CADconform

for Microstation

Administrator and User Guide

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CADconform for Microstation Administrator and User Guide, First Edition

License Agreement

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Chapter 1

Overview

Purpose

CADconform is aimed at organizations that require specific DGN standards for their MicroStation design files. It helps the organization enforce these standards through four distinct methods:

1) Provision of a drafting tool to interactively place features automatically to specification.
2) Generation of report tables detailing which elements fail to match the required standards.
3) A Feature Conforming tool to correct invalid features to the required standards.
4) A certification process to flag design files which have been approved, and warn the user when a certified drawing has been modified.

CADconform stands out in the field of standards correction software in that it is a true server solution. This means that the product was written from the ground up to be available simultaneously for many users distributed across a corporate network, and employs a client-server architecture to enforce privilege based access to reading from and writing to a central DGN standards dictionary. This dictionary is implemented as an ODBC database, allowing a degree of flexibility in the choice of application used to view, edit, import and export data from the dictionary.

Due to the requirement of data being shared amongst different users, CADconform keeps extensive records of which users have performed certain actions. This allows a complete document trail to be maintained automatically. Records are kept of the user name, date and time of any changes to the database, including modifications to Feature Tables, generation of reports and changes made to every design file by Conform.

Additionally, a watermarking facility is provided with the certification process. This allows administrators to have design files automatically stamped with a signed and dated watermark cell representing that the file has been approved. This watermark will automatically invalidate itself if any of the elements in the design file become modified after it was certified.

Compatibility

CADconform is written as an MDL application to run in MicroStation. There are three delivered executables, one each for MicroStation J, v8 and 2004. The CADconform client currently runs on Microsoft Windows NT, 2000 and XP. The server machine can run any Operating System, as long as it is visible to all client machines for purposes of locating the database through ODBC. This means that the server machine could potentially be running Linux, as long as the database file can be seen by mapping a network drive or entering a UNC (Uniform Naming Convention) path.

Note that some functionality in CADconform is only available when running under MicroStation v8. Most of these v8 extensions are only relevant to specific v8 enhancements (such as Text Styles and Models), while some are due to the enhanced functionality of the v8 user interface (such as automatic column sorting in the Report Generator). In either case, functionality that differs between MicroStation J (and below) and MicroStation v8 will be explicitly defined in this manual. On occasion, the manual refers to the “Active Model”. This is analogous to the current design file under MicroStation J.

All CADconform screenshots in this manual are taken from the MicroStation v8 compatible version, since the MicroStation J version is effectively a subset of this version.

System Requirements

CADconform requires the following:

- Operating System – Windows 2000 or higher
- Drafting Software – MicroStation J or higher (Microstation 2004 recommended)
- Database Application – Any ODBC-compliant database application.

Implementation

CADconform is implemented as a MicroStation MDL application, which also uses an ODBC connection to a database to manage its information storage. The Server is used only as a data repository; therefore, there are no network overheads involved in running CADconform when there is no data being read from or written to the database. Additionally, database reads only occur when the database has been modified, because database information is cached locally on each machine.

The Report Generator

The report generation process involves the creation of various report tables in the database. These tables can also be viewed and processed using SQL syntax within CADconform, without the necessity of having a database application installed locally. Reports can be manually or automatically exported to text or comma separated (CSV) files.

Conform

Conform is the tool that changes the symbology of design file elements to match a feature in the Feature Table. It operates similar to a word processor, allowing each error to be viewed, corrected or added to the feature table individually.

Feature Tables

Feature Tables are also implemented as database tables. Feature Tables can be selected, created and deleted using the Feature Table Manager, and edited using the Feature Table Editor.

The User Manager

The User Manager is a simple interface to define user names, passwords and privileges. These settings are stored in an encrypted file located in the “Users” directory on the server machine.

The Drafting Menu

The Drafting Menu allows the MicroStation operator to do feature driven drafting. This means that the correct tool and tool settings are started and set when the user chooses a feature from the Drafting Menu.

Delivered Files

Core Program Files

The following files are delivered and internally used by CADconform:

$(INSTALL_DIR)CONFIG/CADconform.CFG

$(INSTALL_DIR)MDLAPPS/CADconform.MA

$(INSTALL_DIR)MDLAPPS/CADconform.DLL

$(INSTALL_DIR)MDLAPPS/CMLIBWDB.DLL

$(INSTALL_DIR)MENUS/CADconform.MENU

Note: “$(INSTALL_DIR)” denotes the installation path of CADconform.

These files are installed when CADconform’s setup program is executed with the install type set to “CADconform” or “CADconform Server & Client”. CADconform can be installed anywhere on the network, as long as every machine can map a UNC path to it. The path to CADconform and its associated files is stored in a configuration file that is created automatically when CADconform is first loaded, by the CADconform Client Configuration dialog box. See the “Readme.html” file for more details on CADconform installation.

The purpose of each file is listed below:

CADconform.CFG

This file is the administrator configuration file; it globally affects all uses and usually resides in the same directory as CADconform.MA. It is a plain ASCII text file that can be edited in any text editor. It shares the same syntax rules as any other MicroStation configuration file (“*.cfg”).

CADconform.MA

This file is the executable program used to run CADconform. It can be loaded by keying in MDL LOAD CADconform from within MicroStation, or automatically when MicroStation starts by ticking the “Auto-Load” option on the CADconform menu.

CADconform.DLL and CMLIBWDB.DLL

These DLLs are also part of the core program files used by CADconform. Upon first running, they will be cached locally in the Bentley folder under “mdlapps” and “mdlsys\asneeded” respectively.

CADconform.MENU

This file controls the CADconform menu, and is a user definable ASCII text file. The administrator can edit this file to add extra menu items for executing external batch files, controlling CADconform via key-ins, adding site documentation, or performing any allowable MicroStation commands. Instructions for editing this file are contained in the comments at the top of the file.

Run-Time Created Files

The following files are created by CADconform at run time:

$(INSTALL_DIR)USERS/CADconform.PWL

$(MS_DATA)CADconform.RSC

CADconform.PWL

This is the Password List file used for managing the user profiles. The CADconform User Profile Manager generates this file automatically. This file is encrypted and cannot be edited by a user without invalidating the data.

CADconform.RSC

This file stores the last dialog settings used in CADconform, and is created automatically. This file resides in the directory pointed to by the MicroStation configuration variable $(MS_DATA) – which should be a local directory under normal circumstances. By default, MicroStation sets this variable to “C:\Bentley\Workspace\system\data”.

Documentation Files

The following documentation is also provided with CADconform in Acrobat PDF format and HTML respectively:

The CADconform Reference Manual: “CADconform Reference Manual.pdf”
The CADconform Installation Guide and Release Notes: “Readme.html”

The CADconform Menu

The CADconform Menu is loaded on to the MicroStation menu as soon as CADconform is loaded. This menu is built dynamically from the ASCII text file “CADconform.MENU” located in the same directory as the “CADconform.MA” file. The default menu supplied with CADconform contains the following options:

Log In…
Log Out
CADconform Draft
CADconform Toolbox
Watermarking Toolbox
About CADconform…
Help on CADconform
Go to Website
Auto-Start
Auto-Login
Exit CADconform

Log In…

Logs in to the CADconform database. Most of the CADconform commands require the user to log into the database before doing anything else.

Log Out

Logs out of the database and closes all dialog boxes.

CADconform Draft

Opens the CADconform Draft menu without officially logging in to the database. Note that the database will still be attached to enable picking feature tables in Feature Table Manager, but no user login will be associated with the action. CADconform will attach the database that was last logged in to.

CADconform Toolbox

Opens the CADconform Toolbox.

Watermarking Toolbox

Opens the Watermarking Toolbox.

About CADconform…

Opens the “About” dialog box, which displays the version number of CADconform, the license information and the contact details.

Help on CADconform

Opens the CADconform online reference manual.

Go to Website

Opens the system default web browser and the associated web page. By default this is the product web page for CADconform (http://www.cadconform.com), but it can easily be reconfigured to point to any website.

This can be useful if you have an online training document, DGN standards manual or any other web page or document that is frequently visited.

Auto-Start

This menu toggle allows the user to make CADconform automatically load when MicroStation is started. If this option is not ticked, then CADconform has to be loaded manually via the “MDL Applications” option in the MicroStation “Utilities” menu.

Auto-Login

This menu toggle allows the password for the user to be remembered between sessions of CADconform, without the user having to retype the password each time. Note that this is not recommended for administrators, since it effectively gives complete control of CADconform to anyone using the administrators machine.

Exit CADconform

Exits the CADconform application.

Menu Configuration

The CADconform Menu is completely configurable by the user. The menu is built each time CADconform is loaded, from a plain ASCII text file located in the same directory as the “CADconform.MA file”.

Instructions for editing the menu file are contained at the top of the menu file. Each menu item is represented by a line in the menu file. Lines proceeded by a hash character ‘#’ will be treated as comments and ignored. The first keyword on the left before the comma represents the command name, as it appears on screen. The second keywords to the right of the first comma represent the key-in command associated with that keyword. For example, the line:

Log ~In..., CADconform DATABASE CONNECT

Creates a menu entry called “Log In…” that issues the key-in “CADconform DATABASE CONNECT”. The tilde character ‘~’ before the ‘I’ means that the next character is the keyboard accelerator. The keyboard accelerator is denoted by an underline under the letter1. This means that the user can activate the login command by pressing the ‘I’ key on the keyboard if the menu is open.

Any valid MicroStation key-in can be used as the command (for example: “place fence”). Most of the commands in CADconform have a supported key-in; so often-used commands can be added to this menu. To view all of the available key-ins for CADconform:

On MicroStation 2004,

1) Select the MicroStation menu item “Utilities > MDL Applications”.

2) Select “CADconform” from the list of loaded applications.

3) Press the “Key-ins…” push button. MicroStation v8 only: Simply press the “Browse Key-in Tool” to view all key-ins.

[1] [1] Under Windows 2000/XP the underline may not be visible until the ALT key is pressed. This can be controlled by the “Control Panel > Display > Effects. > Hide Keyboard…” toggle.

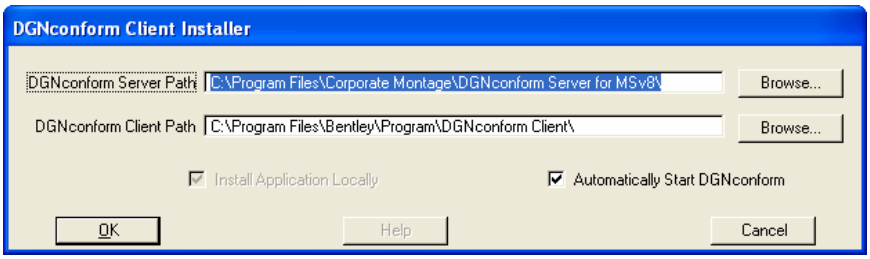
Chapter 2

The Client Configuration Dialog

The CADconform Client Configuration Dialog appears when CADconform is first installed, and additionally whenever there is a vital resource that cannot be located (such as the server machine). It can be manually opened from the “Login” dialog box by left-clicking the “Configure” push-button. It can also be opened at any time by keying in:

CADconform SETTINGS CLIENT

The Configuration Dialog has the following options:



CADconform Path

This is the full path to the installation directory of CADconform. From a client machine, it should begin with the UNC path to the server machine, followed by a share-name path to the install directory. For example:

\\WIN2KSERVER\SHARED_APPS\CADconform Server for MSJ\

This would look for CADconform on a machine on the network called “WIN2KSERVER” with a shared directory called “SHARED_APPS”.

CADconform will look in the specified path for a file called “CADconform.MA” within a sub-directory named “MDLAPPS”. If this file is not found, then the path has been entered incorrectly and the configuration will not continue. If CADconform was installed by the installer, then this path should be set correctly by default.

CADconform Client Path

This represents the full path of where the client will install itself automatically. This should not normally be changed by default unless the “Bentley\Program” path is not local or writeable.

Automatically Start CADconform

This toggle allows the user to specify whether CADconform starts automatically in every workspace or not. This is analogous to the “Auto-Start” option in the CADconform menu.

OK

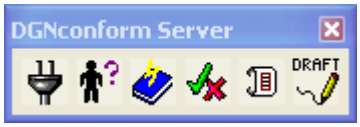
After pressing “OK”, CADconform will write out a local configuration file in the directory:

($MSDIR)\config\appl\CADconform_server.cfg

This file contains all the settings necessary for CADconform to find the server machine.

Chapter 3

The Toolbox Interface



The CADconform Toolbox

All of CADconform's tools are activated from the CADconform toolbox. The toolbox opens automatically after the user has loaded the application and logged on to the database.

From left to right, the tools are:

Connect to Database
Manage Users
Feature Table Editor
Conform
Report Generator
Drafting Menu

Note that the CADconform Toolbox will have a different number of icons depending on the privileges of the person logging in. If the user does not have the "Edit Dictionary" privilege, then the "Edit Feature Table" icon will not be available. Similarly, the "Conform" and "Generate Reports" icons will only be available to users with these respective privileges.

The following chapters list the purpose of every item on the user interface, in order of each tool (left to right) on the main toolbox.

Chapter 4

Connecting to the Database

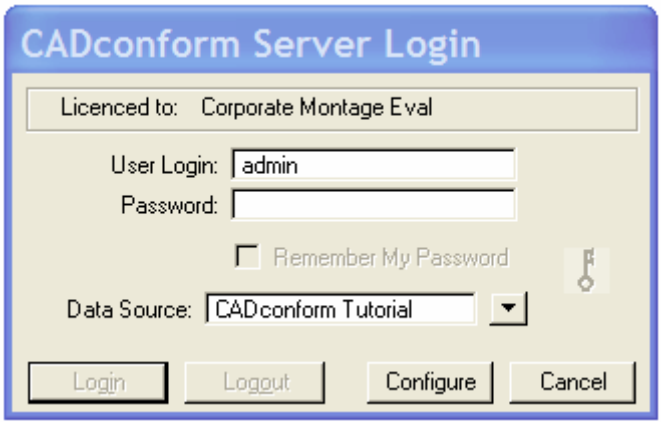
The “Connect to Database” icon on the CADconform toolbox allows the user to log on to a specific database as a specific user. Starting this command will open the CADconform Login window. This window opens up automatically when the toolbox is initially opened, or the “Log In…” command is chosen from the CADconform menu. This tool can be used after start-up to either:

Log out if you are finished.
Log in as a different user.
Log in to a different ODBC Data Source.
Toggle between stored passwords.
Configure the CADconform Server and Client paths
Open Windows Explorer at the directory containing the File DSN definitions.

