, letter spacing will not be considered when calculating placement coordinates for the symbol character.

If **Disable** is selected (the default mode), the character(s) will be treated as ordinary text for calculation of placement coordinates.

Usage

SymbolFont is valid for translating both directions.

Example

SymbolFont enable

will result in character placement coordinates calculated without consideration for letter spacing.

TextHeightFactor Command

Dgn <> Dwg

Purpose

Use **TextHeightFactor** to apply a scale factor to the character height of text elements.

Format

TextHeightFactor = *nn*

Arguments

nn = the decimal multiplier to be applied to the height of all text meeting the selection criteria.

Usage

TextHeightFactor is valid for translating both directions.

Example

TextHeightFactor = .6667

will adjust characters coming in to 2/3 of their original height and width.

Remarks

By default, character height will not be scaled. This feature is primarily useful when converting between fonts of different sizes.

TextObliqueAngle Command

Dgn > Dwg

Purpose

Use **TextObliqueAngle** to define an AutoCAD obliqueing angle to be assigned to text entities.

Format

TextObliqueAngle = *nn*

Arguments

nn = the decimal obliqueing angle to apply to text placed in AutoCAD.

Usage

TextObliqueAngle is only valid when translating from MicroStation to AutoCAD.

Example

TextObliqueAngle = 10

will apply an obliqueing angle of **10** degrees to all AutoCAD text.

TextToLine Command

Dgn > Dwg

Purpose

Use **TextToLine** to replace a selected text string with a line.element.

Format

TextToLine

Usage

TextToLine is valid for translating from MicroStion to AutoCAD.

Example

```
If font=120 Then TextToLine Endif
```

```
If font=120 TextLine='--Hh--' Then Linetype=dashed Endif
```

place to a line and assign a linetype.

Remarks

This command is useful when replacing a line style created using a special font in MicroStation.

TextStyle Command

Dgn > Dwg

Purpose

Use **TextStyle** to define the AutoCAD text style name to be assigned to text entities which satisfy the selection criteria.

Format

TextStyle = textstylename

Arguments

textstylename = the AutoCAD text style name 1 to 31 characters in length to be assigned. User variables and/or CATS system variables may be used as a portion of or the complete AutoCAD text style name.

The default textstyle assignments are:

<u>AutoCAD</u>	<u>MicroStation</u>
TXT	0
MONOTXT	3
ROMANS	1
ROMANC	2
ROMAND	2
ROMANT	2
ITALICC	23
SCRIPTS	30
SCRIPTC	30

Usage

TextStyle is only valid when translating from MicroStation to AutoCAD.

Example

TextStyle = SPECIAL16

will assign AutoCAD text style **SPECIAL16** to text elements.

Remarks

By default, if a specific text style is not assigned, text entities will be written referencing AutoCAD default text style **STANDARD**.

TextWidthFactor Command

Dgn <> Dwg

Purpose

Use **TextWidthFactor** to apply a scale factor to the character width of text elements.

Format

TextWidthFactor = *nn*

Arguments

nn = the decimal multiplier to be applied to the width of all text meeting the selection criteria.

Usage

TextWidthFactor is valid for translating both directions.

Example

TextWidthFactor = .95

will apply an additional width factor of **.95** to text.

Remarks

By default, character width will not be scaled. This feature is useful when converting text with variable width fonts.

UseExistingEnterData Command

Dwg > V7 Dgn

Purpose

Use **UseExistingEnterData** to use the existing parameters for enter data text (size, position, rotation, field size, etc) from an existing cell library rather than recalculating the text parameters based on the DWG attribute data. The attribute value will be placed in the enter data field according to the field justification for the enter data field. The DWG attributes must occur in the same order as the enter data text in the cell and must be the same number of characters or fewer.

Format

UseExistingEnterData

Usage

UseExistingEnterData is only valid for translating from AutoCAD to V7 MicroStation.

Example

If Block = * then UseExistingEnterData endif

will use existing enter data text for all blocks being translated.

UseGlobalOriginForInsbse

V7 Dgn > Dwg

Purpose

Use **UseGlobalOriginForInsbse** to set the insbase in the Dwg using the Dgn global origin. Xref placement are also adjusted to match.

Format

UseGlobalOriginForInsbse

Usage

UseGlobalOriginForInsbse is only valid for translating from V7 MicroStation to AutoCAD.

Example

UseGlobalOriginForInsbse

will set the Dwg insbase using the Dgn global origin.

Weight Command

Dwg <> Dgn

Purpose

Use **Weight** to set the MicroStation weight or AutoCAD 2000++ lineweight for all elements satisfying the selection criteria.

Format

Weight = nn

or

Weight = Default

or

Weight = Bylayer

or

Weight = Byblock

Arguments

nn = MicroStation weight to be assigned. Weight must be an integer ranging from **0** to **31**. For AutoCAD 2000++, lineweight is a integer representing hundredths of a mm, e.g., a lineweight of 5 represents .05 mm.

DEFAULT will get the weight from the **LWDEFAULT** value in AutoCAD.

BYLAYER indicates the lineweight will be set per the layer table. **BYLAYER** is only valid when generating an AutoCAD file.

BYBLOCK indicates the lineweight will be assigned at block placement time. **BYBLOCK** is only valid during MicroStation cell to AutoCAD block translation.

Valid lineweight values for AutoCAD are: 0.00, 0.05, 0.09, 0.13, 0.15, 0.18, 0.20, 0.25, 0.30, 0.35, 0.40, 0.50, 0.53, 0.60, 0.70, 0.80, 0.90, 1.00, 1.06, 1.20, 1.40, 1.58, 2.00, 2.11

Usage

Weight is valid for translating both directions.

Example

Weight = 3

will set weight to **3** in MicroStation.

Remarks

If not specifically assigned, MicroStation elements will be created with weight set to **0**; AutoCAD lineweights will be set to Default.

Width Command

Dgn > Dwg

Purpose

Use **Width** to set the AutoCAD polyline width for polylines.

Format

Width = *nn*

or:

Width = \$ReferenceScaleFactor

Arguments

nn = decimal polyline width to be assigned.

ReferenceScaleFactor = this value will be multiplied by the current reference file scale; the product will be written out as the polyline width.

Usage

Width is only valid when translating from MicroStation to AutoCAD.

Example

Width = 1.4

will set the polyline width to **1.4**.

Remarks

By default, width will not be defined. For any polylines generated, the width will be **0.0**.

3DElevation Command

Dgn > Dwg

Purpose

Use **3DElevation** to control translation of 3D Z coordinates.

Format

3DElevation *mode*

Arguments

mode = **Enable** or **Disable**.

If **Enable** is selected, group 38 elevation data will be generated.

If **Disable** is selected, regular 3D Z coordinates will be generated.

Usage

3DElevation is only valid when translating from MicroStation to AutoCAD DXF.

Example

3DElevation Enable

will convert all 3D data to group 38 elevation in AutoCAD instead of group 30 Z coordinates.

Remarks

By default, regular 3D coordinates (group 30) will be generated.

SkipReferences Command

V7 Dgn > Dwg

Purpose

Use **SkipReferences** when you need to omit all reference files.

Format

SkipReferences

Usage

SkipReferences is only valid when translating from V7 MicroStation to AutoCAD.

Remarks

By default, all reference files attached with display on in the view being translated will be merged into a single output file along with the main file. You can conditionally omit reference files with the SkipFile command.

IncludeElement Command

Dgn <> Dwg

Purpose

Use **IncludeElement** when you want to translate elements which have been previously excluded.

Format

IncludeElement

Usage

IncludeElement is valid for translating both directions.

Example

```
If Level = 10 Then ExcludeElement Endif  
If Level = 10, Color = 3 Then IncludeElement Endif
```

will exclude from translation all elements on level **10** except for those elements that are on level **10** and have color set to **3**.

Remarks

By default, all elements will be included in the translation output unless specifically excluded.

Rules

Rules choose translation options based on the characteristics of each entity as it is translated. A command entered on a line by itself applies to all entities. If you want to use a command or set of commands for only certain entities in the file being translated, then use a rule to tell the translator which entities the commands apply to.

Rules can be defined based on cell or block name, color, class, entity type, graphic group, text justification or font, level, layer, linetype, and weight — singly or in combination. All numeric selections can use lists and/or ranges. All string selections can use lists and/or DOS wildcards. The use of string variables is also provided for. There are some pre-defined system variables available and you can also define your own. Specific descriptions and examples are included in this section.

Rules are free form. One or more spaces or tabs or commas separate each part of a rule for the rest. Rules can be contained on a single line or continued on several lines for clarity. Blank lines can be used for additional clarity. Place a ; in the first position of a line for comments. Rules can be entered in upper or lower case or a mixture.

Note:

Rules apply to all elements, including elements within a cell definition. If the following rule selection is entered:

If Cell = NotCell Then ... Endif

the rule will not apply to elements in a cell definition. Enter data text placed as an AutoCAD attribute is considered part of the cell placement.

Rules Format

If *selections* Then *commands* Endif

selection are any 1 or combination of 2 or more of the following criteria. Spaces, tabs, or commas separate selections.

commands are any one or combination of two or more of commands. Spaces, tabs, or commas separate commands.

For a set of actions to occur, all of the specified rule criteria must be satisfied. A blank or a comma separate multiple rules or actions.

Selections

Block	Justification	Width
Cell	Layer	
Class	LayerFrozen	TagName
Color	LayerOff	TextLine
CustomLinetype	Level	
EntityType	Linetype	
FileDirectory	OutLayer	
FileExt	ReferenceLogical	
FileName_	ReferenceScale	
FileType	Symbol	
Font	TextStyle	
GraphicGroup	Units	
Hatch	Weight	

Commands

ArcStrokes	DeleteASE-LPN	Level	TextHeightFactor
AddToModelspace	EditText	Linetype	TextObliqueAngle
AddToPaperspace	EnterData	LowerLeftText	TextStyle
AttachWithSavedPath	ExcludeElement	LwDefault	TextWidthFactor
ArrtributeTag	ExitRules	MakeAnonymousBlock	UseExistingEnterdata
AttributeToTag	ExplodeCell	MakeLinkage	UseGlobalOriginForInsbse
AttDefToText	FileBlockName	NoGlobalOriginAdjustment	Weight
BCurveStrokes	FileToBlock	NoXclip	Width
BindXref	FileToRef	Offset	3DElevation
Block	FileToXRef	PlaceEndCaps	
Cell	FittedText	PlineToShape	AttributeName
Color	Font	PointCell	ForceType2Cells
ConversionFactor	FontToShape	Polyline	ForceAllLayersOn
CurveStrokes	FontToSymbol	ProcessLeftOverTags	MakeUserBlock
CustomLinetype	ForceCellLevel	ReferenceLevelSymbology	TextToLine
CustomLinetypeScale	ForceMuTag	ScaleCell	
DefineHeader	Force3DPolylines	Set	
DefineLayers	FractionStack	SkipFile	
DefineStyles	IncludeElement	SkipPoint	
DefineDxfHeader	KeepAllReferences	SkipReferences	
DefineDxfLayer	Layer	SymbolFont	

Wildcards in Rule Selections

All string selections can use lists and/or DOS wildcards. If you need to specify a list of names, they must be placed within parentheses and separated by commas, i.e.

(BDR,S01,myname)

You can use DOS wildcards * and ? to match a group or a single character. The * matches all following characters. The ? matches a single character. For example, the following are valid selections using wildcards.

(BDR*, S??T, HVAC*, Z?C)

Lists and Ranges in Rule Selections

All numeric selections can use lists and/or ranges. If you need to specify a list of numbers, they must be placed within parentheses and separated by commas, i.e.

(1,3,10)

To select a range of values, use a hyphen, i.e.

1-10

Lists and ranges can be combined as follows:

(1,3,5-10,25)

Using Rules

Control files usually have many commands and rules. Each entity in the input file is examined to see if the commands and rules apply. The commands and rules are applied one by one from the top down. Every command and rule is used for every entity. Several rules may apply to a particular entity and the final result will be the combination of all commands applied. Thus it is a good idea put your more general rules first followed more specific rules.

Example

If Layer=Wall Then Level=2 Endif

If Layer=Wall LineType=Dashed Then Level 3 Endif

In this example, a line on layer Wall and linetype Continuous would be assigned to level 2. A line on layer Wall and linetype Dashed would be assigned to level 3. If the rules were reversed -

If Layer=Wall LineType=Dashed Then Level 3 Endif

If Layer=Wall Then Level=2 Endif

both lines would match the second rule and be assigned to level 2.

Selections always check the characteristics of the original entity. If you have a rule which changes the color of an entity. A following rule such as:

If Color=3 Then Color=4 Endif

will check the original color, not the color as changed by a previous rule.

Block Selection

Dwg > Dgn

Purpose

Use **Block** when you need to select data based on the full or partial AutoCAD block name.

Format

If Block = *blockname* Then.....Endif

Arguments

blockname = AutoCAD block name. Lists and wildcards are supported.

Usage

Block is only valid when translating from AutoCAD to MicroStation.

Example

If Block = E* Then EnterData Enable Endif

will select all incoming AutoCAD blocks beginning with the letter **E** and convert any attributes attached to MicroStation enter data text.

Remarks

Hatches are a special case of a block. To select, specify **_X*** as the block name to select all hatches in a file.

Cell Selection

Dgn > Dwg

Purpose

Use **Cell** when you need to handle data differently based on whether or not is part of a cell; or, if you have requirements based on the full or partial cell name.

Format

If cell = *cellname* Then.....Endif

or:

If cell = not_ cell Then.....Endif

Arguments

cellname = MicroStation cell name. Lists and wildcards are supported.

If **not_cell** is specified, actions will apply only to elements not part of a cell.

Usage

Cell is only valid when translating from MicroStation to AutoCAD.

Example

If cell = (bdr,s01,hvac*,z?c)Then.....Endif

will affect cells named **BDR, S01**, all cells beginning with **HVAC** and all cells with 3-letter names of which the first is **Z** and the last **C**.

```
If cell = not_cell,  
  GraphicGroup = 12  
then  
  layer= 28  
endif
```

will select all elements with graphic group code **12** that are not part of a cell and place them on AutoCAD layer **28**.

Class Selection

Dgn > Dwg

Purpose

Use **Class** when you need to perform specific actions based on an incoming MicroStation element's class.

Format

If Class = *classno* Then.....Endif

Arguments

classno = MicroStation class number from the following table. Lists and ranges are supported.

MicroStation Classes

- 0 = primary element
- 1 = pattern component
- 2 = construction element
- 3 = dimension element
- 4 = primary rule element
- 5 = linear patterned element
- 6 = construction rule element

Usage

Class is only valid when translating from MicroStation to AutoCAD.

Example

**If class = 2 level= 34
then color = 16 endif**

will select all incoming construction (class **2**) elements on MicroStation level **34** and assign them AutoCAD color **16** on output.

Remarks

MicroStation provides special handling for elements based on class.

Color Selection

Dgn <> Dwg

Purpose

Use **Color** when you need to perform specific actions based on an incoming element's color.

Format

If Color = *colorno* Then.....Endif

Arguments

colorno = color number. Lists and ranges are supported.

Usage

Color is valid for translating both directions.

Example

```
If color = 2 then  
  set (prefix = abc)  
endif
```

will set the variable **prefix** to **abc** for later use when the incoming element's color is **2**.

CustomLinetype Selection

Dgn > Dwg

Purpose

Use the **CustomLinetype** rule when you have MicroStation custom linetypes that need to be used as selection criteria for AutoCAD assignments.

Format

If CustomLinetype = *mscustom* Then.....Endif

Arguments

mscustom = the name of the MicroStation custom linetype.

Usage

The **CustomLinetype** rule is only valid for MicroStation to AutoCAD translations.

Example

If CustomLinetype = DASHED Then Linetype = BOUNDARY Endif

will select all MicroStation elements with the custom linetype **DASHED** and translate to AutoCAD referencing the **BOUNDARY** linetype.

Remarks

If the custom linetype name contains parentheses () or blanks, you must enclose the name in single quotes , e.g.,

If CustomLinetype = '(Border)' Then Linetype = BORDER Endif

Note:

The actual linetype definitions are not translated by CATS. This command provides a means for you to assign the appropriate AutoCAD linetype based on the MicroStation custom linetype name.

EntityType Selection

Dgn <> Dwg

Purpose

Use **EntityType** when you need to perform specific actions for different incoming element types.

Format

If EntityType = entno Then.....Endif

Arguments

entno = entity type from the following table corresponding to your input file type. Lists and ranges are supported.

AutoCAD Entities

1 = line	14 = insert	201 = hatch
2 = point	15 = attribute definition	202 = lwpline
3 = circle	16 = attribute	208 = proxy circle
4 = shape	19 = polyline	209 = proxy 3pt circle
5 = ellipse	21 = 3Dline	210 = proxy arc
6 = spline	22 = 3Dface	212 = proxy pline
7 = text	23 = dimension	213 = proxy polygon
8 = arc	25 = solid3d	216 = proxy text
9 = trace	28 = mtext	217 = proxy text2
11 = solid	29 = leader	
12 = block	31 = mline	

MicroStation Elements

1 = cell definition	15 = ellipse	
2 = cell instance		16 = arc
3 = line		17 = text
4 = line string		21 = BSspline pole
6 = closed shape	26 = BSpline knot	
7 = text node		28 = BSpline weight factor
11 = curve string	33 = dimension	
12 = complex chain		34 = shared cell definition
14 = complex shape		35 = shared cell instance

Usage

EntityType is valid for translating both directions.

Example

```
If EntityType = 1Then
  ExcludeElement
Endif
```

will select all incoming AutoCAD lines (type **1**) and omit them from the generated MicroStation file.

FileDirectory Selection

Dgn <> Dwg

Purpose

Use **FileDirectory** when you need to perform specific actions on all the files in selected directories.

Format

If FileDirectory = *directory* Then.....Endif

Arguments

directory = DOS directory where selected files reside. Lists and wildcards are supported.

Usage

FileDirectory is valid for translating both directions.

Example

```
If FileDirectory = \PROJECTA\ Then  
  Font = 23  
Endif
```

will assign font **23** to MicroStation text coming from AutoCAD files in directory **\PROJECTA**.

FileExt Selection

Dgn <> Dwg

Purpose

Use **FileExt** when you need to perform specific actions on groups of files having the same full or partial extension.

Format

If FileExt = ext Then.....Endif

Arguments

ext = the DOS file extension to be selected. Lists and wildcards are supported.

Usage

FileExt is only valid for translating from MicroStation to AutoCAD. This is because AutoCAD only supports the fixed extensions DWG and DXF.

Example

```
If
  FileExt = BDR,
  ReferenceScale = 2
Then
  ConversionFactor = 0.5
Endif
```

will select all reference files with the file extension **BDR** and attached with a scale of **2**. The elements within these reference files will be scaled by **.5** during translation.

FileName Selection

Dgn <> Dwg

Purpose

Use **FileName** when you need to perform specific actions on all files having the same full or partial name.

Format

```
If FileName = filename Then.....Endif
```

Arguments

filename = the DOS file name to be selected. Lists and wildcards are supported.

Usage

FileName is valid for translating both directions.

Example

```
If FileName = MYNAME Then  
  FiletoBlock  
  FileBlockName = FILEA  
Endif
```

will place the data from files named **MYNAME** in an AutoCAD block named **FILEA**.

Remarks

FileName only refers to the name portion of the file spec. Extension and directory are handled by FileExt and FileDirectory , respectively.

FileType Selection

Dgn > Dwg

Purpose

Use **FileType** to define one set of actions for the main design file and a different set of actions for all attached reference files.

Format

If FileType = DGN Then.....Endif

or:

If FileType = REF Then.....Endif

Usage

FileType is only valid for MicroStation to AutoCAD translations.

Example

If FileType = REF Then SkipFile Endif

will not translate any reference files.

Remarks

If file type is **DGN**, the resultant actions will apply to the main drawing only. If file type is **REF**, the resultant actions will apply to all attached reference files.

Font Selection

Dgn <> Dwg

Purpose

Use **Font** when you need to perform specific actions based on an incoming text element's font name or number.

Format

If Font = *font* Then.....Endif

Arguments

font = AutoCAD font (shape file) name or MicroStation font number. Lists and ranges are supported.

Usage

Font is valid for translating both directions.

Example

```
If Font = (0-127) Then  
  TextStyle = ROMANS  
Endif
```

will translate all MicroStation text to AutoCAD **ROMANS**; or:

```
If Font = ROMANS Then Font = 2 Endif
```

will translate all AutoCAD text referencing shape file **ROMANS** to MicroStation text with font **2**.

GraphicGroup Selection

V7 Dgn > Dwg

Purpose

Use **GraphicGroup** when you need to perform specific actions based on an incoming MicroStation element's graphic group number.

Format

If GraphicGroup = *ggno* Then.....Endif

Arguments

ggno = MicroStation graphic group number. Lists and ranges are supported.

Usage

GraphicGroup is only valid for MicroStation to AutoCAD translations.

Example

```
if
  graphicgroup = (4-12)
  weight = 3
Then
  color = 4
endif
```

will select all incoming elements having a graphic group code between **4** and **12** and weight set to **3**. These elements will be written out with color **4**.

Hatch

Dwg > Dgn

Purpose

Use **Hatch** when you need to perform specific actions based on an incoming Hatch name.