By default, CADconform is delivered with a tutorial database called “CADconform Tutorial”. Upon first starting CADconform, this Data Source Name (DSN) will appear as the only selection in the “ODBC Data Source” combo box. More data sources can be added by adding File DSN data sources to the “File DSN” directory of the CADconform install path. To browse to the File DSN directory, press the “?” icon. If you wish to connect to an existing User or System DSN instead of a File DSN, then enter the DSN in the “Data Source” text field.

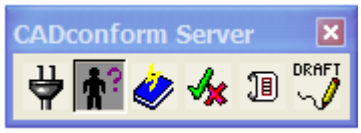
If the user is logging in to the database for the first time then no users will be set in the User Manager and the login name and password will be ignored. It is presumed that the first person to log in to a database will be the administrator, and hence the “User Manager” dialog box will automatically open. Once the User Manager has created the Users Table, the Login dialog box will again open so that the administrator can define which person in the User Table they are.

Depending on the privileges set for each user, some of the dialog items may be disabled. For example, the administrator can lock a certain user into a specific Data Source, or disallow a user from opening the Server Configuration dialog box. If a privilege is not granted, then the dialog item is greyed out.

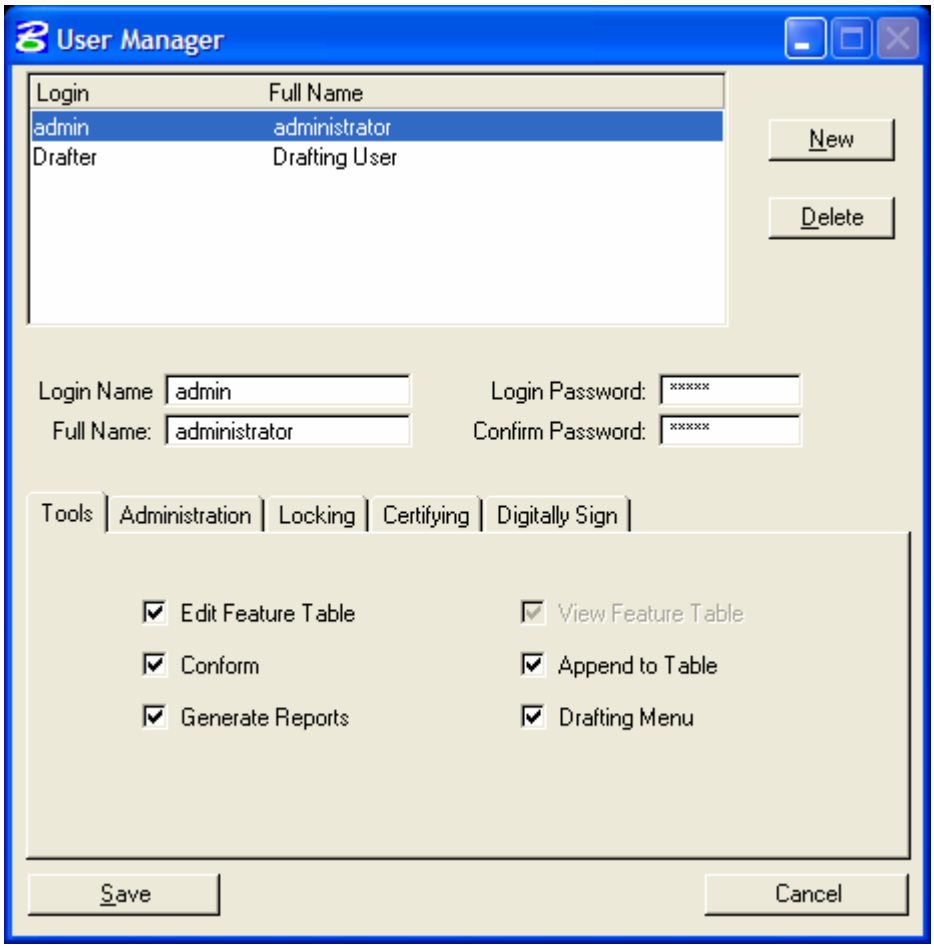


Logging in to the database

Chapter 5: The User Manager



The User Manager allows the database administrator to define who can use CADconform and which tools are available to them.



The User Manager can be used to:

Create New users
Delete existing users
Set or modify user passwords
Set privileges

The privileges that can be set for each user are spread across five tab pages:

Tools
Administration
Locking
Certifying
Digitally Sign

Additionally, each user's login, full name and password can also be entered by the administrator.

If the user does not have the "Administrator" privilege, then the only function they can perform with the User Manager is to change their own password.

Dialog Items

Login

This is the unique name assigned to represent each user. This is analogous to the Windows login name, and a similar system can be used for CADconform as is used for your corporate network.

Full Name

This is the users full name (first name, last name). It is used internally for Reporting.

Login Password, Confirm Password

Passwords are used to validate that the correct person is logged in to CADconform with the appropriate privileges. Passwords are defined twice to protect the user from a typographical error being entered unknowingly. The password can be any number of alphanumeric characters, and should be kept private. The password does not expire, but can be redefined by the user at any time.

New

Adds a New User to the User Profile. By default, the new user will inherit the privileges of the currently selected user in the User Profile list-box.

Delete

Deletes the currently selected user from the User Profile.

Tab Pages

The tab pages at the bottom half of the User Manager dialog define the privileges for each selected user in the list box. These privileges are defined in more detail below.

Tools

The Tools options define which tools each user has available to them. These include:

Edit Feature Table
View Feature Table
Conform
Append to Table
Generate Reports
Drafting Menu

Edit Feature Table

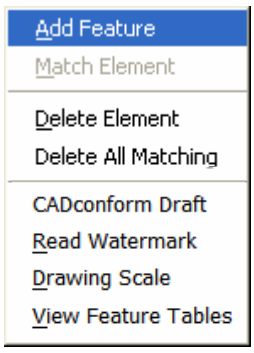
A user with the Edit Dictionary privilege can open the Feature Table Editor. This means that the user has full control over the DGN standards database. This privilege is effectively an administrator privilege, and should not be granted to many users. Users without the Edit or View Feature Table privilege will not see the Feature Table Editor icon on the CADconform toolbox, even if they have the “Administrator” privilege (see above).

View Feature Table

A user with the View Dictionary privilege can also open the Feature Table Editor, however they only see it in “read-only” mode. This means that most of the feature modify tools are disabled (greyed out). This tool is useful so that drafters can see what the DGN standard has been defined as for each feature.

Conform

“Conform” allows the user to open the Feature Conform tool to fix invalid design files. If the user does not have this privilege, then the Conform icon will not be visible on the CADconform toolbox.



Append To Table

The “Append to Dictionary” privilege allows a user running the Conform tool to automatically add an unmatched feature to the current Feature Table. This privilege should be used sparingly and only usually at the beginning of a project, since it allows the user to circumvent the correction of invalid features. The append privilege appears as the “Add Feature” menu command in the Conform dialog’s “Tools” menu. This menu option will be disabled for user’s who do not have this privilege.

Generate Reports

Allows the user to open the Report Generator. Users without this privilege will not see the Report Generator icon on their CADconform toolbox.

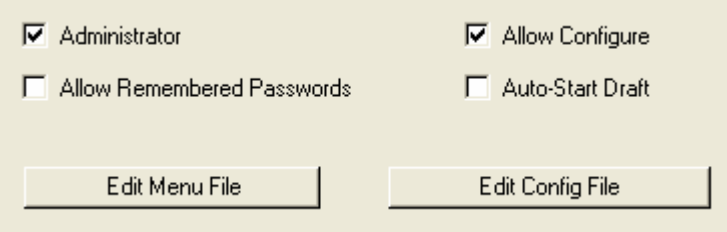
Drafting Menu

This privilege allows users to draft to the DGN standard using a menu built automatically from the feature table. Any user involved in the drafting process should be granted this privilege. It appears as an icon on the CADconform toolbox. If this privilege is not granted, then the user will not see the “CADconform Draft” icon on the main toolbox.

Administration

The Administration options define various privileges and command related to CADconform’s operation.

Administrator
Allow Configure
Allow Remembered Passwords
Auto-Start Draft
Edit Menu File
Edit Config File



Administrator

The administrator privilege allows the user full control over user privileges for CADconform. There will usually only be a small number of administrators in any organization. The administrator is the only user who is allowed to edit the User Profiles. Users who do not have this privilege will only be allowed to change their own password, and all other options on the User Manager dialog box will be disabled. Additionally, an administrator can overwrite a report in the Report Generator, even if the report belongs to someone else.

Allow Configure

If the “Allow Configure” privilege is granted, then the user will be able to open the “CADconform Client Configuration” dialog box from the “Configure” button on the login dialog box. This allows the user to modify the path to CADconform on the server machine. Normally, users shouldn’t be able to do this.

Allow Remembered Passwords

This option allows the administrator to decide which users are allowed to have locally stored passwords. Typically, an administrator might decide that only users with low privileges (e.g. can only draft or report) are allowed to have remembered passwords, while other users can not. A more security conscious administrator might decide that no one is allowed to have remembered passwords, while another may grant this privilege to all users to save time at log on.

In general it is safe for most drafters to have this option enabled, however it is not recommended for administrators, since it means anyone with access to the administrator machine could potentially modify the dictionary or user profiles.

Auto-Start Draft

This option makes the Drafting Menu open up automatically when the user logs in to the database.

Usually, the Feature Table Manager (FTM) will display, allowing the user to choose the feature tables to draft to. If the feature tables are locked (see below), then the FTM will not display.

Edit Menu File

This command button opens the menu file “CADconform.MENU”, which controls the “CADconform” pull-down menu on MicroStation’s menu bar. Note that MicroStation needs to be restarted after modifying this file in order for the changes to take affect. The configuration file will open using the application Windows has associated with the file type “MENU”, for example, “Notepad”. If no file types are associated with this extension, then Windows will open the “Open With…” dialog to allow the user to choose which application to use. If no Text Editors have been installed by the user, then Windows Notepad will suffice.

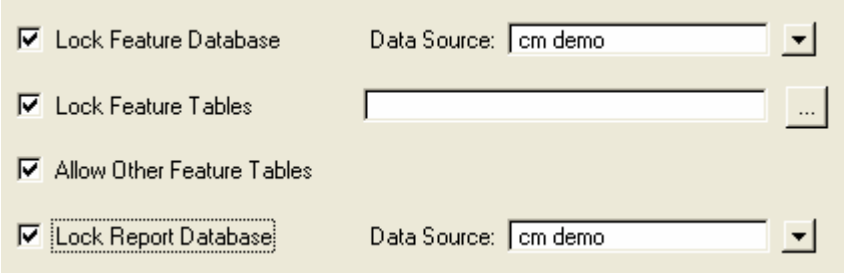
Edit Config File

This command button opens the administrators configuration file “CADconform.CFG”, which controls the behaviour of CADconform for all users. Note that MicroStation needs to be restarted after modifying this file in order for the changes to take affect. The configuration file will open using the application Windows has associated with the file type “CFG”, for example, “Notepad”. If no file types are associated with this extension, then Windows will open the “Open With…” dialog to allow the user to choose which application to use. If no Text Editors have been installed by the user, then Windows Notepad will suffice.

Locking

The locking options allows the administrator to lock all (or specific) users into using defined databases or feature tables. The options are:

Lock Feature Database
Lock Feature Tables
Allow Other Feature Tables
Lock Report Database



Lock Feature Database

This allows the administrator to lock certain users into using a particular Data Source Name (DSN). This can be useful if an organization has several different databases for different departments. A user who has this privilege set will not be able to log in to any other database than the one defined by the Data Source combo-box field to the right of this privilege.

Lock Feature Tables

This option allows the administrator to lock users into using specific feature tables within a chosen database. If a feature table is locked, then it will always be opened automatically when the user opens any tool that requires feature tables. The one exception to this rule is opening the Feature Table Editor in read-write mode (requires the Edit Feature Table privilege). Locked feature tables will appear in the Feature Table Manager with a lock icon to the left of the name in the list box. The feature tables to lock can be defined by either typing in their names (separated by commas) in the text field, or chosen by clicking on the "…" button to the right of the text field.

Allow Other Feature Tables

This option is only applicable to users who have been locked into using specific feature tables. If this privilege is granted, then the user will be able to open other feature tables in addition to the locked tables. If the user does not have this privilege, then the Feature Table Manager is never displayed, since they can't select or deselect any tables in the FTM anyway.

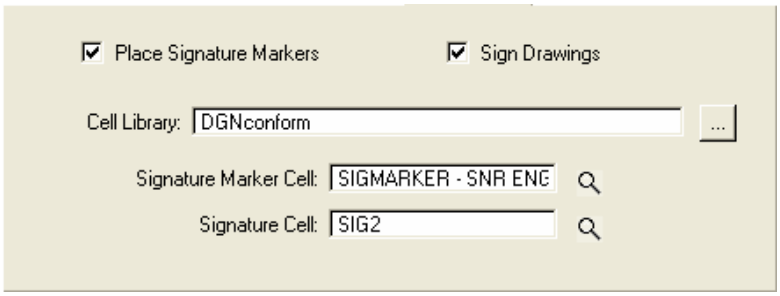
Lock Report Database

This allows the administrator to lock a particular user into using a specific database for report generation. Reports will automatically be stored in the database named in the Data Source text field to the right of this privilege. If this toggle is OFF, then the user can choose to write their reports to any database. This option affects the enabled status of the "Data Source" combo-box at the bottom of the Report Generator dialog box.

Watermarking

The watermarking options define the cells used for marking and watermarking design files during certification and reporting. This is an optional component of CADconform, and may be ignored if not required. The options on the dialog box are:

Place Certify Markers
Certify Design Files
Cell Library
Marker Cell
Watermark Cell



Place Certify Markers

If this toggle is ON, then the user will be prompted to place marker cells in every design file they open that does not either:

1) Contain a marker cell

2) Contain a watermark cell

3) Contain a marker cell in a referenced file

Marker cells are an important part of the watermarking process, as they define the location and scale of the watermark when a file is certified.

Certify Design Files

The Certify privilege is designed to allow some users the ability to certify valid design files as having been checked and approved by the user. During certification, certain levels are turned off and a Watermark stamp is placed in the design file to signify approval. Users who have this privilege must also have the “Place Certify Markers” privilege.

Cell Library

This text field defines the path to the cell library used to define the marker and watermark cells. It can be defined by using the browse button denoted by ellipses “…”. If the path is relative to a path defined by the workspace configuration variable “MS_CELL”, then only the file name is stored; otherwise the full path of the cell library is stored.

Marker Cell

This text field defines the name of the marker cell. It can be picked by using the locate icon to the right of the text field.

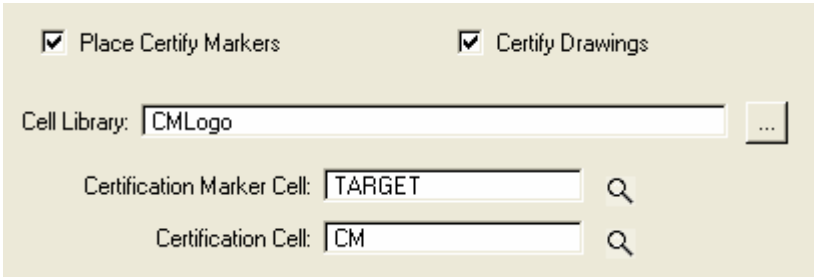
Watermark Cell

This text field defines the name of the watermark cell. It can be picked by using the locate icon to the right of the text field.

Digital Signatures

The digital signature options define the cells used for marking and signing design files. This is an optional component of CADconform, and may be ignored if not required. The options on the dialog box are:

Place Signature Markers
Sign Drawings
Cell Library
Signature Marker Cell
Signature Cell



Place Signature Markers

If this toggle is ON, then the user has the privilege required to place signature marker cells. The marker cells are an important part of the digital signature process, as they define the location and scale of the signature when a file is signed.

Sign Drawings

The Sign Drawings privilege is designed to allow some users the ability to sign design files as having been checked and approved by the user. During signing, a Signature stamp is placed in the design file to signify that the drawing has been signed. Users who have this privilege must also have the “Place Signature Markers” privilege.

Cell Library

This text field defines the path to the cell library used to define the signature marker and signature cells. It can be defined by using the browse button denoted by ellipses “…”. If the path is relative to a path defined by the workspace configuration variable “MS_CELL”, then only the file name is stored; otherwise the full path of the cell library is stored.

Signature Marker Cell

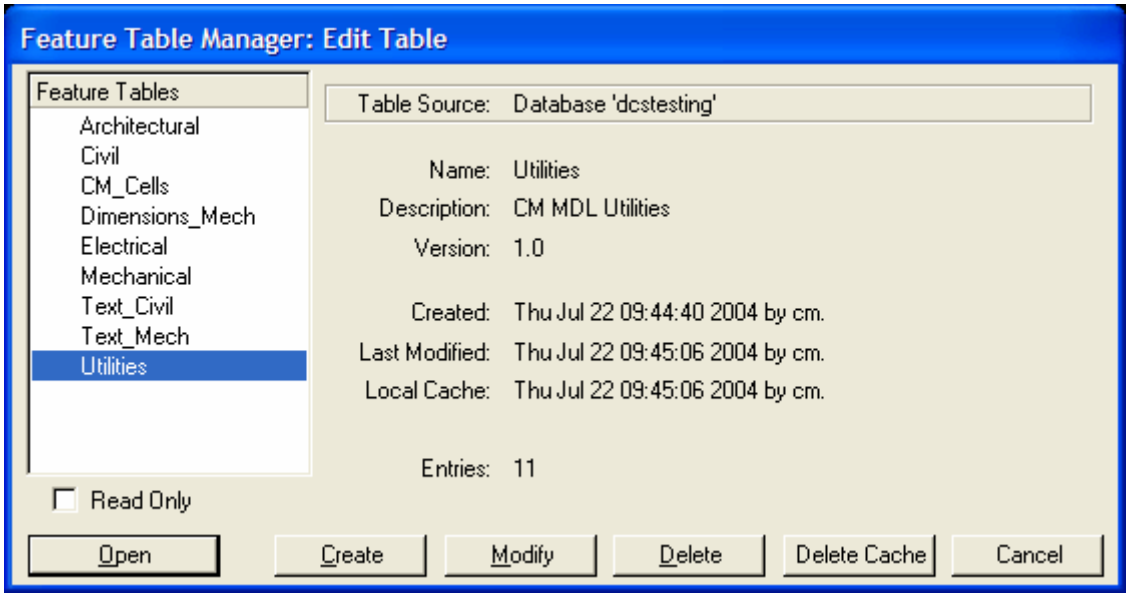
This text field defines the name of the signature marker cell. It can be picked by using the locate icon to the right of the text field.

Signature Cell

This text field defines the name of the signature cell. It can be picked by using the locate icon to the right of the text field.

Chapter 6

The Feature Table Manager (FTM)



The Feature Table Manager opens whenever CADconform requires the user to select one or more Feature Tables. If the user is logged on as and has the privilege to “Edit Feature Tables,” then the user will be able to create, modify or delete Feature Tables using the “Create”, “Modify” and “Delete” buttons on the dialog box. It also allows users to delete cache files for tables by using the “Delete Cache” button. If the user does not have the privilege to “Edit Feature Tables,” then these buttons will not be available. They will also be unavailable if they are opening any tool other than the Feature Table Editor, or the “Read Only” toggle is toggled “ON”.

Most tools that use the Feature Table Manager allow multiple Feature Tables to be chosen. The Feature Table Manager supports multiple selection through the standards Windows <CTRL> and <SHIFT> keys, and also by drag selection with the mouse. The exception to this is the Feature Table Editor, which only allows the user to edit one Feature Table at a time.

CADconform maintains a document audit of all changes made to Feature Tables in the database. For every Feature Table created, there are two tables created in the database. The first table contains the feature data of the Feature Table itself, and the second table tracks all modifications to this Feature Table. For example, if a table is created called: “Example1”, then there will be two tables created in the database: “Table_Example1” and “iTable_Example1” (Note the preceding ‘i’ in the latter name). The latter table contains information on the Feature Table, which is summarized in the Feature Table Manager when a Feature Table is selected. This information includes the creator, the version number of the table (see Delete Cache on page 25), the time and date of creation, the number of entries in the Feature Table and the time, date and name of the last person to modify the Feature Table. Individual modifications to the Feature Table can be viewed by opening the information table within a database program, such as Microsoft Access.

Additionally, a local cache of each Feature Table is created upon first opening, or when a modification to an existing table is detected. This cache file is then used instead of downloading from the database each time. Cache files are created in the local "Bentley\Program\CADconform Client\Cache" directory, and are in the standard CADconform DICT file format, which means they can also be imported using the Feature Table Editor. The creation date of the local cache file will also be displayed along with the other information for each table. If this date is older than the last modification to the table, then CADconform will prompt the user if they want to download the latest version.

The buttons on the Feature Table Manager (FTM) are described below:

Open

Opens the selected feature table(s).

Create

The 'Create' button allows the user to create a new database. The database can be created either as a blank table, or imported from an existing dictionary (DICT) file. Creating a new table requires the definition of a table name and optionally a table description. Note the limitations on table names mentioned below.

Delete

The 'Delete' button deletes the currently highlighted table. This command can not be undone, and only one table can be chosen at any time.

Modify

This allows the user to modify either the name or the description of an existing feature table.

Delete Cache

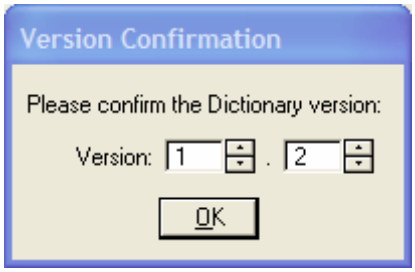
The "Delete Cache" button allows users to delete the cache file of the selected table. This will force CADconform to reload the table from the database instead of using the local cache files. This will slow the loading process but may be useful if any changes are ever made directly to the database.

Cancel

Cancels the current command; either open, create, modify or delete.

Table/Dictionary Versioning

CADconform stores a version number for all tables when the configuration variable _CADconform_USEDICTVERSIONING is turned on. This version number consists of major and minor release numbers and is displayed as: "major.minor" (e.g. Version 1.2). This allows the administrator to track the changes made to tables. Administrators will be prompted to update the version number by the dialog below when any changes are made to the Feature Table. Table/Dictionary version numbers can also be used as replacement text in watermarks to signify which dictionaries and versions a drawing has been certified against.



Limitations on table names

Note that there are certain limitations imposed upon the naming of Feature Tables. The database server used by CADconform does not allow table names that contain a space, punctuation, or any other non-alphanumeric character. Feature Table names should be kept short (under 32 characters). The description field can optionally be used to enter more detailed information about the Feature Table.

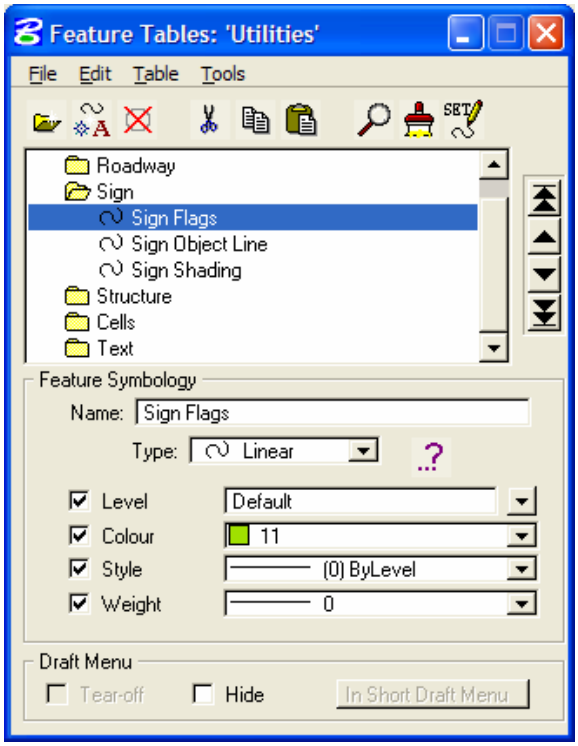
Chapter 7

The Feature Table Editor (FTE)

The Feature Table Editor allows a DGN standards administrator to define the allowable features that are in the Feature Tables. In order to edit a Feature Table, you must first select the Feature Table using the Feature Table Manager. This dialog box opens automatically whenever the Feature Table Editor, the Report Generator, Conform or the Drafting Menu is run.

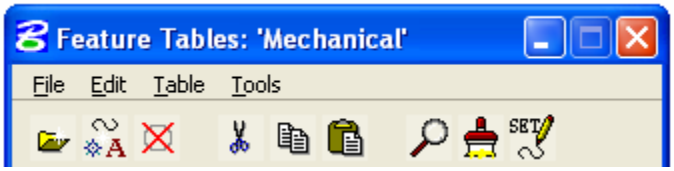
When the Feature Table Editor is opened, the selected Feature Table is automatically imported from the database. If the Feature Table is empty, then the only entry in the Feature Table will be the table name.

The Feature Table is organized into a group-based hierarchy, similar in concept to files and directories in Windows Explorer. Features can be organised into groups, which can then be children of larger groups. For example, a feature called “Lakes” can be a member of a group “Water Features” which can be a child of a larger group “Mapping”. Water Features can then contain other features, such as “River” and “Swamp”. As with Windows Explorer, groups can be expanded and contracted by double-clicking with the left mouse button.



Icons

The icons on the Feature Table Editor dialog box are shortcuts to the equivalent menu bar commands. The purpose of each icon is listed below:



New Group

This icon inserts a new Feature Group underneath the currently selected group. This group will have a default name inherited from the parent group, which can then be modified. New groups at the top level will inherit the name “New Group”. To add a new group at the top level, select the top most list-box row, which is the Feature Table name.

New Feature

This icon inserts a new feature underneath the currently selected group. If a feature is selected, then the new feature will be a copy of the selected feature. Otherwise, it will inherit the active settings for the feature type.

Delete

The icon deletes all selected features and groups. If a feature group is not empty, then you will be prompted whether you wish to delete all contents of this group. More than one feature can be selected for use with the delete tool.

Cut

This icon tags the selected list box rows as “Cut”. List box rows that are tagged will be displayed in red. This tool is used in conjunction with the “Paste” icon. When a list of rows is pasted, the rows tagged as “cut” will be deleted.

Copy

This icon tags the selected list box rows as “Copied”. List box rows that are tagged will be displayed in red. This tool is used in conjunction with the “Paste” icon.

Paste

This icon pastes the currently tagged list box rows into the location of the currently selected row. If the tagged rows were “Cut”, then the tagged fields will be deleted after the paste operation.

Locate Tools

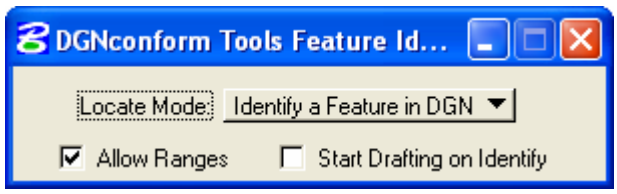
This icon allows the user to locate existing MicroStation elements from the design file. Three operation modes can be used with this tool:

Identifying Features
Adding Features
Finding Features
Reading Watermarks

Identifying Features

To identify which feature an element matches:

1) Start the Locate Feature tool and set the mode to “Identify a Feature in CAD”.
2) Select the element and accept it with a data-point.
3) The matching feature will now be selected in the Feature Table Editor.

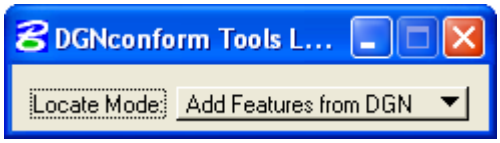


If the “Start Drafting on Identify” toggle is turned ON, then the user key-in associated with this feature will automatically be started when the identified element is accepted with a data-point. This can be convenient when you want to start drafting in an area with existing elements that match the feature you wish to draw. A warning dialog box will appear if the selected element does not match any of the features in the current Feature Table.