Format

If Hatch = hatchname Then.....Endif

Arguments

hatchname = AutoCAD hatch name. . Lists and wildcards are supported.

Usage

Hatch is only valid for AutoCAD to MicroStation translations.

Example

If Hatch = AR-CONC Then ScaleCell = .5 Endif

will scale all AR-CONC hatches by .5.

Justification Selection

Dgn <> Dwg

Purpose

Use **Justification** when you need to perform specific actions based on a text element's incoming justification code.

Format

If Justification = *justno* Then.....Endif

Arguments

justno = text justification code number from the table below. Lists and ranges are supported.

<u>MicroStation</u>		<u>AutoCAD</u>
0	left top	30
1	left center	20
2	left bottom	0
6	center top	31
7	center center	21
8	center bottom	1
12	right top	32
13	right center	22
14	right bottom	2

Usage

Justification is valid for translating both directions.

Example

```
If Justification = 5 Then
  TextHeightFactor = 2.0
Endif
```

will double the height of all text with justification code 5.

Layer Selection

V8 Dgn > Dwg, Dwg > Dgn

Purpose

Use **Layer** when you need to perform specific actions based on an incoming AutoCAD or V8 MicroStation entity's full or partial layer name.

Format

If Layer = *layername* Then.....Endif

Arguments

layername = AutoCAD or V8 layer name. Lists and wildcards are supported.

Usage

Layer is only valid when translating from AutoCAD to MicroStation or V8 MicroStation to AutoCAD.

Example

If Layer = BDR* Then ExcludeElement Endif

will omit data on layers beginning with **BDR** from the output file.

LayerFrozen Selection

Dwg > Dgn

Purpose

Use **LayerFrozen** when you need to perform specific actions when the AutoCAD entity's layer is frozen.

Format

If LayerFrozen Then.....Endif

Usage

LayerFrozen is only valid when translating from AutoCAD to MicroStation.

Example

If LayerFrozen Then ExcludeElement Endif

will omit data on AutoCAD layers that are frozen.

LayerOff Selection

Dwg > Dgn

Purpose

Use **LayerOff** when you need to perform specific actions when the AutoCAD entity's layer is turned off.

Format

If LayerOff Then.....Endif

Usage

LayerOff is only valid when translating from AutoCAD to MicroStation.

Example

If LayerOff Then ExcludeElement Endif

will omit data on AutoCAD layers that are turned off.

Level Selection

Dgn > Dwg

Purpose

Use **Level** when you need to perform specific actions based on an incoming MicroStation element's level number.

Format

```
If Level = levelno Then.....Endif
```

Arguments

levelno = MicroStation level number. For versions below V8, from 1 to 63. Lists and ranges are supported.

Usage

Level is only valid when translating from MicroStation to AutoCAD.

Example

```
If Level = 13 Then  
    Color = BYLAYER  
Endif
```

will select all incoming MicroStation elements on level **13**. They will be written to the AutoCAD file with color defined BYLAYER.

Linetype Selection

Dgn <> Dwg

Purpose

Use **Linetype** when you need to perform specific actions based on an incoming element's linetype.

Format

If Linetype = *linetype* Then.....Endif

Arguments

linetype = AutoCAD linetype name 1 to 10 characters in length or MicroStation linetype number between **0** and **7**. Lists, ranges, and wildcards are supported.

MicroStation Linetypes

0	solid
1	dotted
2	medium dashed
3	long dashed
4	dash dot
5	short dashed
6	dash dot dot
7	long dash short dash

Usage

Linetype is valid for translating both directions.

Example

```
If Linetype = 0 Then  
  Set (LT = CONTINUOUS)  
Endif
```

will set the user variable **%LT%** to **CONTINUOUS** when the current element linetype is **0**.

OutLayer Selection

Dgn <> Dwg

Purpose

Use **OutLayer** to match the layer previously assigned to an entity. Other selections, layer, color, etc., check the attributes on the original entity. This selection checks the layer as assigned by all previous rules.

Format

If OutLayer = *layername* Then.....Endif

Arguments

layername = an AutoCAD layer name. Lists and wildcards are supported.

Usage

OutLayer is valid for translating both directions.

Example

If OutLayer = RoomText Then Color=1 Endif

will set the color to 1 for all layer assigned to the RoomText layer.

Remarks

Other selections, like layer, color, etc., check the attributes of the original entity. This selection checks the value of layer as currently assigned by the previous rules.

ReferenceLogical Selection

Dgn > Dwg

Purpose

Use **ReferenceLogical** to select a group of MicroStation reference files based on either the full or partial logical name.

Format

If ReferenceLogical = *logname* Then.....Endif

Arguments

logname = reference file logical name as attached. Lists and wildcards are supported.

Usage

ReferenceLogical is only valid when translating from MicroStation to AutoCAD.

Example

If ReferenceLogical = REF1 Then SkipFile Endif

will omit from translation reference files attached with the logical name **REF1**.

Remarks

When reference files are attached in MicroStation, a logical name is assigned and stored as part of the attachment.

ReferenceScale Selection

Dgn > Dwg

Purpose

Use **ReferenceScale** when you need to perform specific actions to all reference files attached at a particular scale.

Format

If ReferenceScale = *scaleno* Then.....Endif

Arguments

scaleno = a decimal value representing the scale factor applied to the reference file. Lists and ranges are supported.

Usage

ReferenceScale is only valid when translating from MicroStation to AutoCAD.

Example

**If ReferenceScale = 5 Then
 ConversionFactor = 10 Endif**

will select all data in reference files attached with a scale of **5**. During translation, the elements will be scaled by the conversion factor of **10**.

Symbol Selection

Dgn <> Dwg

Purpose

Use **Symbol** when you need to perform specific actions to all symbols (blocks or cells) matching either a full or partial name.

Format

If Symbol = *symname* Then.....Endif

Arguments

symname = AutoCAD block name or MicroStation cell name. Lists and wildcards are supported. For versions below V8, MicroStation cell names support up to 6 characters.

Usage

Symbol is valid for translating both directions.

Example

```
If Symbol = SCL* Then  
  Layer = 100  
  Color = BYLAYER  
Endif
```

will place all cells beginning with **SCL** on AutoCAD layer **100** with color assigned BYLAYER.

TagName Selection

Dgn > Dwg

Purpose

Use **TagName** when you need to identify a specific tag name value.

Format

If Tagname = *name* Then.....Endif

Arguments

name = MicroStation tag name.

Usage

TagName is valid for translating from MicroStation to AutoCAD.

Example

```
If TagName = Valve Then  
  AttributeName = AValve  
Endif
```

will create AutoCAD attributes with name AValve for MicroStation tags named Valve.

TextLine Selection

Dgn > Dwg

Purpose

Use **TextLine** to select a MicroStation text string to be replaced with a line entity.

Format

```
If TextLine='string' Then.....Endif
```

Arguments

'*string*' = is a text string in quotes.

Usage

TextLine is valid for translating from MicroStation to AutoCAD.

Example

```
If font=120 Then TextToLine Endif
```

```
If font=120 TextLine='--Hh--' Then Linetype=dashed Endif
```

place to a line and assign a linetype.

Remarks

This command is useful when replacing a line style created using a special font in MicroStation.

TextStyle Selection

Dwg > Dgn

Purpose

Use **TextStyle** when you need to perform specific actions based on an incoming AutoCAD text entity's style.

Format

If TextStyle = *stylename* Then.....Endif

Arguments

stylename = AutoCAD text style name. Lists and wildcards are supported.

<u>AutoCAD</u>	<u>MicroStation</u>
TXT	0
MONOTXT	3
ROMANS	1
ROMANC	2
ROMAND	2
ROMANT	2
ITALICC	23
SCRIPTS	30
SCRIPTC	30

Usage

TextStyle is only valid when translating from AutoCAD to MicroStation.

Example

```
If TextStyle = SPECIAL Then
  TextWidthFactor = 1.4
Endif
```

will select all text defined with AutoCAD style **SPECIAL** and apply an additional width factor of **1.4** during translation to MicroStation.

Units Selection

Dgn > Dwg

Purpose

Use **Units** when you need to perform specific actions based on the master units definition of the MicroStation file.

Format

```
If Units = unitsname Then.....Endif
```

Arguments

unitsname = MicroStation master units description. Lists and wildcards are supported.

Usage

Units is only valid when translating from MicroStation to AutoCAD.

Example

```
If Units = (ft,"in,") Then  
  ConversionFactor=0.08333333  
Endif
```

will scale all entities by 0.083333 when the DGN units are **ft, ' in, or .**

Note:

Single quotes (') must be specified as two consecutive single quotes (").

Weight Selection

Dgn <> Dwg

Purpose

Use **Weight** when you need to perform specific actions based on an incoming MicroStation or AutoCAD 2000+ element's weight.

Format

If Weight = *weightno* Then.....Endif

Arguments

weightno = MicroStation weight number from **0** to **31** or AutoCAD 2000+ weight number. Lists and ranges are supported.

Usage

Weight is valid for translating both directions.

Example

```
If weight = (1-5) then  
  layer = 15  
endif
```

will select all incoming elements with weights between **1** and **5**. They will be placed on AutoCAD layer **15** during translation.

Width Selection

Dwg > Dgn

Purpose

Use **Width** when you need to perform specific actions based on an incoming AutoCAD polyline's width.

Format

If Width = *widthval* Then ... Endif

Arguments

widthval = AutoCAD polyline width value. This can be a decimal number or an integer. Lists and ranges are supported.

Usage

Width is only valid when translating from AutoCAD to MicroStation.

Example

If Width = 0.5 then Level = 10 Color = 2 Endif

will select all incoming AutoCAD polylines with a width of **0.5**. They will be placed on MicroStation level **10** and have the color set to **2**.

Using Variables

CATS provides you the ability to define your own variables for subsequent use.

Defining a Variable

To define a variable use the Set command.

Format

Set (variable=value)

Arguments:

variable = a name of your choosing representing your data.

value = the value you wish to assign to the variable name.

Example

Set (middle=123)

Using Variables

To use the variable, you must enclose it in percent signs, i.e.

Layer = ABC%middle%XYZ

Example

In the following example:

Set (middle=123)

Layer=ABC%middle%XYZ

the layer name produced will be **ABC123XYZ**.

System Variables

CATS provides some predefined system variables for your use. They are:

%cellname%	name of the current Dwg block or Dgn cell
%color%	color of the current entity
%filedirectory%	directory spec of the current input drawing file
%fileext%	file extension of the current input drawing file
%filename% drawing file	filename minus extension of the current input

%font% current text entity	Dgn font number or Dwg shape file name for the
%layer% name for the current entity	Dgn level name(level number if no level name) or Dwg layer
%level% current entity	Dgn level number or Dwg layer name for the
%linetype%	linetype for the current element
%referencelogical%	logical name of the current input reference drawing file
%weight%	weight of the current Dgn or A2000+ Dwg entity
%width%	width of the current Dwg entity

Translation Specifications

AutoCAD to MicroStation Translations

This section describes how CATS handles various aspects of a translation from AutoCAD to MicroStation. It is arranged alphabetically by topic for easy reference.