Adding Features

To add new features to the Feature Table from design file elements:

1) Start the Locate Feature tool and set the mode to “Add Features from CAD”.
2) Select the element to add and accept it with a datapoint.
3) The new feature will now be added to the Feature Table.

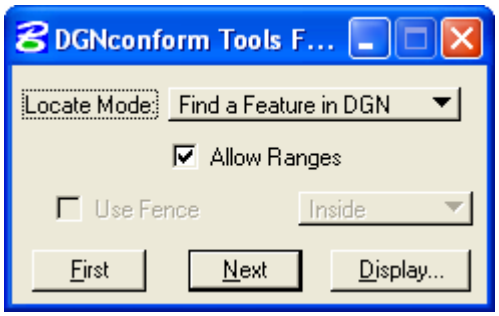


Note that this tool supports selection sets to add multiple elements in one hit. To use a selection set, the elements must be selected before the Add Feature tool is activated. By default, this tool will not add features to the Feature Table if they match any existing features.

Finding Features

To find every instance of a particular feature in the current design file:

1) Start the Locate Feature tool and set the mode to “Match a feature From CAD”.
2) Select the Feature to be changed in the Feature Table list-box.
3) Click “First” to find the first instance of the element in the design file.
4) Click “Next” to find each successive instance of this feature.
5) Click “Display” to change the display options for displaying the feature, or “Use Fence” to restrict the search by the active fence.



Reading Watermarks

The Read Watermark tool will extract the information associated with a chosen watermark, and display it to the user. See the Read Watermark tool for more information.

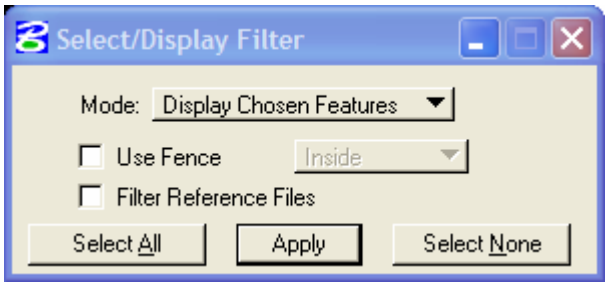
 Filtering Tools

This icon is used to filter the view display or selection set for certain features. This makes locating undefined features a lot easier. Unlike the other icon commands, the Filter tool behaves like a toggle. To turn the filter OFF once it is ON, simply click the icon again. The icon imagery changes to reflect the current status, i.e.

 Means the filter is currently OFF.

 Means the filter is currently ON.

When this command is activated, the Select/Display Filter dialog will open with the follow options:



Use Fence

This option will restrict the filter by the active fence. It will be disabled if there is no active fence.

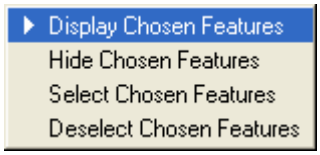
Filter Reference Files

This option will include the reference files in the filtering process when turned on.

Mode

Mode changes the filtering command. The options are:

Display Chosen Features
Hide Chosen Features
Select Chosen Features
Deselect Chosen Features



Display Chosen Features

This will only display the currently selected features, or all of the features inside a group if a group is selected. This choosing the top row of the FTE will display all elements in the master design file that have matches in the current feature table.

Hide Chosen Features

This is the opposite of the “Display Chosen Feature” option; it will hide only matching features. This can be particularly useful to get a quick overall preview of what doesn’t conform to your standards in the active design file, assuming that the current feature table contains every allowable feature.

Select Chosen Features

Instead of toggling the display of features, the select commands will toggle the selection of matching features. This can be useful if you want to perform further processing of elements via a selection set. For example, you could copy all piping to the clipboard simply by selecting the “Piping” feature, applying this command and pressing <CTRL>-C.

Deselect Chosen Features

This is the opposite of the “Select Chosen Features” mode; it will remove matching features from the current selection set. This can be useful in combination with the MicroStation PowerSelector tool, since you can create a selection set of nonconforming elements by pressing “Select All” and then removing the conforming elements by applying this tool mode.

Select All

Select All is simply a short-cut for the MicroStation “Edit > Select All” command. It selects all elements in the design file. This is useful in combination with the Select/Deselect filtering modes.

Select None

Select None is a shortcut for the MicroStation “Edit > Select None” commands.

Apply

Apply will apply the current display filtering options to the active view. If Hide or Display is chosen, then the view will be updated accordingly. If the Select/Deselect mode is active, then the active selection set will be updated.

Settings Tools

The Set Features tool allows the user to set either selected features or selected design file elements, according to the settings mode. Each of these commands requires a datapoint to accept the operation. The available commands are:

Set Active Symbology by a Feature
Set Features by Active Symbology
Set Elements by a Selected Feature
Set Features by Selected Element

Set Active Symbology by a Feature

This command will set MicroStation’s active symbology and settings to match the selected feature. For example if a text feature is chosen, then CADconform will set the active font, text size, colour, level and any other setting defined by the feature.

Set Features by Active Symbology

This command will change the symbology of all selected features to match the active symbology and settings.

Set Elements by a Selected Feature

This command will change the symbology of all selected elements to match a selected feature. This can be a convenient way to change an element from one feature type to another.

Set Features by Selected Element

This command allows a selected feature to be redefined to match a selected design file element.

The Feature Symbology Options

Once a feature has been added (either manually using “New Feature” or automatically via “Locate Feature”) the Feature Symbology Options can be set to uniquely define it. Note that matching on each symbology type (level, colour, weight, etc) is optional. For example, if line work on a particular level can be any colour, then the colour toggle should be OFF. The Feature Table Editor offers the following options for every feature:

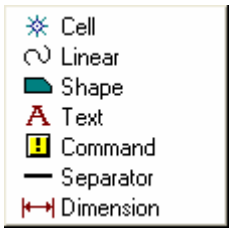
Name

This is the descriptive name of the feature. It is currently limited to less than 64 characters in length.

Type

This defines which of the seven categories of element the feature belongs to. The seven categories are:

Cell
Linear
Shape
Text
Command
Separator
Dimension



Command and separator types are used exclusively by the Drafting Menu, and are not discussed in this section.

Level

Level defines which MicroStation level the feature should be on. On MicroStation J or below, the level will be a number between 1 and 63, whereas under MicroStation v8 it can be any alphanumeric name.

If a feature can be on a number of levels, then a feature will have to be added for each level (e.g. “Annotation for Level 7”, “Annotation for Level 8”, etc). This makes it easier to correct an invalid level when using CADconform, since the user will have a list of choices to change the level to. Alternatively, ranges can be specified using the “Symbology Ranges” options.

For Graphic Cells spanning multiple levels, the level represents the lowest (or “Base”) level.

Colour

Defines the MicroStation line colour for the feature. Under MicroStation v8, “By Level”, Background Colour and “By Cell” (for DWG files) are also available options for colour. Note that this is the element colour as stored in the element, which is not necessarily the colour that the element displays with. Level symbology and text styles can override the element colour. This option may be disabled for cell features defined as “Graphic Cell” types.

Weight

Defines the MicroStation line weight for the feature. Under MicroStation v8, “By Level” and “By Cell” are also available options for line weight. This option may be disabled for cell features defined as “Graphic Cell” types.

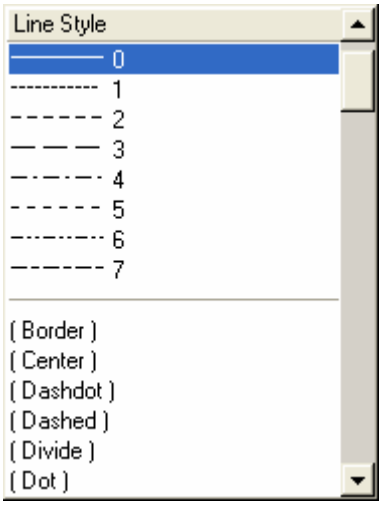
Style

Defines the MicroStation line style for the feature. Custom linestyles are fully supported in both the MicroStation J and v8 versions of CADconform, however the method of choosing them differs slightly. Under MicroStation v8, custom linestyles can be chosen directly from the line style combo-box. If a custom linestyle is required on MicroStation J or below, follow these steps:

1) Set the required custom linestyle as the active MicroStation linestyle.

2) Select “Custom” from the Feature Symbology “Style” combo box.

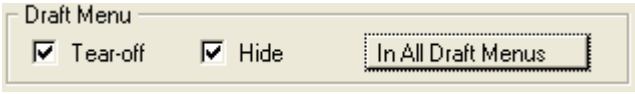
MicroStation v8 allows the user to choose custom linestyles directly from the combo box.



Choosing a custom line style in MicroStation v8

Draft Menu Options

The draft menu options only affect the way features are displayed in the Drafting menu. They have no effect upon defining the feature itself, nor how it is matched, conformed or drafted.

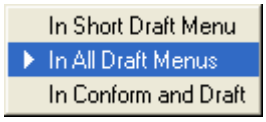


Draft Menu: Tear-Off

This option is available only to feature groups. It allows a group to appear in its own menu on a separate Draft dialog box. This only affects the appearance of the feature in CADconform Draft. Groups set to open as New Menus will display in the Feature Table Editor and the Draft list-box as dark green.

Draft Menu: Hide

This option only affects the appearance of the feature in the Draft menu. If hide is ON, then the feature will not appear in the either the Draft menu bar or the draft list-box. There are three settings for Hide:



- Hide in Short Menu
- Hide in All Draft Menus
- Hide in Conform and Draft

“Hide in Short Menu” will only hide the feature if the current menu length is set to “Short Menus”. If the menu type is set to “Long Menus” the feature will appear in the drop down menu, but will appear magenta (but still selectable) in the list-box interface mode. Note that if a group if hidden, then all features in that group will also be hidden from Draft menu.

“Hide in All Draft Menus” will hide the feature or group in ALL menus and list-boxes in CADconform Draft. This is intended as an administrator option to define features that are allowable but never drawn by the draftsperson. This may include the certification cell and it’s associated invalidation cross, or any obsolete DGN features that are allowable but deprecated in use. Features hidden in all menus will not appear in Draft, but will still appear as legitimate features to “Change To” in Conform.

“Hide in Conform and Draft” will hide the feature (or group of features) in ALL draft menus as well as Conform, which means that an invalid feature can not be changed to the hidden feature because it will not appear in the “Change to…” list box. This can be useful for allowing bad features from legacy DGN drawings, whilst discouraging the draftsperson from creating or correcting to these features.

The Feature Type Options

The Feature Type Options are options specific to the Feature Type (cells, linear, text, shape, command or dimension). This includes options such as text size, cell name and fill colour. The Feature Type Options dialog box can be opened in three ways:

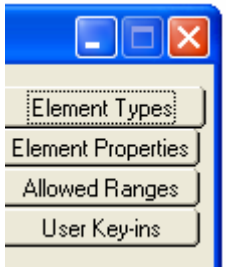
1) By double-clicking on a feature in the list box.

2) By choosing from the menu: “Edit > Feature Options”.

3) By clicking on the “Feature Options” icon.

Some Feature Options are common to multiple feature types:

Element Types
Element Properties
Allowed Ranges
User-Key-ins

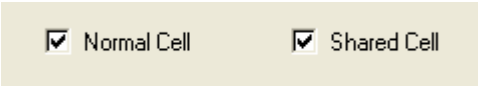


Tab page options

These options are available as tab pages on the Feature Type Options dialog boxes.

Element Types

Element types represent the allowed types for a particular feature. For example, a concrete object line might be allowed as a line (type 3), line string (type 4) or shape (type 6). Element types are usually all on by default for each feature, and are mutually exclusive.

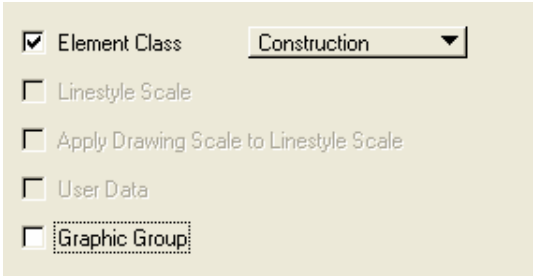


The Element Type Options for a Cell

Element Properties

Element properties contain match criteria for generic properties of an element. This includes:

Element Class (primary, construction, dimension, etc.)
Custom Linestyle Scale
Whether the custom line style scale inherits the active drawing scale.
Graphic group



Element Class

Element Class is stored for each design file element, and is displayed in the "General" tab page of MicroStation's "Analyze Element" tool in MicroStation v8, or the main Analyze dialog box in v7 or below.

Most elements have an element class of "Primary" or "Construction", but special element types automatically set a different class. For example, dimensions are set by default to "dimension" class (including dropped dimension primitive elements), and non-associative patterns have their class set to "Pattern Component".

Linestyle Scale

Note that Custom Linestyle Scale is only available if the user is matching on "Style" and the style has been set to a custom linestyle. If matching on linestyle scale is active, then a third option becomes available: "Apply Drawing Scale to Linestyle Scale". This is similar to the Cell Scaling options, it defines whether the active drawing scale will be taken into account when drafting, conforming, reporting or matching this feature.

For example, assume that the linestyle scale is set to match on "1.000", the "Apply Drawing Scale to Linestyle Scale" is turned "ON" and the current drawing scale is set to "50.000".

If a user was running Conform and an element was found that matched our feature but had a linestyle scale of "1.000" it would be rejected, and the reason would be reported as having an incorrect scale (50 expected). Correcting the feature by clicking "Change" would then scale the linestyle up by a factor of 50 because we had told CADconform to apply the drawing scale to the perceived linestyle scale.

Graphic Group

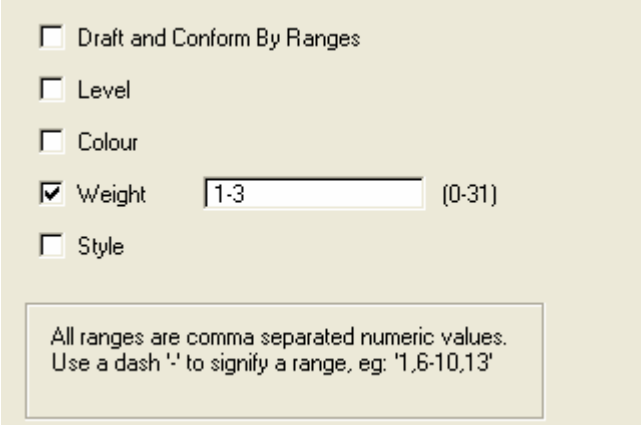
This is not normally useful for MicroStation users, since graphic groups are automatically allocated by MicroStation, and will vary between files. However, some 3rd party MDL applications use unique graphic groups as a method of identifying features as belonging to a certain type. The graphic group option can be useful for design files that adhere to this methodology. Defining the graphic group for a feature will affect the way it is interpreted by Conform and Report, as well as the way it is placed by the Drafting Menu.

Allowed Ranges

This tab page allows us to define a range of allowable values for matching symbology. This is particularly useful for correcting legacy drawings where the DGN standards have

either changed or not been adhered to. These ranges are only applied when CADconform is looking for a match, not when drafting or conforming.

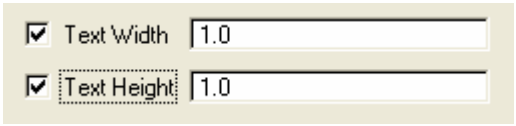
For example, a concrete object line might have to be drawn on level “Concrete” with a colour and style of zero and a weight of one, but we might also allow weights of two and three in certain circumstances. To achieve this, we would set the weight to one as normal, but set our symbology ranges such that “Weight” is ticked, and a range is entered of “1-3”.



Setting the weight range for a Concrete Object Line

Note #1: custom linestyles are not supported in symbology ranges. If more than one custom linestyle is allowed for a feature, then multiple features for each style will be required.

If the feature is of type “Text”, then ranges can also be specified for text width and text height. This can be useful for P&IDs, for example, where the text width has been stretched or shrunk slightly to fit inside a callout bubble.



User Key-ins

All six of the Feature Type Option dialog boxes have settings for the “User Key-in”2. The key-in is the command that is associated with drawing that feature, and is only used by the Draft menu. For example, to draw a feature called “Concrete Line”, the key-in might be set to “place smartline”. This means that the smartline tool is automatically started every time you want to place a Concrete Line. Multiple commands can be entered as long as they are separated by a semicolon.

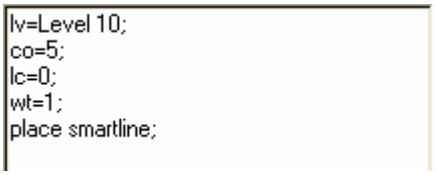
User key-ins are built automatically from the Feature Table Editor’s dialog settings, and a default drawing command is associated to every feature created depending on its feature type. In addition to this, there are a number of tools provided for recording key-ins directly. This works by converting any command activity driven by the user, into its equivalent key-in command. These tools are described below, from left to right.

Key-in Tools

Record

[2] See the MicroStation manual for more information on available key-ins.

Begins recording key-ins. This will add any commands entered into the User Key-in box, including tool commands and mouse clicks until Stop is pressed.



An example of a key-in to draw a concrete object line

Play

Executes the current User Key-in. This is useful for testing.

Previous

Highlights the previous command in the list.

Next

Highlights the next command in the list.

Stop

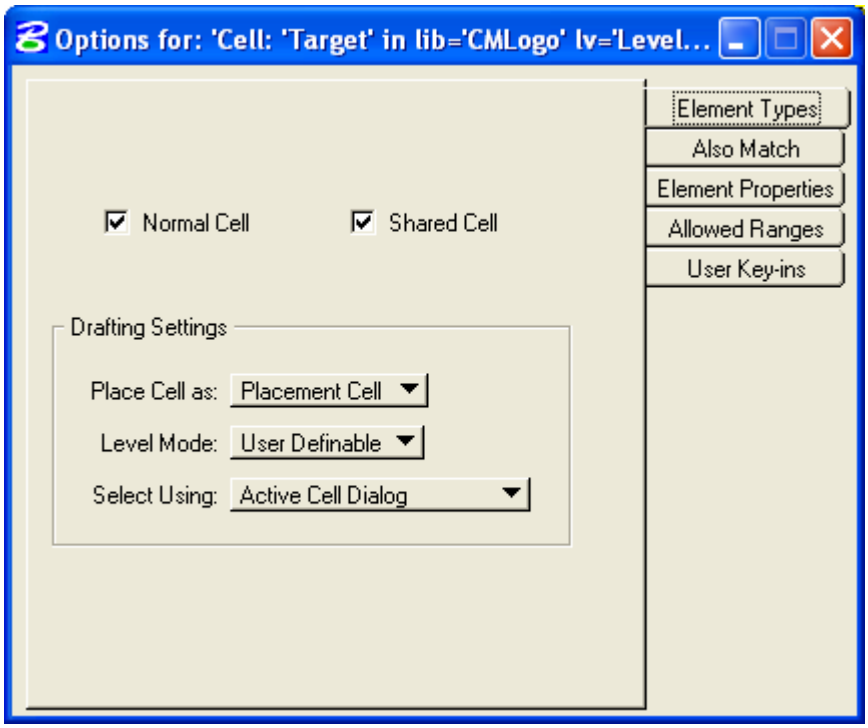
Stops recording of key-ins.

Under MicroStation J, the Previous and Next commands move the cursor but do not highlight the current row. This is a pre-MicroStation v8 limitation.

Cell Options

The Cell Options dialog box has the following tab pages:

Element Types
Also Match
Element Properties
Allowed Ranges
User Key-ins



Element Type options for Cell features

Element Types

Normal Cell

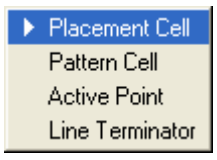
Allows the feature to match a normal MicroStation cell (type 2) element.

Shared Cell

Allows the feature to match a Shared Cell (type 35).

Place Cell As

This option button defines how the cell is used, which affects the user key-in created when placing and scaling the cell.

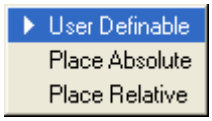


The options are:

- Placement Cell
- Pattern Cell
- Active Point
- Line Terminator

Level Mode

The level mode option button defines how the levels of graphic cells are handled. The options are:

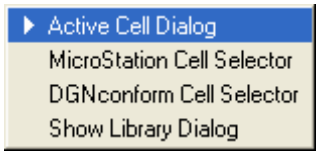


- User Definable – means the user can toggle the "Relative" mode
- Place Absolute – means the cell is always placed on the levels defined by the cell library
- Place Relative – means the cells are always placed in relative mode.

See the MicroStation manual for more information on how relative levels work with cells.

Select Using

This option defines how a cell is chosen from a library. If the cell name is already defined and matched on (see below), then some of these options will not be available. The options are:



- Active Cell Dialog – this is the dialog opened when the user clicks on the "Place Cell" icon on MicroStation's main tool box.
- MicroStation Cell Selector – this is MicroStation's cell selector dialog, as opened from MicroStation's "Utilities" menu.
- CADconform Cell Selector – this is CADconform's built-in cell selector. It is similar to the MicroStation Cell Library dialog, but it supports filtering the cells by either name or description.
- Show Library Dialog – this is MicroStation's dialog as opened from the "Element > Cells" menu.

Also Match

These options are available on the "Also Match" tab page for cell features:

Cell Name

Forces the feature to only match cells of a given cell name.

Cell Angle

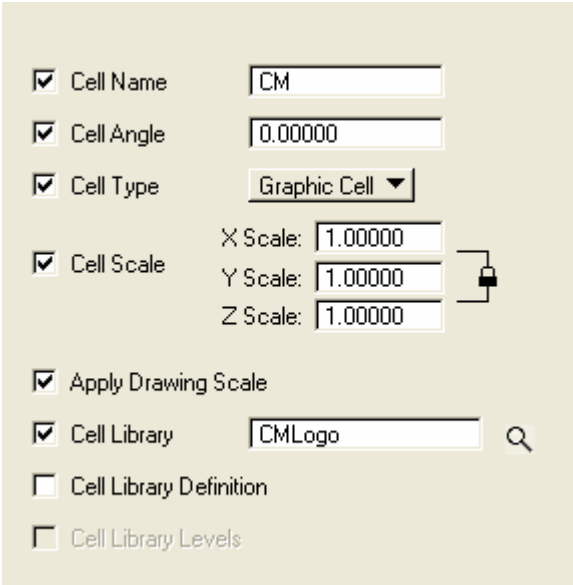
Forces a match on the top view 2D rotation angle.

Cell Type

Forces a match on the type of cell, either Graphic or Point Cell.

Cell Scale

Forces a match on the 3D scale of the cell. The Z Scale is ignored in 2D cells.



Apply Drawing Scale

This option allows the interpreted scale of the cell to be a product of the drawing scale and the cell scale. This affects the scale of the cell in Conform, Reporting, feature matching and Drafting. If this toggle is OFF, then the cell scale will always be exactly as stated, regardless of the drawing scale. The effect is similar to that of the Custom Linestyle Scale option.

Cell Library

This option specifies the cell library from which the cell must be taken. If this option is on and a located cell does not exist in the specified library, then the feature will not match. This cell library is also automatically attached when the Draft command is issued for the current feature. If cell names are not unique across multiple libraries, then it is strongly recommended that the Cell Library option is used, otherwise CADconform will use the first library found that matches the cell name. The Cell Library can be chosen by either keying in the path - relative to $(MS_CELL) or full path including sub directories - or browsing for it using the Browse icon.

 **Cell Library - Browse**

The Browse command will open a standard file selector dialog box. If a cell library is chosen that is not part of the standard cell library path - as defined by $(MS_CELL) – then the Browse tool will automatically include the full path of the library, otherwise it will only include the path relative to $(MS_CELL).

Cell Library Definition

This option specifies whether a cell in the design file must match its library definition or not. This match is calculated by comparing the cell in the design file with the cell in the cell library.

If the Cell Library option is not used, then the cell is searched for in all libraries defined by $(MS_CELLLIST), otherwise it uses the library defined by the Cell Library criteria. Similarly, if the Cell Name is not matched on, then the cell will be searched for based on the name of the design file element found.

The criteria used for deciding whether a design file cell matches its library definition is based on these cell component properties:

- Element type
- Element order
- Element symbology
- Number of vertices (for linear elements)
- Text properties (for text and text node elements)

Note that the following criteria are not checked, and will therefore not constitute a discrepancy between design file and cell library definition:

- Element range
- Vertices (for linear elements)
- Origin (text, text nodes, ellipses and arcs)
- Angles (text, text nodes, ellipses and arcs)
- Graphic Group

This criteria (except Graphic Group) is not checked because the lack of precision inherent in the design file format means that numerically significant discrepancies can exist between library definition and cell instance even when the cell is up to date.

User Key-ins

Useful key-ins for placing cells include:

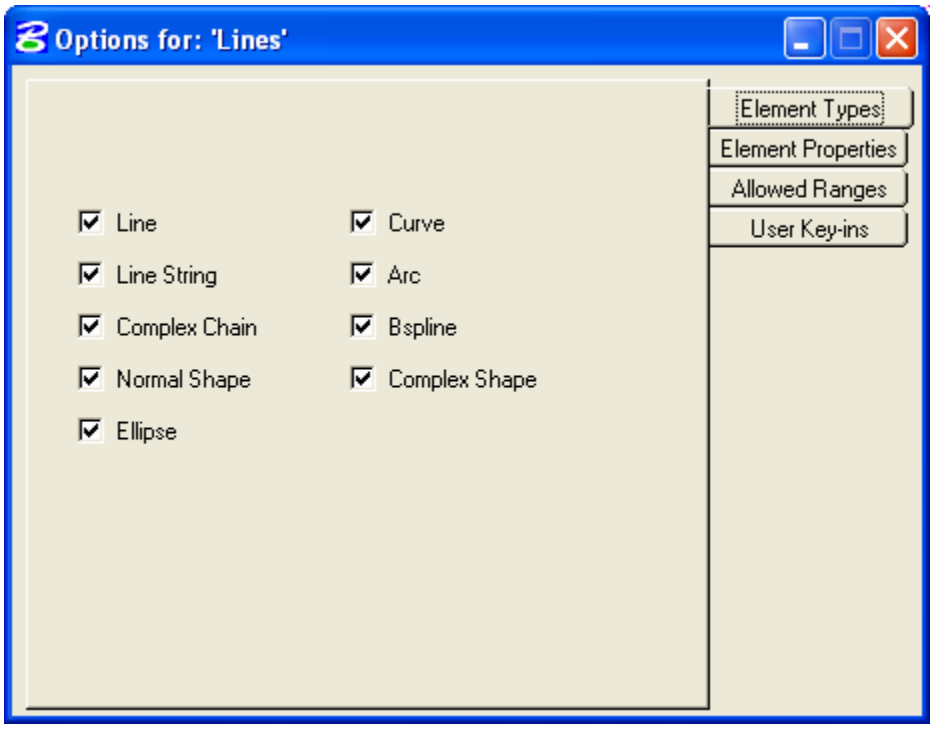
“place cell relative|absolute|interactive” – place a cell in relative, absolute or interactive mode.

“rc=mycelllib” – attaches the cell library called “mycelllib”.

Linear Options

The Linear Options dialog box has the following tab pages:

Element Types
Element Properties
Allowed Ranges
User Key-ins



Element Type options for Linear features

Element Types

Line
Allows the feature to be a line (type 3) element.

Line String
Matches a line string (type 4).

Complex Chain
Allows the feature to match any complex chain (type 12).

Curve
Allows the feature to match a curve (type 11).

Arc
Allows the feature to match an arc (type 16).

B-spline
Allows the feature to match an open B-spline curve (type 27).