Each element is read one at a time from the AutoCAD drawing file. The element is then checked to determine if it will be included in the output MicroStation design file. The element will be included if it has not been specifically excluded by a user program rule.

Several of the topics mention associated control file commands which may be used to control particular aspects of the translation. For specific details on how to format and use these commands, please refer to Customizing CATS

Arcs

Dwg to Dgn

AutoCAD arc entities are translated to MicroStation arc elements.

ASCII Characters

Dwg to Dgn

Text strings are represented by ASCII characters. CATS translates text entities on a character to character basis. Some special characters are remapped to a different ASCII value during conversion. You can remap characters by using the EditText rule.

Attributes

Dwg to Dgn

The AutoCAD attribute entity is converted only if defined as visible. Invisible attributes are ignored.

MicroStation text elements with enter data fields are generated as cell components from the visible attributes unless EnterData Disable is specified. Attribute entities which follow the associated insert entity are used to fill in the enter data fields of the previously defined text element. ATTDEF entities which occur in the entities section of the AutoCAD file but are not associated with a block are translated to text elements.

To convert attributes to MicroStation tags instead of enter data text, use the AttributeToTag command.

AutoCAD Version

Dwg to Dgn

CATS will handle all AutoCAD releases 12 through 2002. It is not necessary for you to either know or specify the incoming version number.

Blocks

Dwg to Dgn

The AutoCAD blocks section is processed to create or update a MicroStation cell library. Each block entry defines a block name; origin point; and the AutoCAD entities which make up the block. A MicroStation cell header is put into the cell library and the entities of the block are translated normally to MicroStation graphic elements. The first 27 characters of the AutoCAD block name will be used as the cell description in the cell library. For versions below V8, If the block name exceeds 6 characters in length, a unique cell name will be generated as **ZZnnnn**; if the block name is 6 or fewer characters in length, the block name will be used as is for the cell name. Blocks which contain no graphic entities will be omitted during translation.

BYBLOCK

Dwg to Dgn

Entities which have their layer, color, linetype, and/or weight defined **BYBLOCK** will have the corresponding MicroStation level, color, linetype, and lineweight assigned at placement time if the resultant cell was defined as a point cell.

If any one of the parameters is set **BYBLOCK**, the resultant cell will be defined as a point cell. When a point cell is placed, all parameters are set at placement time to the currently active value.

You can specify whether cells are to be defined as point cells or not by using the PointCell action.

BYLAYER

Dwg to Dgn

Entities which have their color, linetype, and/or lineweight defined **BYLAYER** will have the corresponding MicroStation color, linetype, or weight assigned using the defined layer defaults.

Cells

Dwg to Dgn

MicroStation cells are created from AutoCAD block definitions. For versions below V8, The cell name is set to the first 6 characters of the AutoCAD block name. If the block name exceeds 6 characters in length, a unique cell name consisting of **ZZ** followed by the AutoCAD block name index (**ZZnnnn**) will be generated for each cell created; if the block name is 6 or fewer characters in length, the block name will be used as is for the cell name.

Circles

Dwg to Dgn

The AutoCAD circle entity is translated to a MicroStation ellipse element with equal axes.

Color

Dwg to Dgn

MicroStation elements generated will be given the AutoCAD entity color after default table lookup unless set otherwise using the Color command.

The Color rule allows you to select entities based on the AutoCAD color.

The default color table follows:

<u>AutoCAD</u>	<u>MicroStation</u>	<u>Color</u>
1	3	red
2	4	yellow
3	2	green
4	7	cyan
5	1	blue
6	5	magenta
7	0	white
30	6	orange

Coordinates

Dwg to Dgn

AutoCAD coordinates are converted to MicroStation coordinates by multiplying the AutoCAD coordinates by the MicroStation UORs value (obtained from the seed file), adding in the offset (defined by Offset or obtained from the seed file), and multiplying by the conversion factor (defined by ConversionFactor).

The global origin from the Dgn seed file will be adjusted as needed to fit the AutoCAD coordinates in the Dgn coordinate space. The NoGlobalOriginAdjustment command can be used to turn off automatic global origin adjustment.

Curves

Dwg to Dgn

AutoCAD polylines which are defined as curves are converted to MicroStation curve elements. Curve-fit points are not placed in the MicroStation curve. Only the original polyline vertices are used to define the MicroStation curve string.

Custom Linetypes

Dwg to Dgn

AutoCAD custom linetypes can be mapped to MicroStation linetypes. The actual linetype definition is not translated. Use CustomLinetype to select the custom linetypes and define a suitable assignment.

Dimensions

Dwg to Dgn

Automatic dimensioning produces INSDIM entities. They are handled by CATS much like insert entities and result in a cell definition and placement.

Donuts

Dwg to Dgn

Donuts are translated to a filled circle or filled complex shape depending on the polyline settings.

Elevation

Dwg to Dgn

AutoCAD elevation (group 38) data will be converted to MicroStation Z coordinates. Most often, elevation data was generated by AutoCAD release 9.

Ellipses

Dwg to Dgn

AutoCAD ellipse entities are converted to MicroStation ellipse elements.

Enter Data Text

Dwg to Dgn

AutoCAD attributes will be converted to MicroStation enter data text. If you prefer to generate regular text, specify EnterData Disable in your control file.

File Extensions

Dwg to Dgn

CATS supports both AutoCAD DWG and DXF formats and requires that the file extension correspond accordingly.

CATS will use **.dgn** for the MicroStation drawing and **.cel** for the MicroStation cell library files generated. Dwg paperspace is converted to a separate MicroStation drawing file, a sheet file with the extension **.s01**.

Fonts

Dwg to Dgn

MicroStation text fonts are assigned by the Font setting. If the font is not specifically assigned, the AutoCAD shape file used will be looked up in the following table and the corresponding font number assigned to the MicroStation text element generated.

AutoCAD Style MicroStation Font

TXT	0
MONOTXT	3
ROMANS	1
ROMANC	2
ROMAND	2
ROMANT	2
ITALICC	23
SCRIPTS	30
SCRIPTC	30

Frozen Layers

Dwg to Dgn

AutoCAD entities on frozen layers will be translated unless specifically excluded. You can use If LayerFrozen to manipulate these entities.

Hatches

Dwg to Dgn

AutoCAD filled, user defined, ANSI, and selected pre-defined Hatches are converted to equivalent MicroStation area patterns.

AutoCAD supported pre-defined hatches

AR-B816	AR-B816C	AR-B88	AR-BRELM
AR-BRSTD	AR-CONC	AR-HBONE	AR-RSHKE
BRASS	BRICK	CLAY	CORK
CROSS	DASH	DOLMIT	EARTH
FLEX	GRASS	GRATE	HEX
HONEY	INSUL	PLAST	PLASTI
SQUARE	STARS	TRIANG	ZIGZAG

The hatch rule can be used with the ScaleCell command to adjust the hatch scale.

Inserts

Dwg to Dgn

AutoCAD insert entities define the specifications for the placement of a block defined in the blocks section of the AutoCAD file. The insert entity specifies placement coordinates; a block name; X and Y scale factors; and a rotation angle.

The MicroStation cell equivalent will have been placed in the cell library from the AutoCAD file block definition during the blocks conversion or it can be pre-existing.

If an insert entity is nested within a block definition, the MicroStation cell library is searched for the equivalent cell.

The cell placement point; rotation; and X and Y scale factors are taken from the insert specifications. The cell will then be placed as a type 2 MicroStation cell. If either the X or Y scale factors are 0, the cell will not be placed.

The only exception to this rule occurs with an AutoCAD insert with a block name of **ELLIPSE**. An AutoCAD **ELLIPSE** insert will be translated to a normal MicroStation ellipse element with major and minor axis lengths calculated from the X and Y scale factors.

AutoCAD multiple inserts will be handled by generating the equivalent array of MicroStation cells.

Justification

Dwg to Dgn

AutoCAD text justification can be used as a selection criteria by using Justification. Text will be converted using the closest possible justification in MicroStation.

These values and their MicroStation defaults are:

<u>AutoCAD</u>	<u>MicroStation</u>	<u>Justification</u>
0	2	Left
1	8	Center
2	14	Right
3	2	Aligned
4	7	Middle
5	2	Fit
10	2	Bottom Left
11	8	Bottom Center
12	14	Bottom Right
20	1	Middle Left
21	7	Middle Center
22	13	Middle Right
30	0	Top Left
31	6	Top Center
32	12	Top Right

Layers

Dwg to Dgn

AutoCAD layers are used to assign default levels to the generated MicroStation elements. Layer names can be used as a selection criteria by use of Layer. For selection by layer, either complete layer names or partial names incorporating wildcards can be specified. All data on all layers will be translated. If you do not want data on layers that are turned off or frozen, you must specifically exclude those layers using either the LayerOff or the LayerFrozen rules.

AutoCAD layer names will be used as the MicroStation level names unless specified otherwise in the control file.

Leaders

Dwg to Dgn

AutoCAD leaders are translated to a MicroStation line string with an arrowhead shape.

Levels

Dwg to Dgn

For versions below V8, MicroStation levels are numbers ranging from **1** to **63**. Assignment is accomplished by the use of the **Level** action. If level is not assigned by the user, level numbers are assigned sequentially to the AutoCAD layer name table. For versions below V8, If more than 63 layers exist, level numbering restarts at **1** for each group of 63 layers.

Lightweight Polylines

Dwg to Dgn

Lightweight polylines are supported by AutoCAD Release 14+. They are treated in the same way as regular polylines.

Lines

Dwg to Dgn

The AutoCAD line entity is translated to a MicroStation line element.

Linetypes

Dwg to Dgn

MicroStation elements generated will be given the AutoCAD entity linetype after default table lookup unless set otherwise using the Linetype action.

Linetype allows you to select entities based on the AutoCAD linetype.

The default linetype table follows:

<u>AutoCAD</u>	<u>MicroStation</u>
CONTINUOUS	0
DOT	1
HIDDEN	2
DASHED	3
DASHDOT	4
DIVIDE	5
PHANTOM	6

CENTER	7
BORDER	8

Offset

Dwg to Dgn

MicroStation allows users to define drawing origin at any coordinate through use of the global origin setting. The global origin value will be extracted from the MicroStation seed file in use to adjust MicroStation coordinates. CATS provides an additional Offset assignment to allow you to add an offset to your drawing coordinates.

MText

Dwg to Dgn

AutoCAD mtext entities will be converted to MicroStation text string elements. Special operations inside mtext entities (e.g., changing font, bolding, etc.) are not supported. Also, multiline text with automatic wrapping at the mtext boundary may not wrap at the same location due to differences in font sizes. Stacked fractions will be converted to Dgn stacked fraction characters.