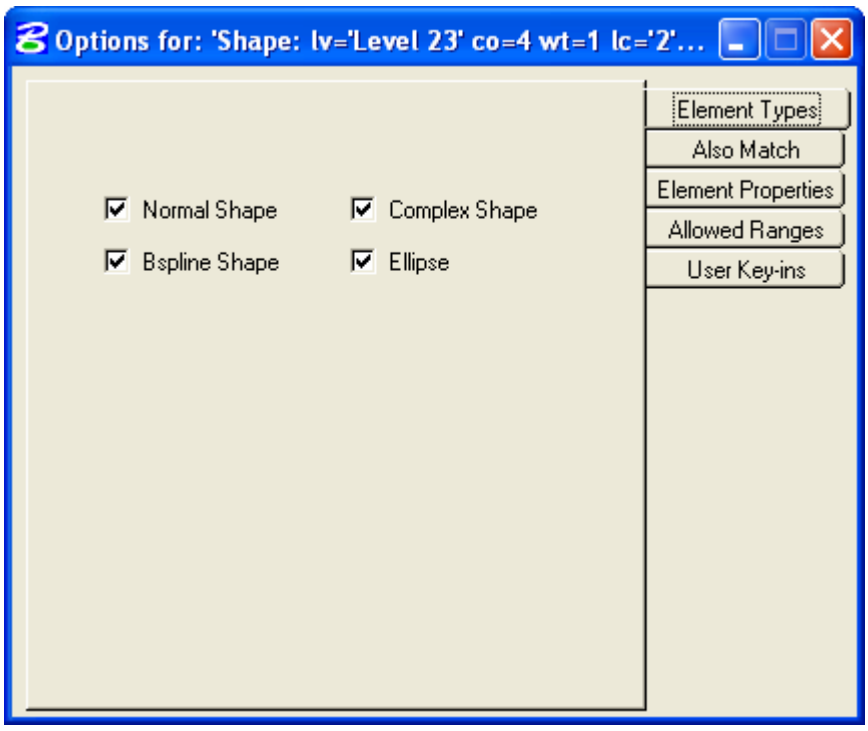
User Key-ins
Useful key-ins for placing linear elements include:

“place arc | block | curve | ellipse | line | shape | smartline” – place an arc, block, curve, ellipse, line, shape or smartline.

Shape Options

The Shape Options dialog box has the following tab pages:

- Element Types
- Also Match
- Element Properties
- Allowed Ranges
- User Key-ins



Element Type options for Shape features

Element Types

These options are available on the “Element Types” tab page:

Normal Shape

Matches any normal MicroStation shape (type 6) element.

Complex Shape

Matches a complex shape (type 14) element. Note that, as with Complex Chains, a complex shape can contain other element types, including linear elements.

B-spline Shape

Matches a closed B spline curve (type 27) element.

Ellipse

Matches an ellipse (type 15) element.

Also Match

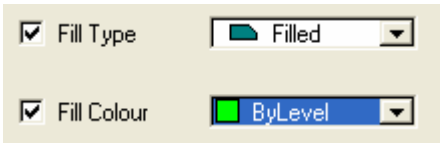
These options are available on the “Also Match” tab page:

Fill Type

Optionally specifies that the shape must be either filled or unfilled.

Fill Colour

Optionally specifies that a filled shaped must have a particular fill colour.



User Key-ins

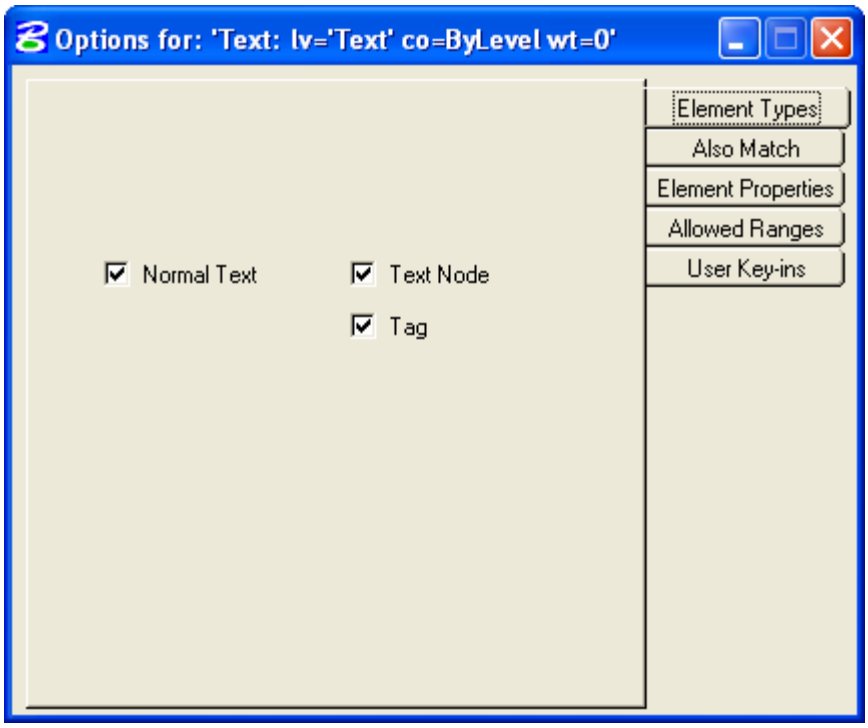
Useful key-ins for placing linear elements include:

“place block | ellipse | polygon | shape” – place a block, ellipse, polygon or shape.

Text Options

The Text Options dialog box has the following tab pages:

- Element Types
- Also Match
- Element Properties
- Allowed Ranges
- User Key-ins



Element Type options for Text features

Element Types

These options are available on the “Element Types” tab page:

Normal Text

Allows the feature to match a normal MicroStation text element (type 17).

Text Node

Matches a text node (type 7) element.

Tag

Matches any Tag element (type 37).

Also Match

These options are available on the “Also Match” tab page:

Text Style (v8 only)

Defines an optional text style that must be applied to the feature. Note that because text styles do not apply to Tag elements, the text style criteria will be ignored for this element type.

Text Style Overrides (v8 only)

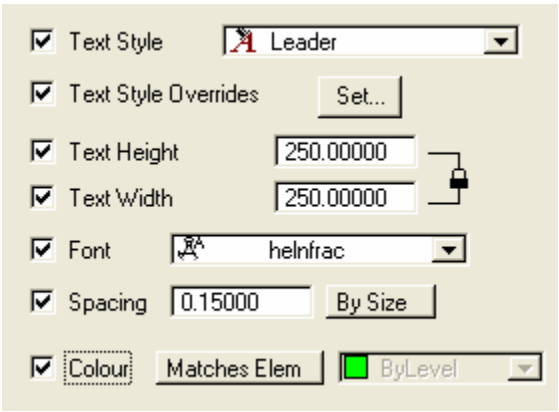
Forces a match on the text style overrides. This means that a match may not be granted for a feature even if the text style is correct, if one of the criteria (such as font) is overridden for the element.

Set Text Style Overrides (v8 only)

Opens the Text Style Overrides dialog box (see “Text Style Overrides” below).

Text Width

Optionally enforces a match on the text width, in master units. Note that text width is always affected by the current Drawing Scale, so text size should be defined for 1:1 scale only.



Optional match criteria for text features in MicroStation v8

Text Height

Optionally enforces a match on the text height, in master units. Note that text height is always affected by the current Drawing Scale, so text size should be defined for 1:1 scale only.

Font

Optionally enforces a match on the font name. The font can be chosen by the font name from the combo box.

Spacing

Defines the line spacing used in text nodes. There are two methods for defining text node line spacing:

- By Scale
- By Size

“By Scale” will make the line spacing relative to the current text height, so a value of 0.5 will equate to a line spacing of 2.0mm if the text height is set to 4.0mm.

If the method is set to “By Size”, then the line spacing works the same way MicroStation line spacing does – which means that the value of the line spacing represents the exact distance to use regardless of the text size. For example, a line spacing of 2.0 by Size will be 2mm apart in a metric file.

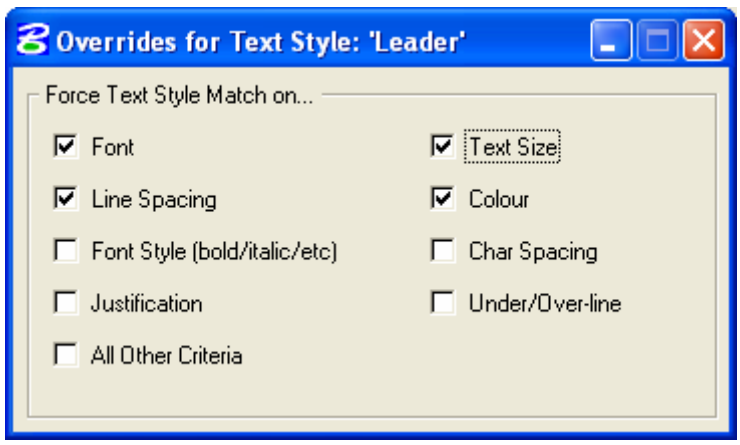
The line spacing is always affected by the current scale, so that if the drawing scale is 1:50, then the line spacing will be 50 times the value predicted in the user key-in.

Colour

In MicroStation v8 and above, the text colour can actually override the element colour. The colour options on this tab page define whether this is allowed or not. If the text colour should not be allowed to override the element colour, then set the Colour toggle ON and set the option button to “Matches Elem”.

Text Style Overrides (v8 only)

Defines which criteria are not allowed to override the chosen text style (OFF means the criteria can be overridden). Note that criteria selected with this tool may deactivate other criteria. For example, if text size is not allowed to be overridden (toggle is ON), then the text size option will be disabled and set to the text size as defined by the text style. This means that a feature that uses the Text Style required but overrides the criteria, will produce a Conform error.



The options on this dialog box are defined below. See the “Text Styles” chapter in the MicroStation online manual for more information on using these override options.

Font

Force text style match on font (deactivates font option on Text Option dialog box).

Text Size

Force text style match on text size (deactivates text size options on Text Option dialog box).

Line Spacing

Force text style match on line spacing (deactivates line spacing options on Text Options dialog box).

Colour

Force text style match on text style colour (deactivates override colour on Text Options dialog box). Note that this may differ from the actual element colour as defined on the Feature Table Editor dialog box.

Font Style

Forces text style match on these characteristics: bold, italic, slant, superscript and subscript.

Char Spacing

Forces text style match on inter-character spacing.

Justification

Forces text style match on these characteristics: text justification, text node justification and full justification.

Under/Over-line

Forces text style match on these characteristics of underlines and overlines: activated, line offset, line colour, line weight and line style.

All Other Criteria

This included all other (rarely used) criteria that the user can set in both the “General” and “Advanced” tab of the MicroStation Text Styles dialog box, that is not covered by any of the preceding options. For example: fraction, upside-down, background colour, etc.

User Key-ins

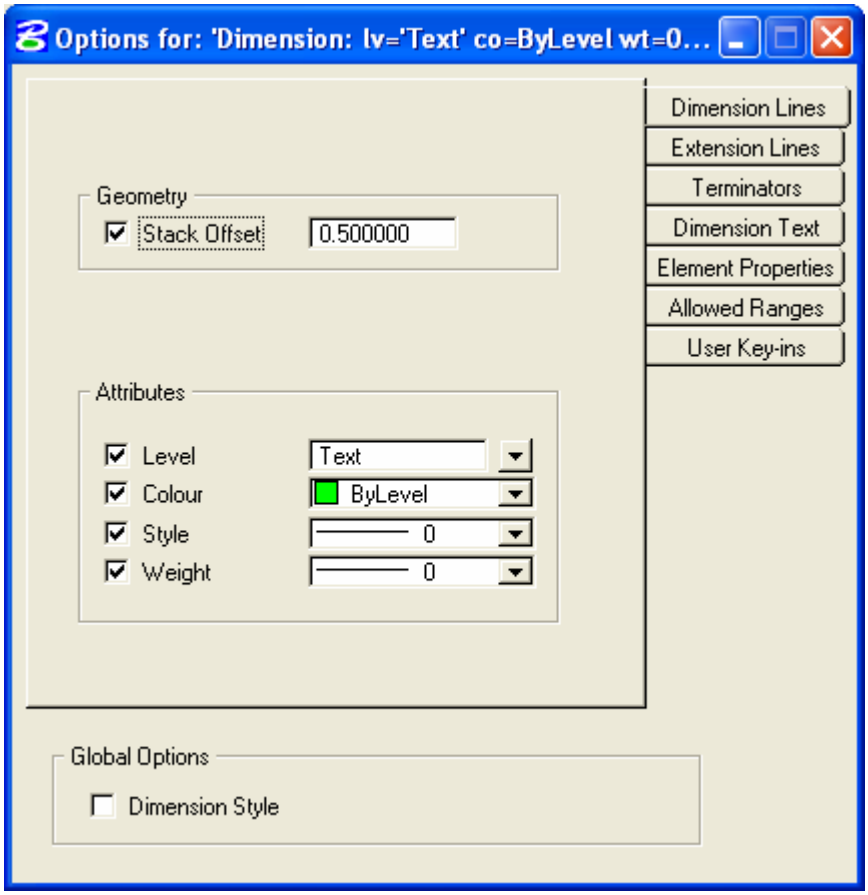
Useful key-ins for placing text elements include:

“place text | dialogtext | node” – place text, open place text dialog, place text node.

Dimension Options

The Dimension Options dialog box has the following tab pages:

- Dimension Lines
- Extension Lines
- Terminators
- Dimension Text
- Element Properties
- Allowed Ranges
- User Key-ins



Element Type options for Dimension features

Most of the settings are analogous to the standard MicroStation dimension settings dialog box, which can be opened in MicroStation using the “Element > Dimensions” menu item. For more detailed information on these settings, consult the MicroStation on-line help manual.

There is no “Element Types” tab page for dimensions, since there is only one mandatory element type for them – type 33.

All of the measurement settings in dimensions are in “text scale units”, which effectively means a proportion of the current text height. The only exception to this rule is Dimension Line Stack Offset, which is measured in working units.

Dimension Lines

This tab page replicates some of the settings on the Dimension Lines tab page in MicroStation's Dimension settings.

Stack Offset

Stack offset controls the distance between stacked dimensions in working units.

Dimension Line Attributes

Controls the symbology (colour, weight, style and level) of the dimension lines. These symbology attributes match the element symbology on the main FTE dialog.

Extension Lines

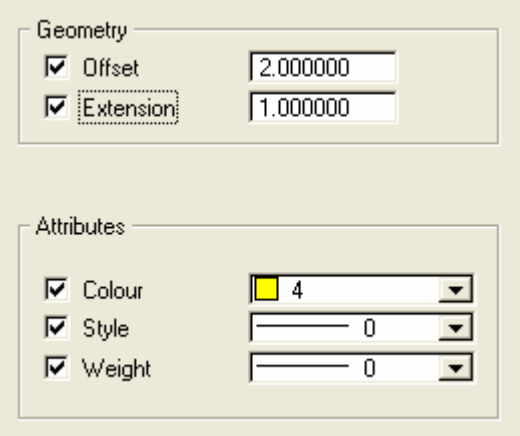
This tab page replicates some of the settings on the Extension Lines page in MicroStation's Dimension settings.

Geometry: Offset

Represents the distance between the element and the start of the dimension's extension lines. This distance is measured as a scale of the text size.

Geometry: Extension

Represents the distance beyond the dimension line that the extension line overhangs. This distance is measured as a scale of the text size.



Extension Line Attributes

Controls the symbology (colour, weight and style) of the extension lines of the dimension. They can override the symbology inherited from the Feature Symbology settings.

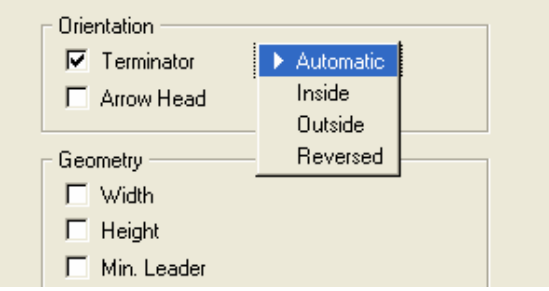
Terminators

This tab page replicates some of the settings on the "Terminators" page in MicroStation's Dimension settings.

Orientation: Terminator

This setting controls the placement of arrow heads with respect to the extension lines. The options are:

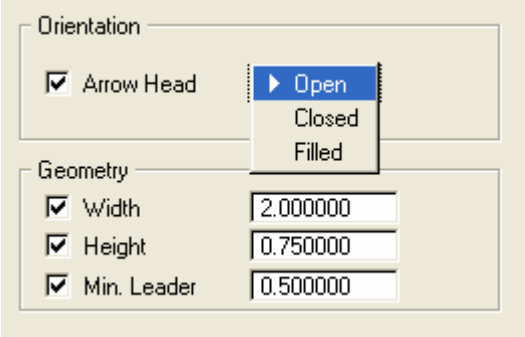
- Automatic
- Inside
- Outside
- Reversed.



Orientation: Arrow Head

Controls the appearance of the dimension arrowheads, either:

- Open
- Closed
- Filled



Geometry: Width

Specifies the width of terminators, in text scale units.

Geometry: Height

Specifies the height of terminators, in text scale units

Geometry: Min. Leader

Specifies the distance between the extension lines and the edge of the dimension text, in text scale units.

Terminator Attributes

Controls the symbology (colour, weight and style) of terminators. This can override the symbology inherited from the Feature Symbology settings.

Dimension Text

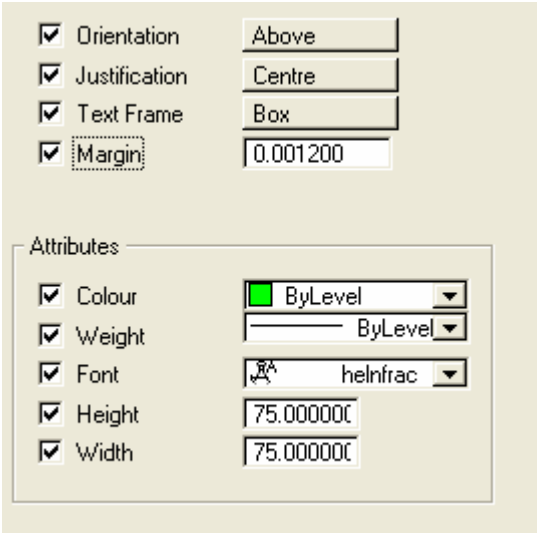
This tab page replicates some of the settings on the “Text” page in MicroStation’s Dimension settings.

Text: Orientation

Controls the placement of text relative to the dimension line.

Options are:

- In Line
- Above
- Horizontal



Text: Justification

Controls the justification of the dimension text, either:

- Left
- Centre
- Right

Text Frame

Allows a frame to be placed around dimension text, either:

- None
- Box
- Capsule

Margin

Specifies the minimum distance between the leader line and the dimension text. This is equivalent to the “Left Margin” setting in MicroStation v8, and the “Margin” setting in MicroStation J and below. It is measured in text scale units.

Text Attributes

Controls the symbology (colour and weight) of the dimension text, as well as the font and text size. This can override the colour and weight inherited from the Feature Symbology settings, as well as the active settings for font and text size.

The text height is critical in determining the value of all match criteria that is measured in text scale units. It is highly recommended that the text height is matched on (toggle ON) if any of the other criteria is enabled that uses text scale units (eg: text margin, terminator width, etc.)

User Key-ins

Useful key-ins for placing dimension elements include:

- “dimension element” – activate the dimension element icon.
- “dimension size” – activate the dimension size icon.
- “dimension size stroke” – activate the dimension size with stroke icon.
- “dimension angle size” – activate the dimension angle size icon.

Command Options

Command feature types are reserved for all generic commands that are not associated with any one particular feature or element type. Command features are not used for matching, Conforming or Reporting, they are only used by CADconform Draft.

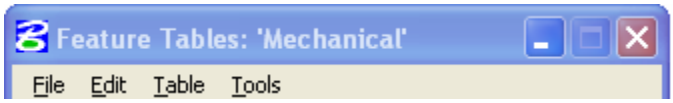
Command features have only one options tab page for the User Key-in. This key-in can be any type of command, such as a macro, user command, mdl command or anything else a user might type in to the MicroStation key-in field. It can also be used for displaying documents or opening web pages. An example of a command that opens a web page might have a key-in such as:

“CADconform START http://www.corporatemontage.com.au”
“CADconform START” is a command added to MicroStation’s command table when CADconform is loaded, which allows any URL to be opened in the system default web browser.

To open a file, such as the CADconform manual, you might create a key-in such as:

CADconform RUNPROGRAM "$(_CADconform_DOCS_DIR)\CADconform for Microstation Guide.pdf
The “CADconform RUNPROGRAM” command allows any program or file to be opened or executed. As in this example, taken from the default menu for CADconform, it also supports automatic expansion of configuration variables.

The Menu Bar



The Feature Table Editor menu bar appears along the top of the Feature Table Editor dialog box. As well as providing a duplicate interface for most of the icon commands, the menu bar can be used to import data from files, export the Feature Table to the database, sort the features by name and type, clear the table and remove duplicate features. The various menus are listed below:

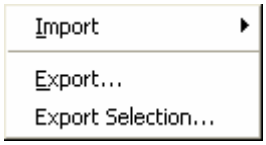
- File
- Edit
- Table
- Tools

The File Menu

The File menu is used to import data from existing files, and export data to dictionary files.

There are three options in the File menu:

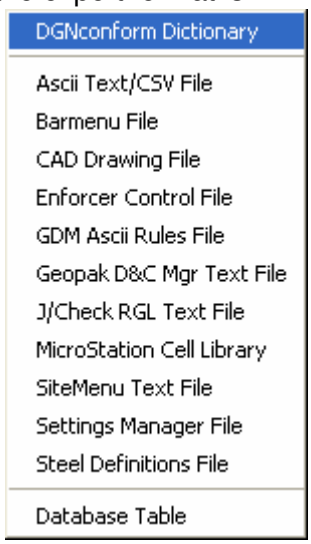
Import
Export
Export By Selection



Import

Import will bring data into the current feature table at the position of the currently selected list box row. This means multiple data sources can be merged together into the one Feature Table.

CADconform can import data from many different file types, but the export format is limited to Feature Dictionary files, as well as the database.



The file formats currently supported by CADconform for importing are:

- Feature Dictionaries
- ASCII Text / CSV Files
- Barmenu files
- DGN Drawings files
- Enforcer Control Files
- GDM ASCII Rules files
- Geopak D & C Manager Text Files
- J/Check RGL Text files
- MicroStation Cell Libraries
- SiteMenu Text files
- Settings Manager files
- Steel Definition files
- Database Tables

Feature Dictionaries

Feature Dictionaries are binary files designed for use with CADconform, CADconform PE and CADconform CE. They can also be used to back up Feature Tables outside of the database, or to transfer data between different Feature Tables or databases. If DGN standards are to be deployed to users outside the corporate intranet, then Feature Dictionaries can be used as a portable file format.

ASCII Text / CSV Files

ASCII Text files are simple text files, which are usually exported from other applications such as spreadsheet or database programs. CSV files are similar, but contain Comma Separated Values (CSV). Choosing a CSV file to import will open the CSV Preview dialog box, described below.

Barmenu Files

Barmenu files are files created for use with "Barmenu", an MDL application supplied with MicroStation to allow users to define their own custom menus quickly and easily. Support for Barmenu files is particularly useful for importing files to use with the Drafting Menu.

DGN Drawings (DWG/DWX for v8 only)

CAD/DWG/DXF file import brings every unique feature that exists in the DGN drawing into the current Feature Table, into a flat (non-hierarchical) list. This include features on levels/layers that are not displayed.

Enforcer Control Files

Control Files are used by Enforcer3™, a standards compliance application. CADconform can import most of the data from these file types, retaining the group hierarchy.

GDM ASCII Rules files

These files are created by the GDM symbology checking application. CADconform can import most of the data from these files, retaining the group hierarchy. It can also import with or without the header file.

Geopak D & C Manager Text Files

D&C Manager files are text files exported from D&C Manager from within Geopak™. CADconform will import most of the relevant data from these files, retaining the group hierarchy.

J/Check RGL Text files

These files are created by J/Check, another symbology checking package. CADconform can import most of the data from these files.

[3] Enforcer is a registered trademark of Australian Data Systems Pty Ltd.

MicroStation Cell Libraries

MicroStation cell libraries can be opened to automatically add all cells in the library to the Feature Table. This saves the user having to enter all of the cells by hand. By default, graphic cells will be matched only on base level, whereas point cells will be matched on also symbology – level, colour, weight and style.

MicroStation J cell libraries must be converted to v8 format before they can be imported into MicroStation v8. This is not necessary if running CADconform on MicroStation J.

Settings Manager Files

"Settings Manager" files are used by MicroStation's Settings Manager (which can be opened by clicking on the MicroStation menu: "Settings > Manage"). Some third party applications use Settings Manager files, such as Plantspace P&ID. Currently, only MicroStation J Settings Manager files are supported.

Site Menu Files

These are ASCII text files used with SiteMenu4™. CADconform can import all of the data from these files into a two level hierarchy.

Steel Definition files

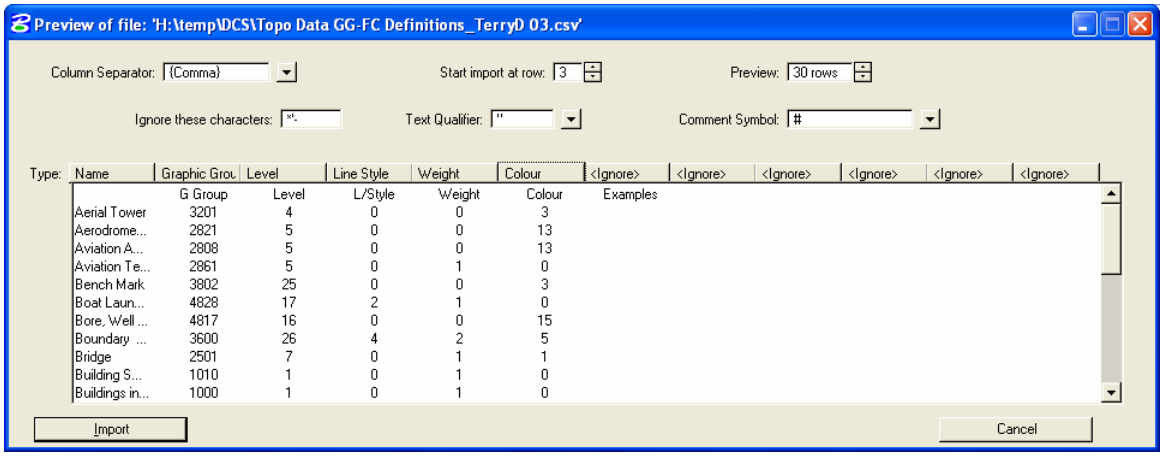
Steel definition files are created by hand for use with the Steel applications, such as Steel 97, Steel v8 and Structural for Triforma. CADconform can import data from all of these formats to create a list of cells, which can be allowed when conforming. These cells will be hidden from Draft by default, since they will usually be drawn using the Steel application.

Database Tables

Database tables can be imported from an ODBC source as long as a data source name (DSN) has been defined and configured for it. Importing a database table opens the Database Import Preview dialog, explained in detail below.

[4] SiteMenu is a registered trademark of Rowse Company

CSV Preview Dialog



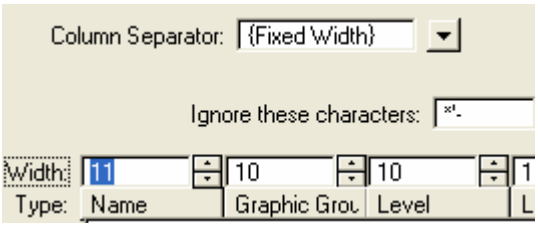
Importing a CSV File

The CSV Preview Dialog shows a preview of the CSV file to import. All settings are "live", in that changes are reflected immediately as to how the data in the file is interpreted. Sometimes it is necessary to edit the CSV file using a Text Editor (such as NotePad) prior to importing the CSV file if, for example, there are commas in the feature name in addition to the column separators. The Preview dialog box allows the user to see how the CSV file will be interpreted so that changes can be implemented prior to import, if necessary. Each setting on this dialog is described below.

Column Separator

The column separator defines what character is used to differentiate each column. This is usually a comma, but sometimes a tab or semicolon is used. If all of the data is "bunched up" in the first column, or spread across too many columns, then it is likely that the wrong column separator has been chosen. Any character can be entered in the text field if it is not in the drop-down list (for example, a percentage '%' character).