Paper Space

Dwg to Dgn

Paper space is supported for XRefs attached in both paper space and model space. Nesting is not supported for the attached XRefs. When paper space entities exist in a DWG, a new DGN file with the extension **.s01** is created for the paper space entities. Each paper space viewport will create reference file attachments for the model space file (.dgn) and each model space xref with the appropriate scale, rotation, and clipping.

Points

Dwg to Dgn

The AutoCAD point entity is defined by the X and Y coordinates of the point. A MicroStation zero-length line element is produced with equivalent end point coordinates set to the coordinates of the AutoCAD point.

If SkipPoint Enable is specified, point elements will not be processed.

Point Cells

Dwg to Dgn

Point cells are the closest MicroStation equivalent to AutoCAD blocks with layer, color, and linetype assigned BYBLOCK (i.e., assigned to current values at placement time).

If blocks have layer, color, and/or linetype assigned BYBLOCK on any graphic entities within the block definition, a MicroStation point cell will be defined. You can also force blocks to be converted to point cells by specifying PointCell for blocks selected by name or other criteria.

Polyface Meshes

Dwg to Dgn

The AutoCAD polyface mesh is defined by a series of vertices and a series of pointers to the vertices. A set of pointers is defined for each edge of the mesh. Invisible edges are also indicated.

A series of MicroStation closed shapes and/or linestrings is generated according to the pattern of edges and invisible edges.

Polylines

Dwg to Dgn

The AutoCAD polyline sequence is defined by a series of vertices. The polyline header defines the default start and end widths, and control flags.

Vertices are accumulated as a coordinate string until an arc is defined or the MicroStation weight changes. At that time, any accumulated vertices will be placed as a MicroStation linestring element.

If the polyline header denotes a closed polyline, a MicroStation closed shape element is created instead of a linestring.

Vertices flagged as frame points for a BSpline curve are ignored in the curve string generation.

A MicroStation arc element will be generated for each polyarc segment.

When the polyline terminator entity is located, any accumulated vertex coordinates are written as a linestring or closed shape element.

If the polyline sequence requires generation of multiple MicroStation elements, a complex header will be placed to retain the elements as a single entity.

When the polyline tapers from one vertex to the next, a MicroStation filled shape will be generated to represent the arrowhead shape of the vertex taper.

Reference Files

Dwg to Dgn

CATS will translate XRef attachments to MicroStation reference file attachments to the main file being translated when FileToRef is used. By default, XRef attachments are skipped.

Paper space is supported for XRefs attached in both paper space and model space. Nesting is not supported for the attached XRefs. When paper space entities exist in a DWG, a new DGN file with the extension **.s01** is created for the paper space entities.

Xclipping for Xrefs will be translated to Dgn reference file clip boundaries,

Shapes

Dwg to Dgn

The AutoCAD shape entity is defined by the X and Y coordinates of the placement point; the shape name; the shape size; the rotation angle; and the X axis scale factor.

A MicroStation cell will be generated as described for inserts. The only difference is that the shape must already exist in the MicroStation cell library. Since the shape's graphics are not defined in the AutoCAD file, the equivalent symbol must be pre-defined on the MicroStation system.

Solids

Dwg to Dgn

The AutoCAD solid entity is defined much like the trace entity. The only difference is for 3-sided solids, in which the third and fourth points are given equal X and Y coordinates.

The MicroStation shape element generated will consist of all the vertices specified in the AutoCAD solid, only the third and fourth vertices will be swapped, and the fifth vertex coordinates will be set equal to the first to close the solid. The filled attribute will be set.

Splines

Dwg to Dgn

AutoCAD spline entities are translated to a MicroStation closed rational BSpline curve.

Symbols

Dwg to Dgn

Symbol font characters will normally be converted like other text characters. Use the FontToSymbol action to specify translating a symbol font character to a MicroStation cell instance.

Stacked Fractions

Dwg stacked fractions are converted to Dgn stacked fraction characters

Text**Dwg to Dgn**

The AutoCAD text entity is defined by the placement point for the text; the character height; the character width scaling factor; the angle of rotation about the origin; the actual text characters (maximum of 256); text style name; and type of justification. Additional data will be present to define an alignment point should the justification type be non-0 (0 being lower left).

The MicroStation text element will be placed at the origin defined in the AutoCAD file. The text string will have rotation, scaling, and justification applied as specified by the AutoCAD file. If there is not a match from the control file, text elements will be created using MicroStation font 0.

Text Styles**Dwg to Dgn**

MicroStation text fonts are assigned by the Font action. If the font is not specifically assigned, the AutoCAD shape file used will be looked up in the following table and the corresponding font number assigned to the MicroStation text element generated.

<u>AutoCAD</u>	<u>MicroStation</u>
TXT	0
MONOTXT	3
ROMANS	1
ROMANC	2
ROMAND	2
ROMANT	2
ITALICC	23
SCRIPTS	30
SCRIPTC	30

Traces**Dwg to Dgn**

The AutoCAD trace entity is defined by the coordinates of the four corners of the trace outline.

A MicroStation closed shape element is created consisting of 5 vertices. The first and fifth vertices have equivalent coordinates, thereby closing the shape. The filled attribute will be set.

This method of conversion is used to maintain the exact placement of the trace upon translation. If traces are used in AutoCAD to have width, it is preferable to use a polyline which will convert to a string element with weight.

Units**Dwg to Dgn**

AutoCAD coordinates are converted to MicroStation coordinates by multiplying the AutoCAD coordinates by the MicroStation UORs value (obtained from the seed file), adding in the offset (defined by Offset = or obtained from the seed file), and multiplying by the conversion factor (defined by ConversionFactor).

Weight**Dwg to Dgn**

Weight is a logical line thickness that does not vary with scale. If you zoom in and out, the apparent thickness of the line does not vary. The Weight rule can be used to test the weight and set any appropriate action. Weight is supported in MicroStation and AutoCAD 2000+.

CATS ignores element weight values for versions below AutoCAD 2002.

Valid lineweight values for AutoCAD are: 0.00, 0.05, 0.09, 0.13, 0.15, 0.18, 0.20, 0.25, 0.30, 0.35, 0.40, 0.50, 0.53, 0.60, 0.70, 0.80, 0.90, 1.00, 1.06, 1.20, 1.40, 1.58, 2.00, 2.11 mm.

For AutoCAD 2000+, the Dwg weight is used to set the Dgn weight as follows:

<u>AutoCAD</u>	<u>MicroStation</u>
0.00 mm	0
0.05	1
0.09	2
0.13	3
0.15	4
0.18	5
0.20	6
0.25	7
0.30	8
0.35	9
0.40	10
0.50	11
0.60	12

0.70	13
0.80	14
0.90	15
1.00	16
1.06	17
1.20	18
1.40	19
1.58	20
2.00	21
2.11	22

Width

Dwg to Dgn

By default, CATS ignores polyline width values. MicroStation weight values can be assigned using Weight. The Width selection can be used to define actions based on AutoCAD width.

XRefs

Dwg to Dgn

CATS will translate XRef attachments to MicroStation reference file attachments to the main file being translated when the FileToRef rule is used. By default, XRef attachments are skipped and omitted from translation.

Xclipping for Xrefs will be translated to Dgn reference file clip boundaries,

3DFaces

Dwg to Dgn

The AutoCAD 3Dface entity is defined by the coordinates of the 4 corners of the face. Each vertex is defined with X, Y, and Z values. The Z values are ignored by CATS to generate a 2D representation.

A MicroStation closed shape element is created consisting of 5 vertices. The 4 corners are the first 4 vertices and the fifth vertex is set equal to the first to close the shape.

3DLines

Dwg to Dgn

The AutoCAD 3Dline entity is defined by the X, Y, and Z coordinate values of the end points of the line. The Z values are ignored by CATS to generate a 2D representation.

A MicroStation line element is generated with the start and end point coordinates set to the X and Y coordinates from the AutoCAD line.

MicroStation to AutoCAD Translations

This section describes the translation process from MicroStation to AutoCAD and how CATS handles various aspects of the translation from element types to level symbology to reference files. It is arranged alphabetically by topic for easy reference.

Several of the topics mention associated user-defined rules which may be used to control particular aspects of the translation. For specific details on how to format and use these rules, please refer to Customizing CATS.

Each element is read one at a time from the design file. The element is then checked to determine if it will be included in the output AutoCAD file. The element will be included if it is inside any clipping fences, and its level is on in the selected view, and it has not been excluded by a user rule.

The element characteristics are set to default values unless set to different values by a user-defined rule. If level symbology has been enabled in the design file, the level symbology values will be used instead of the original element characteristics.

Arcs

Dgn to Dwg

MicroStation arc elements have 5 possible AutoCAD equivalents: arc, circle, point, polyarc, or polyline.

MicroStation circular arcs are translated directly to AutoCAD arcs.

MicroStation arcs with a sweep angle of 0 or +/- 360 degrees are converted to an AutoCAD circle.

MicroStation arcs with an axis length of 0 are translated to an AutoCAD point.

Circular arcs with a non-0 width (set by the Width user program rule) are converted to polyarcs to maintain width.

Since AutoCAD arcs must be circular for R12 and below, CATS converts elliptical arcs (partial ellipses) regardless of weight to AutoCAD polylines. For each *N* degree interval (specified by the ArcStrokes rule) of the arc's sweep, a straight segment is drawn to approximate that portion of the arc. Since CATS creates several segments per elliptical arc, it can be expected that the size of the output AutoCAD file will increase depending on the number of elliptical arcs in the drawing.

For releases 13+ conversions, elliptical arcs are converted to AutoCAD elliptical arcs.

Arrowheads

Dgn to Dwg

MicroStation arrowheads which are text characters referencing fonts **85**, **102**, **111**, and **126** are translated to special AutoCAD block **XXXXAHD**. The block **XXXXAHD** contains an AutoCAD solid entity appearing as a filled arrowhead symbol.

Associative Patterns

Dgn to Dwg

MicroStation 4, 5, and 95 associative pattern elements are not handled directly by CATS. You must freeze them through MicroStation prior to translation. You can also choose the Freeze DGNs Before Translation option on the Parameters tab.

The freeze MDL components must be copied to the appropriate MicroStation directories before using this option. See the **freeze.txt** file for details.

Attributes

Dgn to Dwg

Enter data text in a cell is translated to attribute text in a block. EnterData Disable can be used to disable the creation of attribute text in a block. The enter data text will be placed as regular text after the insert for the block.

Tags are also converted to attributes.

AutoCAD Version

Dgn to Dwg

CATS DWG output is compatible with AutoCAD releases 12 through 2009. To produce a DWG file compatible with a previous version, use the Seedfile option from the options menu or select the appropriate version in the Output tab of the options menu.

Blocks

Dgn to Dwg

MicroStation cells are translated to AutoCAD blocks with insert references. Whenever a cell header is encountered, an AutoCAD insert entity is placed in the AutoCAD file with a block name; insertion point; rotation; and X and Y scale factors.

The first occurrence of each cell in the design file is used to create the block definition.

If the cell has component graphic elements on various levels, the levels not selected for translation will be omitted in the generated block definition.

If a cell has been selected for explosion via the ExplodeCell rule, the cell header will be ignored and its component graphic elements will be translated individually.

By default, the block will be named with the cell name. The Block command can be used to specify any cell name assignments.

Blank cell names are not translated. If a MicroStation cell has an all blank name the graphics are translated individually but the block is not created.

Text elements which consist of enter data fields will be placed in the AutoCAD block as attribute data to maintain the actual text characters in AutoCAD.

BSplines

Dgn to Dwg

MicroStation BSplines are stroked to the visually equivalent polyline sequence. If PolyLine Disable is specified, each stroked segment is written out as an individual line entity for AutoCAD release 12 and below.

To adjust the size of the generated polyline, use BCurveStrokes. This value defaults to **3** and represents the number of points that will be inserted between each original vertex pair. Increasing this value will increase the number of points generated, the smoothness of the curve, and the size of the AutoCAD file. Decreasing this value will decrease the number of points generated, the smoothness of the curve, and the size of the AutoCAD file.

For AutoCAD releases 13+, spline entities are created.

Cells

Dgn to Dwg

MicroStation cells are translated to AutoCAD blocks with insert references. Whenever a cell header is encountered, an AutoCAD insert entity is placed in the AutoCAD file with a block name; insertion point; rotation; and X and Y scale factors.

The first occurrence of each cell in the design file is used to create the block definition.

If the cell has component graphic elements on various levels, the levels not selected for translation will be omitted in the generated block definition.

If a cell has been selected for explosion via the ExplodeCell rule, the cell header will be ignored and its component graphic elements will be translated individually.

By default, the block will be named with the cell name. The Block command can be used to specify any cell name assignments.

Blank cell names are not translated. If a MicroStation cell has an all blank name the graphics are translated individually but the block is not created.

Text elements which consist of enter data fields will be placed in the AutoCAD block as attribute data to maintain the actual text characters in AutoCAD.

Characters

Dgn to Dwg

Characters in a text string are translated to AutoCAD as is, with the following exceptions:

- The degree symbol (ASCII value **94**) is translated to the AutoCAD degree symbol (%%d).
- The plus/minus symbol (ASCII value **200**) is translated to the AutoCAD plus/minus symbol (%%p).

EditText can be used to map additional characters as desired.

Class

Dgn to Dwg

Each MicroStation element contains a class field giving the class of the element. The Class selection rule can be used to test the value of this field to take an appropriate action. The values of the class field are:

- 0 = primary element
- 1 = pattern component
- 2 = construction element
- 3 = dimensioning element
- 4 = primary rule element
- 5 = linear patterned element
- 6 = construction rule element

Clip Boundaries

Dgn to Dwg

CATS supports clipping to clip boundaries in attached reference files during normal translations. Clipping is translated to Xclipping when FileToXRef is specified.

Graphic elements will normally be clipped at the boundaries and partial elements written to the generated AutoCAD file. Text elements are handled as either in or out of the clip boundaries. The actual text string is not clipped.

Clip Mask Areas

Dgn to Dwg

CATS supports clip mask areas during translation to AutoCAD. Clip masks are not handled when FileToXRef is specified.

Color**Dgn to Dwg**

AutoCAD entities generated will be assigned an element color using the MicroStation color for color table lookup.

Level symbology for both the main and reference files, if enabled, will be retrieved prior to conversion and colors adjusted accordingly.

Color provides user color assignments including BYLAYER and BYBLOCK as needed.

The default color table follows:

<u>MicroStation</u>	<u>AutoCAD</u>	<u>Color</u>
0	7	white
1	5	blue
2	3	green
3	1	red
4	2	yellow
5	6	violet
6	30	orange
7	4	cyan

Complex Shapes**Dgn to Dwg**

MicroStation complex shapes are translated to AutoCAD closed polylines. All of the elements within the complex shape definition will be converted into a single polyline to maintain the closed property of the complex shape. Complex shapes with no component elements are ignored.

Complex Strings**Dgn to Dwg**

MicroStation complex strings are translated to an AutoCAD polyline. All of the elements within the complex string definition will be converted into a single polyline to maintain the single element status. Complex strings with no component elements are ignored.

Construction Class**Dgn to Dwg**

MicroStation construction class elements are normally translated by CATS. If you need to exclude construction elements, use the following rule:

If Class=2 then ExcludeElement endif

Coordinates

Dgn to Dwg

AutoCAD coordinates are computed using MicroStation master units minus global origin. These coordinates correspond to those seen using MicroStation tentative point with tpmode set to locate. If sub units are 12, i.e., inches, then coordinates are output in inches for use with AutoCAD architectural units (4). Coordinates may be modified by the Offset or ConversionFactor user program rules in the control file.

Curves

Dgn to Dwg

The MicroStation curve element is defined by the count of vertices and the coordinates of the vertices defining the curve. The first 2 and last 2 vertices of each curve element are ignored since they are present only for curve fitting purposes.

The MicroStation curve element will be translated into an AutoCAD polyline entity in much the same manner as linestring elements. Additional curve fit vertices are added for smoothing. The CurveStrokes rule may be used to adjust the smoothness of the generated curve. The first 2 and last 2 vertices of every curve are not included in the translation. If PolyLine Disable is set, each segment is written out as an individual line entity.

Custom Linetypes

Dgn to Dwg

MicroStation custom linetypes can be mapped to AutoCAD linetypes. The actual linetype definition is not translated. Use CustomLinetype to select the custom linetypes and define a suitable assignment.

Dimensions

Dgn to Dwg

MicroStation associative dimension elements are not handled directly by CATS. You must freeze them through MicroStation prior to translation. You can also choose the Freeze DGNs Before Translating option on the Parameters tab.

The freeze MDL components must be copied to the appropriate MicroStation directories before using this option. See the **freeze.txt** file for details.

Ellipses

Dgn to Dwg

MicroStation ellipse elements are defined by the ellipse center point coordinates; major and minor axis lengths; and angle of rotation. For a description of the handling of partial ellipses, refer to arcs.

There are 3 possible AutoCAD entity equivalents of a MicroStation ellipse: a circle entity, a special **ELLIPSE** insert entity, and a point entity.

An AutoCAD circle entity is produced if the major and minor axis lengths of the MicroStation ellipse are equal. A circle entity is defined by center point coordinates and the radius.

If the major and minor axes are not equal, an AutoCAD insert will be generated with a block name of **ELLIPSE** for release 12 and below. The **ELLIPSE** insert entity will have X and Y axis scale factors calculated to represent the major and minor axes; and it is given the rotation angle of the MicroStation ellipse element.

For AutoCAD releases 13+, an ellipse entity is created.

An AutoCAD point entity is created when the MicroStation ellipse major axis length equals 0.

Filled Circles

Dgn to Dwg

MicroStation filled circle elements are converted to AutoCAD donuts.

Filled Shapes

Dgn to Dwg

MicroStation filled shape elements are defined in much the same way as shape strings. They have the filled attribute in user data set which causes them to display filled in.

MicroStation filled shape elements are normally translated to a series of AutoCAD solids for R13 and below. For R14+, a solid hatch is generated.

Fonts

Dgn to Dwg

MicroStation text elements have a font number representing a font from a font library. This font number is used to select a default text style for AutoCAD. TextStyle can be used for a user selected text style.

The default text style assignments are:

<u>MicroStation Font</u>	<u>AutoCAD Style</u>
0	TXT
1	ROMANS
2	ROMANC
3	MONOTXT
23	ITALICC
30	SCRIPTS

Define Style can be used to include user style definitions in the AutoCAD file. The name, shape file, default height, default width factor, and obliqueing angle can be specified. You can also specify whether to output the style definition always or only if it was used during the translation.

Fractions

Dgn to Dwg

MicroStation text fonts include special characters for fraction values: halves, fourths, eighths, sixteenths, thirty-seconds, and sixty-fourths. The characters are translated to the individual characters of the fraction, e.g., $3/8$. The width factor of the text string is adjusted to keep the approximate length of the string equivalent.

FractionStack Enable can be used to transfer the fraction characters to AutoCAD unchanged for use with AutoCAD shape files containing the special fraction characters. An AutoCAD text font with MicroStation fractions is included with Cats: SimplexF.shx.

Justification

Dgn to Dwg

MicroStation text elements have a justification field giving the position of the text around the placement point. These values and their AutoCAD defaults are:

<u>MicroStation</u>	<u>AutoCAD</u>	<u>Justification</u>
0	30	Left Top
1	20	Left Center
2	0	Left Bottom
6	31	Center Top
7	21	Center Center
8	1	Center Bottom
12	32	Right Top
13	22	Right Center
14	2	Right Bottom

Justification can be used to select different values. FittedText Enable can be used to override justification and use the AutoCAD fit text between two points mode.

Layers

Dwg to V8 Dgn, Dgn to Dwg

AutoCAD layer names will be given the V7 level number or the V7 or V8 Level Name (if given) unless other assignments are made.

Layer provides the ability to assign AutoCAD or V8 layer names.

Layer definitions can be generated with CATS using the Define Layer command. The Define Layer command enables you to define the layer name, default color, default linetype, and default lineweight for the layer. You can also specify whether to output the layer definition always or only if it was used during the translation.

Levels**Dgn to Dwg**

AutoCAD entities generated will be given the MicroStation level name as the AutoCAD layer name. If there is no MicroStation level name, the level number will be used as the AutoCAD layer name.

Layer provides the ability to assign AutoCAD layer names.

Level Symbology**Dgn to Dwg**

Level symbology is processed prior to the actual element conversion. The incoming MicroStation element will have its color, linetype, and/or weight reset per any level symbology defined and enabled. The enabled level symbology settings are used by CATS for any parameter table lookups, assignments, etc. Please keep this in mind when setting up your control file(s).

Lines**Dgn to Dwg**

MicroStation line elements have 3 possible AutoCAD equivalents: the line entity, the point entity, and the polyline entity.

A point entity will be generated if the X and Y coordinates of the start point are equivalent to the X and Y coordinates of the end point of the MicroStation line element.

Otherwise, an AutoCAD line or polyline entity will be generated. A line or polyline consists of the X and Y coordinates of the start and end points of the line element.

If you have used Width to set a width to 0, a line will be generated. If the selected width is greater than 0, a polyline will be created. If PolyLine Disable is specified, each line is written out as a line entity regardless of any width settings.

Linestrings**Dgn to Dwg**

MicroStation linestring elements are normally translated into AutoCAD polyline entities. Polyline begin and end widths are assigned according to the Linetype rule. However, if PolyLine Disable is set, each linestring segment is written out as individual line entities regardless of any width setting.

Linetypes**Dgn to Dwg**

AutoCAD entities generated will be given the MicroStation element linetype after default table lookup. Level symbology for both the main and reference files, if enabled, will be retrieved prior to conversion and linetypes adjusted accordingly.

Linetype provides user linetype assignments including BYLAYER and BYBLOCK as needed.

The default linetype table follows:

<u>MicroStation</u>	<u>AutoCAD</u>
0	CONTINUOUS
1	DOT
2	HIDDEN
3	DASHED
4	DASHDOT
5	DIVIDE
6	PHANTOM
7	CENTER

Logical Names

Dgn to Dwg

Reference files attached to MicroStation files may be assigned a logical name. This name can be used with the ReferenceLogical selection rule to specify appropriate actions. The logical name can also be used with Set variables to use the name in other commands to set layer names, etc.

Multilines

Dgn to Dwg

MicroStation multiline elements are not handled directly by CATS. You must freeze them through MicroStation prior to translation. You can also choose Freeze DGNs Before Translation on the Parameters tab.

The freeze MDL components must be copied to the appropriate MicroStation directories before using this option. See the **freeze.txt** file for details.

Points

Dgn to Dwg

MicroStation points are actually zero-length line elements. They are translated normally to AutoCAD point entities. If the SkipPoint Enable rule is used in the control file, point entities will not be generated.

Point Cells

Dgn to Dwg

Point cells are translated the same as regular cells except for layer references. The block and its graphic components are placed on layer 0. Each insert reference will be placed on the layer corresponding to the first level used by that occurrence of the point cell.

Individual entities in the block definition will have the color and linestyle defined as BYBLOCK .

The PointCell rule can be used to force a cell to be treated as a point cell.

Polylines

Dgn to Dwg

AutoCAD polylines are created from MicroStation linestring, shape, curve, complex string, complex shape, and bspline elements. They may also be created for line and arc elements when the Width rule is used. Polyline creation can be turned off with PolyLine Disable. Polylines are assigned widths based on the current status of the Width rule.

Prototype Files

Dgn to Dwg

An AutoCAD prototype file can be specified by using the Seedfile selection in the Options menu.

Reference Files

Dgn to Dwg

CATS will translate up to 255 reference files attached to the main file being translated. The reference file will be translated if display is on for the selected view in the attachment and if the file can be located. If level symbology is enabled in the main file, reference file level symbology will be used when processing the reference file.

There are four options for handling reference files. These options can be individually selected for each reference file using user program rules.

- Clip and include in the output drawing (for V7 MicroStation only)
- Clip and convert to a block (for V7 MicroStation only)
- Attach as an XRef
- Skip the file

The V7 default option is to ***clip and include in the output drawing***. This option translates the master design file and all attached clip-bound reference files to a single AutoCAD file. CATS handles scaling, rotation, offsets, and clipping to clip boundaries automatically.

The ***clip and convert to a block*** option translates the master design file to a single AutoCAD file. The attached clip-bound reference file is converted to a block and included in the AutoCAD file. CATS handles scaling, rotation, offsets, and clipping to clip boundaries automatically. FileToBlock Enable is used to select this option. The block is named with the reference file logical name or the reference file name if no logical name exists. The block can be given a different name with the FileBlockName rule.

The ***Attach as an XRef*** option (Default for V8) translates the master design file to an AutoCAD file and uses the XRef functionality to create an attachment for the reference file. The FileToXref Enable rule is used to select this option. The XRef attachment is generated retaining the reference file name, scaling, rotation, and origin. Clipping is translated to Xclipping.

The *skip the file* option translates the master design file to an AutoCAD file. The reference file is ignored. SkipFile is used to select this option.

Reference files will be searched for in all combinations of the following locations:

- Master design file directory
- Logical directory (retrieved from the config.log file list)
- Device:directory as attached
- Reference file directories given on the options menu and extensions:
 - Specified extension
 - .DGN extension
 - Master design file extension

If a reference file cannot be located with any of the above combinations, a message will be written to the message file.

The config.log file contains a list of files which have MicroStation logical name definitions. The default file delivered with CATS contains the names of the standard MicroStation configuration files configure.dat and uconfig.dat . See CATS Files for more information on setting up and modifying this file.

Shapes

Dgn to Dwg

MicroStation shape elements are defined in much the same way as linestrings except that the first and last vertices of the shape have the same coordinates, thus closing the shape.

MicroStation shape elements are normally translated to AutoCAD closed polyline entities. Polyline begin and end widths are assigned according to the Width rule. However, if PolyLine Disable is specified, each shape segment is written out as individual line entities regardless of any Width settings.

Shared Cells

Dgn to Dwg

MicroStation shared cells consist of 2 element types: (34) shared cell definition, and (35) shared cell instance. Typically, the type 34 definition occurs once in a MicroStation file, and the type 35 instance is present for each occurrence.

Shared cell definitions are converted to AutoCAD block definitions. An AutoCAD insert entity is placed for each shared cell instance. The individual elements within the shared cell definition are translated as described for regular cells and the individual component elements.

Shared cell name assignments are handled the same as for regular cells. In addition, the suffix **_SH** will be appended to the end of the block name being generated. This will differentiate between a type 2 cell and a type 34

shared cell which may have the same name yet entirely different graphics associated with them. If a shared cell definition and/or instance has an all-blank name, it will be ignored.

If MakeAnonymousBlock is specified, only type 2 cells will be converted in this way. Shared cells will not be converted to an anonymous block.

Symbols

Dgn to Dwg

Symbol font characters will normally be converted like other text characters. Use the FontToSymbol rule action to specify translating a symbol font character to an AutoCAD block instance.

Tags

Dgn to Dwg

MicroStation tags are translated to AutoCAD attributes. The tag name is used for the attribute name. The tag prompt and default value are also converted to the AutoCAD attribute prompt and default value.

Text

Dgn to Dwg

MicroStation text elements are translated to AutoCAD text entities.

FittedText Enable may be used to place text in AutoCAD as aligned. This technique uses the MicroStation element range box to place the text using the same coordinate fit.

MicroStation text strings consisting of enter data text will be translated to an AutoCAD attribute entity as described above.

MicroStation text characters which are arrowheads in fonts **85**, **102**, and **126** are translated to special AutoCAD block XXXXAHD . Block XXXXAHD contains an AutoCAD solid entity appearing as a filled arrowhead symbol.

Text elements which cross reference file clipping boundaries will be included in the translation.

Text Styles

Dgn to Dwg

The following table gives the default text style assignments for most often used MicroStation fonts. Different style values may be assigned using TextStyle.

<u>MicroStation Font</u>	<u>AutoCAD Style</u>
0	TXT
1	ROMANS

2	ROMANC
3	MONOTXT
23	ITALICC
30	SCRIPTS

UORs

V7 Dgn to Dwg

MicroStation coordinates are stored as 4 byte integer UORs: Units of Resolution. The possible UOR range is -2 billion to +2 billion. This range is divided by the user into master units, sub units, and positional units to calculate the actual coordinate. This number is then added to the global origin to compute the final coordinate. These coordinates correspond to those seen using MicroStation tentative point with tpmode set to locate. If sub units are 12, i.e., inches, then coordinates are output in inches for use with AutoCAD architectural units (4). Coordinates may be modified by the Offset or ConversionFactor rules in the control file.

Weight

Dgn to Dwg

Weight is a logical line thickness that does not vary with scale. If you zoom in and out, the apparent thickness of the line does not vary. The Weight rule can be used to test the weight and set any appropriate action. Weight is supported in MicroStation and AutoCAD 2000+.

CATS ignores element weight values for versions below AutoCAD 2002. AutoCAD width values can be assigned using Width. Widths, when requested, will be assigned to those element types which can be represented as polylines, i.e., line, linestring, curve, arc, and bspline. Other elements will ignore width settings.

Valid lineweight values for AutoCAD are: 0.00, 0.05, 0.09, 0.13, 0.15, 0.18, 0.20, 0.25, 0.30, 0.35, 0.40, 0.50, 0.53, 0.60, 0.70, 0.80, 0.90, 1.00, 1.06, 1.20, 1.40, 1.58, 2.00, 2.11 mm.

For AutoCAD 2000+, the Dgn weight is used to set the AutoCAD weight as follows:

<u>MicroStation</u>	<u>AutoCAD</u>
0	0.00 mm
1	0.05
2	0.09
3	0.13
4	0.15
5	0.18
6	0.20
7	0.25

8	0.30
9	0.35
10	0.40
11	0.50
12	0.60
13	0.70
14	0.80
15	0.90
16	1.00
17	1.06
18	1.20
19	1.40
20	1.58
21	2.00
22	2.11

Width

Dgn to Dwg

Width in an AutoCAD drawing is an actual thickness in drawing units. As you zoom in and out the thickness varies with the scale. Width can be used to set width using any appropriate selection.

XRefs

Dgn to Dwg

The **Attach as an XRef** option translates the master design file to an AutoCAD file and uses the XRef functionality to create an attachment for the reference file. FileToXRef Enable is used to select this option. The XRef attachment is generated retaining the reference file name, scaling, rotation, and origin. Clipping is translated to Xclipping. See the Reference Files section for a complete discussion of reference file options.

Cats Files

Sample Directory

The sample directory is created as a subdirectory of the **cats7** directory. It contains sample files to help you gain experience with CATS. The following files are included:

ARCH.DGN

MicroStation DGN with **bor.dgn** as a reference file.

ARCHDWG.DWG

AutoCAD DWG with **bordwg.dwg** as an XRef.

BOR.DGN

MicroStation border DGN.

BORDWG.DWG

AutoCAD border DWG.

DGNDWG.CON

Control file to translate DGN to DWG.

DGNDWG.PRJ

Project file to translate DGN to DWG.

DWGDGN.CON

Control file to translate DWG to DGN.

DWGDGN.PRJ

Project file to translate DWG to DGN.

PIPE.DGN

MicroStation DGN with **bor.dgn** as a reference file.

SECTION.DWG

AutoCAD DWG with **bordwg.dwg** as an XRef.

File Name Conventions

There are some general conventions adhered to by CATS regarding file specifications. Extensions are defaulted as follows:

.CEL	MicroStation format cell library file
.CON	CATS control file
.DGN	MicroStation format design file
.DWG	AutoCAD format binary drawing file
.DXF	AutoCAD standard interchange format ASCII file
.LST	CATS control list file
.LOG	CATS message file
.PRJ	CATS project file
.S01	MicroStation sheet file

CATS Delivery Files

The following files should be present on your system after a successful installation:

ACADSeed.DWG

AutoCAD standard prototype file used to create DWG output files. It has the standard AutoCAD linetypes loaded. This file may be altered or replaced to suit project requirements.

ADINIT8.DAT

System file for reading and writing AutoCAD files.

CATS8.EXE

Main CATS executable.

C8HELP.HLP, C8HELP.CNT

Help files for Cats.

CMDLINE.TXT

Text file explaining command line batch processing.

CONFIG.LOG

ASCII file containing the default path and file spec for the MicroStation environment variable files **config.dat** and **uconfig.dat**. The directories may be modified as necessary to suit your MicroStation configuration. You may also add additional file specs to this list. The only requirement is that the files contain logical assignments in the form *logical=actual*. Multiple assignments separated by a semicolon (;) are also allowed.

DGNDWG.CON

Sample control file for translating DGN to DWG.

DWGDGN.CON

Sample control file for translating DWG to DGN.

FREEZE.TXT, FREEZE.MA, FREEZE.UCF

Files used for dropping dimensions and hatches in V7 Dgn files. Requires MicroStation. See Freeeze.txt for usage instructions.

NEW.PRJ

Default project file for new projects.

PRODUCT.DEF

Text file containing the current version number of the software.

README.TXT

Text file containing information on the latest modifications and changes to Cats.

SEEDARC2.DGN, SEEDARC3.DGN, V8SeedArch2.dgn, V8SeedArch3.dgn

Seed 2D and 3D DGN files with typical architectural setup - feet and inches - 12,8000 working units.

SEEDMAP2.DGN, SEEDMAP3.DGN

Seed 2D and 3D DGN files with typical mapping setup - feet and tenths - 10,100 working units.

SEEDME2.DGN, SEEDME3.DGN

Seed 2D and 3D DGN files with typical mechanical setup - inches and thousandths - 1000,254 working units.

V8SeedMetric2.dgn, V8SeedMetric3.dgn

Seed 2D and 3D DGN files with typical metric setup.-

SIMPLEXF.SHP

AutoCAD standard **SIMPLEX** shape source file with MicroStation-compatible standard single-character fractions added.

SIMPLEXF.SHX

AutoCAD standard **SIMPLEX** compiled shape file with MicroStation-compatible single-character fractions added. This file may be used instead of the AutoCAD standard **SIMPLEX** files when MicroStation single-character fractions occur in a design file. If this file is not used, the fractions are generally displayed as a question mark in the AutoCAD drawing. To reference these single-character fractions while working in AutoCAD, use the %% special character notation or the <Alt> key followed by the desired value from the following table.

For example, enter:

%%133

or:

<ALT>133

in response to the TEXT prompt to place:

3/8

in the drawing as a single-character fraction. The following fractions are predefined in this file:

129=1/2	145=3/32
130=1/4	146=5/32
131=3/4	147=7/32
132=1/8	148=9/32
133=3/8	149=11/32
134=5/8	150=13/32
135=7/8	151=15/32
136=1/16	152=17/32
137=3/16	153=19/32
138=5/16	154=21/32
139=7/16	155=23/32
140=9/16	156=25/32
141=11/16	157=27/32
142=13/16	158=29/32
143=15/16	159=-31/32
144=1/32	

TRAN8.DLL

DLL file for the CATS translator.

2DSEED.CEL

V7 MicroStation standard format 2D cell library seed file. This file may be altered or replaced to suit project requirements.

2DSEED.DGN

V7 MicroStation standard format 2D drawing seed file. This file may be altered or replaced to suit project requirements.

3DSEED.CEL

V7 MicroStation standard format 3D cell library seed file. This file may be altered or replaced to suit project requirements.

3DSEED.DGN

V7 MicroStation standard format 3D drawing seed file. This file may be altered or replaced to suit project requirements.

Error Messages

CATS generates error messages in the message file **cats8.log**. A description of each message follows:

1 - File Not Found *filename*

The *filename* given could not be located.

2 - Invalid Input Type

An invalid input type was encountered in an options file. The valid values are **DGN**, **DXF**, and **DWG**.

3 - Invalid Design File

The input design file was not a MicroStation design file or has serious errors and is not usable.

4 - Invalid Output Type

An invalid output type was encountered in an options file. The valid values are **DGN**, **DXF**, and **DWG**.

5 - Invalid Input Dimension

An invalid input dimension was encountered in an options file. The valid values are **2** and **3**.

6 - Invalid Input View

An invalid input view was encountered in an options file. The valid values are **1** through **8**.

7 - Invalid Output Dimension

An invalid output dimension was encountered in an options file. The valid values are **2** and **3**.

8 - Error Reading DGN File

An unrecoverable error occurred reading the input DGN file.

12 - Invalid DXF Group

An unrecognized group code was encountered reading a DXF file.

13 - Layer Table Overflow

An input AutoCAD DXF or DWG file contained more layers than could be handled by CATS. Please contact DGI Technical Support for help with this problem.

14 - Invalid Layer

A reference to a layer not defined in the layer table was encountered reading an AutoCAD DWG or DXF file.

15 - Invalid DWG Entity Type

An unrecognized entity type was encountered reading a DWG file.

16 - Invalid If Keyword

An unrecognized If selection was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

17 - Invalid Then Keyword

An unrecognized Then action command was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

18 - Invalid Color

An invalid color was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

19 - Invalid Linetype

An invalid linetype was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

20 - Invalid Level Number

An invalid level was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

21 - Invalid Layer Name

An invalid layer was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

22 - Invalid Width

An invalid width was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

23 - Invalid Weight

An invalid weight was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

24 - Invalid Text Style Name

An invalid text style name was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

25 - Invalid Font

An invalid font was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

26 - Invalid Symbol Name

An invalid symbol, cell, or block was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

28 - Invalid Entity Type

An invalid entity type was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

29 - Unknown Define Parameter

An unrecognized item in a Define command was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

30 - Invalid Graphic Group

An invalid graphic group was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

31 - Invalid Class

An invalid class was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

32 - Invalid Cell Name

An invalid cell was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

33 - Invalid File Directory

An invalid file directory was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

34 - Invalid File Name

An invalid file name was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

35 - Invalid File Extension

An invalid file extension was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

36 - Invalid Reference Logical

An invalid reference logical was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

37 - Invalid Reference Scale

An invalid reference scale was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

38 - Invalid Text Height Factor

An invalid text height factor was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

39 - Invalid Text Width Factor

An invalid text width factor was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

40 - Invalid Text Oblique Angle

An invalid text oblique angle was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

41 - Missing Then Clause

An Rules Overview selection set was given without a corresponding Then clause in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

42 - Missing Endif

An Endif was missing to terminate a rule statement in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

43 - Invalid Keyword

An unrecognized keyword was encountered in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

44 - Missing Close Parenthesis

The close parenthesis was missing from an EditText , Offset , Define Layer , Define Header , or Set action in a control file. This message is printed in the message file immediately following the line containing the error when the ExpandedMessageFile option is used.

45 - Max Number Cells Exceeded

An input MicroStation DGN file contained more cells than could be handled by CATS. Please contact DGI Technical Support for help with this problem.

46 - Bad Variable Replacement

When using variable replacement (%*varname*%) the variable had not been defined or was missing the closing %.

47 - Control File Max Size Exceeded

A control file contained more commands than could be handled by CATS. Please contact DGI Technical Support for help with this problem.

48 - Reference BDR Not Found

The BDROFFSET or BDRBOUND command was included in the options file, but the input design file did not have a reference file with the **BDR** logical name attached.

49 - Border Cells Not Found

The BDROFFSET or BDRBOUND command was included in the options file, but the reference file with the **BDR** logical name did not contain the border cells - **LFTBTM** and **RHTTOP**.

50 - Linetype Not Defined

A linetype was requested which was not predefined in the acadseed.dwg file. All linetypes must be predefined in the prototype file when translating to DWG.

51 - Invalid DWG File

The acadseed.dwg file used for creating new DWG files is not a valid AutoCAD drawing file.

52 - Block Not Defined

An insert was requested for a block which had not been defined in the DWG file.

53 - Block Table Overflow

A DWG file contained more blocks than could be handled by CATS. Please contact DGI Technical Support for help with this problem.

54 - Linetype Table Overflow

A DWG file contained more linetypes than could be handled by CATS. Please contact DGI Technical Support for help with this problem.

55 - Shape Table Overflow

A DWG file contained more shapes than could be handled by CATS. Please contact DGI Technical Support for help with this problem.

56 - Invalid PRF File

The input PRF file was not a valid PRF.

57 - Cell Not Defined

A attempt was made to place a cell in a Dgn file which was not defined.

58 - Shapefile Not Loaded

The FontToShape command was used and the necessary shapefile was not preloaded in the Dwg seedfile.

59 - Invalid Cell Library

The input cell library was not a valid cell library file.

61 - Associative Dimension Skipped

A Dgn Associative dimension was encountered and skipped. Associative dimensions are not supported by Cats. You can use the freeze feature in Cats to drop associative dimensions during translation.

62 - Error reading Dwg

An error occurred reading a Dwg file. The Dwg may be bad. Try using the AutoCAD recover or audit command to fix the file.

63 - Clip void not supported

A Dgn reference file with clipping and voids was converted to a Dwg Xref with Xclipping. The clipping voids were ignored.

Contacting DGI

DGI Web Site

Visit our web site www.dgnlink.com for technical support tips, FAQs, updates available for downloading, and information on our other products and services.

Sales

To sign up for Support PLUS, purchase additional copies of this software, or order any other DGI products, call our Sales Dept at 866-514-2994.

Technical Support

DGI will provide technical support for 90 days after software purchase at no additional charge to the registered user.

E-mail Questions

Ask questions and receive detailed answers from the Technical Support Department. Just send e-mail to the account listed below. You will receive a reply by e-mail.

- support@dgnlink.com

Fax

You can fax questions or comments to DGI. Just send a fax to the number listed below. You will receive a reply by fax.

- **256-417-6640**

Telephone

You can reach DGI Technical Support by phone between 8:30 AM and 5:00 PM Central Time, Monday through Friday.

When you call, please be at your computer, have your documentation in hand, and be prepared to provide the following information:

- The type of computer hardware you are using
- The software version number of Windows
- The serial number and software version number of DGNLink
- Exact wording of any messages that appear on your screen

- What you were doing when the problem occurred
- How you tried to solve the problem

DGI Technical Support Phone Number:

- **256-417-6640**

Support PLUS

To receive unlimited toll-free priority support, be sure to sign up for Support PLUS. The cost is only \$700 per year. As a Support PLUS subscriber, you will also receive access to the update files on our web site and email notifications. In addition, any time we have a major upgrade with new documentation, we will ship these to you automatically at no additional charge. For more details or to sign up, check our web site or call the Support PLUS Subscription Dept at 866-514-2994.

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