If the "Fixed Width" option is chosen, then the field widths for each column can be entered in the text fields that appear above the column names. In Fixed Width mode, rather than having a character identify the start of each column, the file has a fixed number of characters allowed for each column. These sorts of files are typically created by database applications or data recorders, as they are much easier to read when printed out as each column lines up for each row. Windows often gives these types of files a "PRN" extension, meaning "print format".



Fixed Width Column Separators

Trailing spaces are ignored

Start Import at Row

This defines where the import is to begin. CSV files often have descriptions at the top couple of rows, meaning useless data will be imported at the beginning of the file. For example, the above screenshot (Figure 15) has the column labels in the first three rows, and hence the start row should be set to “4”.

Preview

This setting defines how many rows of the CSV file to display in the preview window. This can be useful to review more of the file to ensure it is not badly formatted further into the CSV file.

Ignore These Characters

Sometimes CSV files contain punctuation characters, such as tabs, that should be ignored. Any character defined in the “Ignore These Characters” text field will be stripped prior to importing into the feature table.

Text Qualifier

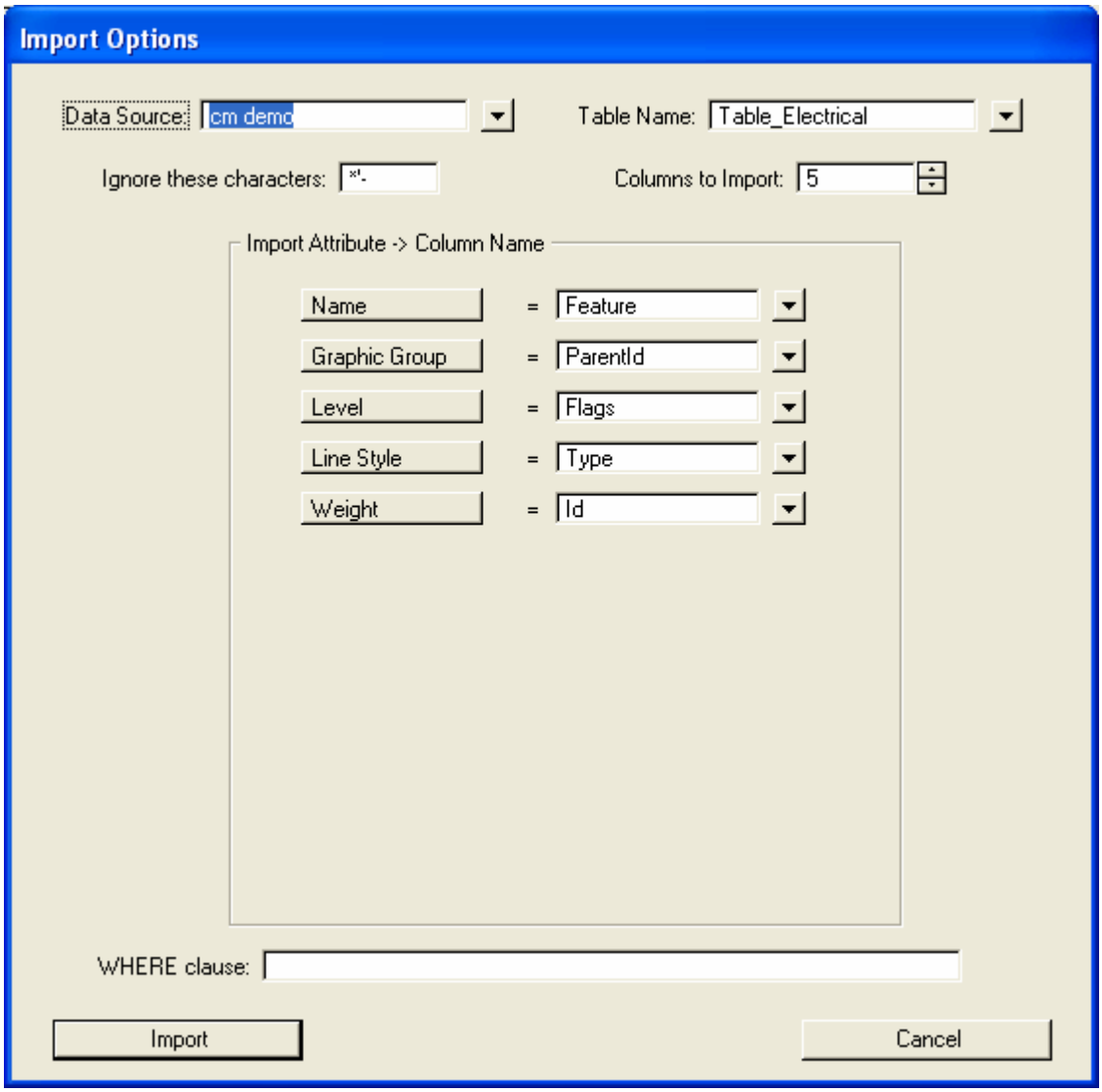
Files exported from other applications (including Microsoft Excel) sometimes add single or double quotes around text, to represent a literal string value. The text qualifier option defines which characters to interpret as a text qualifier, so that these characters can be stripped out. Additionally, text qualifiers allow literal interpretation of strings so that, for example, a comma can be allowed as part of a feature name even thought the text file is comma separated.

Comment Symbol

The comment symbol defines a character that denotes that the text to follow is a comment or remark, and should not be treated as data to import. Typically this is a semi-colon or hash ‘#’ (US: pound) symbol. Any text following the comment symbol on the same line will be ignored.

Database Import Preview Dialog

This dialog opens when the user imports a database table from the file menu. The options on the dialog are described below:



Data Source

This is the connection string for the data source name (DSN). As with the databases used in CADconform, there must be an existing ODBC DSN defined in order for CADconform to connect to the database. This can be defined as one of the standard File DSNs in CADconform's "File DSN" directory, or alternatively a System or User DSN defined in the Windows ODBC Control Panel.

Table Name

This comb-box displays a drop-down list of all available tables in the database connected to by the Data Source name.

Ignore These Characters

This text field allows the user to strip any unwanted characters from each database row as it is read. Examples might include single or double quotes around strings.

Columns to Import

This spin-box allows the user to define number of columns to import from the table. Increasing the number of columns will add extra items to the dialog box. The maximum number of columns is currently set to 12, due to dialog size constraints.

WHERE Clause

The “where” clause defines an optional filter to add to the SQL query prior to importing. An example of this would be if you only wanted to import features on a certain level or with a certain identifier. The “where” clause takes the typical SQL syntax form of:

<COLUMN NAME> <OPERATOR> <VALUE>

where…

<COLUMN NAME> is one of the column names.

<OPERATOR> is one of: < > <= >= <> BETWEEN LIKE

<VALUE> is either a number, or a string enclosed in single quotes.

Some examples of the “where” clause would be:

LEVEL_ID <= 63

PARENT_GROUP = ‘PAVEMENT’

FEATURE_NAME LIKE ‘%CONCRETE%’

Respectively, this would import:

All features with LEVEL_ID less than or equal to 63

All features with the PARENT_GROUP column equal to ‘PAVEMENT’

All features with the word ‘concrete’ in the FEATURE_NAME column

More information on SQL statements can be found by searching on the internet for:

“SQL syntax tutorial”.

Import

Begins the import from the database and closes the dialog.

Cancel

Cancels the import from the database and closes the dialog.

Export

Export will save the current Feature Table to an external Feature Dictionary, also known as a DICT file. This is useful for backing up feature tables, transferring data between databases and supplying feature tables to other users electronically.

Export by Selection

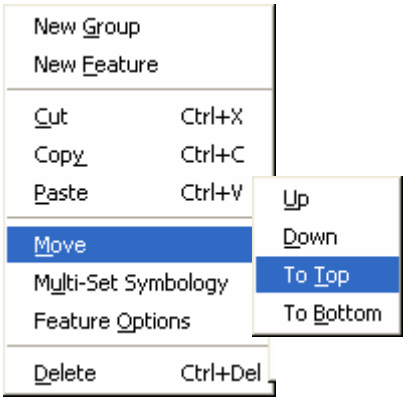
Export by Selection will export to a dictionary file only the selected rows in the list-box. Selected groups will also have all child features exported. This is useful for cutting and pasting between multiple tables.

The Edit Menu

The Edit Menu is used to manipulate the features in the Feature Table. All of the Edit options have icon command equivalents.

New Group

See “Icons: New Group”.



New Feature

See “Icons: New Feature”

Cut

See “Icons: Cut”

Copy

See “Icons: Copy”

Paste

See “Icons: Paste”

Move

Moves a selected feature or group up or down relative to the other features. This is useful for reordering the feature table.

Multi-Set Symbology

This tool is used to set the symbology for multiple features at once. It opens the Multi-Set Symbology dialog, described below.

Feature Options

See “Feature Type Options”

Delete

See “Icons: Delete”

Multi-Set Symbology

The Multi-Set symbology dialog allows the user to set the symbology for many features at once. The user can set the following properties from this dialog:

- Level
- Colour
- Weight
- Style
- Hide Status

The Multi-Set Symbology dialog is similar in operation to the Windows Properties dialog box, as described below.

Default Toggle Colour Meanings

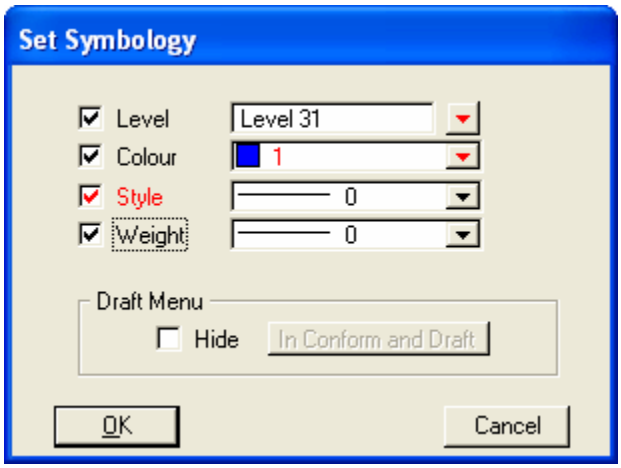
- If a toggle (eg: weight) is ON and black, it means that all selected features currently have the toggle ON.
- If a toggle is OFF and black, it means that all selected features have the toggle OFF.
- If a toggle is red, then it means that at least one feature has the toggle ON and at least one feature has the toggle OFF.

Applied Toggle Colour Meanings

If a toggle is red, then it effectively has three states instead of two:

- Set all features ON
- Set all features OFF
- Leave features alone, with some ON and some OFF

The user must click the toggle three times to cycle through all possible states.



The Multi-Set Symbology dialog. Note that the style toggle is red, meaning that style is not set to ON for some features.

This logic does not apply to the option buttons and combo-boxes, as these have only two effective states: changed and unchanged. Once you set a red option (non-toggle), it changes from unchanged to changed, which means the setting will be applied. Unlike toggles, the option can not then be set to unchanged.

After the user hits “OK”, the active properties are applied to all features that were selected. Note that only black settings will be applied, red settings will be ignored.

The Table Menu

The Table Menu is used to perform operations on the entire table at once. The menu commands are listed below:

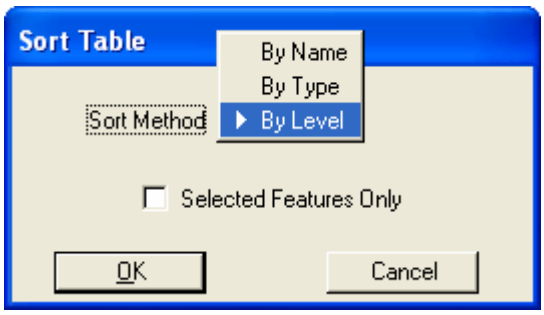
Export

This will export the current contents of the table to the database. This will overwrite the existing contents of the table in the database. A completion bar will open and display the progress of the export operation. If the table is very large or the network connection is particularly slow, then this operation could potentially take some time. Export can be optionally done if the Feature Table has been edited and an attempt to close the Feature Table Editor window is made, or the table is cleared (see below).

Clear

This will clear the contents of the table. If the table has not been saved, then a warning will appear asking if you wish to proceed.

Sort



Sort allows the Feature Table to be sorted by one of three methods:

- By Feature Name
- By Feature Type
- By Feature Level

Note that sorting by type will also sort features by name if they belong to the same type. Sorting by type can be convenient where one group has many entries of different types. Sorting by name will sort features alphabetically by name regardless of feature type.

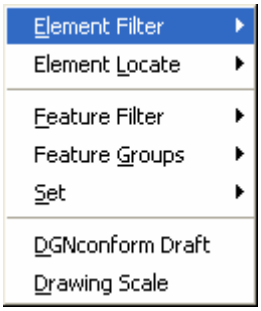
Del Duplicates

This command removes duplicate features from the Feature Table. Duplicate features may have been added manually by the user or imported from external files. There are no problems with Feature Tables containing duplicate entries, however they are effectively redundant and may slow down the opening of Feature Tables if there are many of them. The “Del Duplicates” command works by comparing the name and symbology of every feature in the Feature Table with every other feature in the Feature Table. Where a perfect match is found, the feature is automatically deleted. If the feature names are identical but the symbology differs (or vice versa), then the user will be prompted whether they wish to remove the duplicate, ignore it or cancel. Note that separators and feature groups are not affected by this command.

The Tools Menu

The tools menu has the following items:

Element Filter
Element Locate
Feature Filter
Feature Groups
Set
CADconform Draft
Drawing Scale



Element Filter

Turns on or off the Display Filter.

Element Locate

Activates the Locate Tool with one of the following commands:

Identify Feature
Add Features
Find Feature
Read Watermark

Feature Filter

This tool filters the feature list-box by removing features that are not currently in the active design file / model. The filter can be turned ON or OFF by switching between the two modes:

- Show All Features {OFF}
- Show Used Features {ON}

Feature Groups

Allows global control of the open-status of feature groups. Options are:

- Open All
- Close All

Set

This menu has four options, as described in the Set Icon section of the manual.

CADconform Draft

Opens the Drafting Menu with the currently opened feature table.

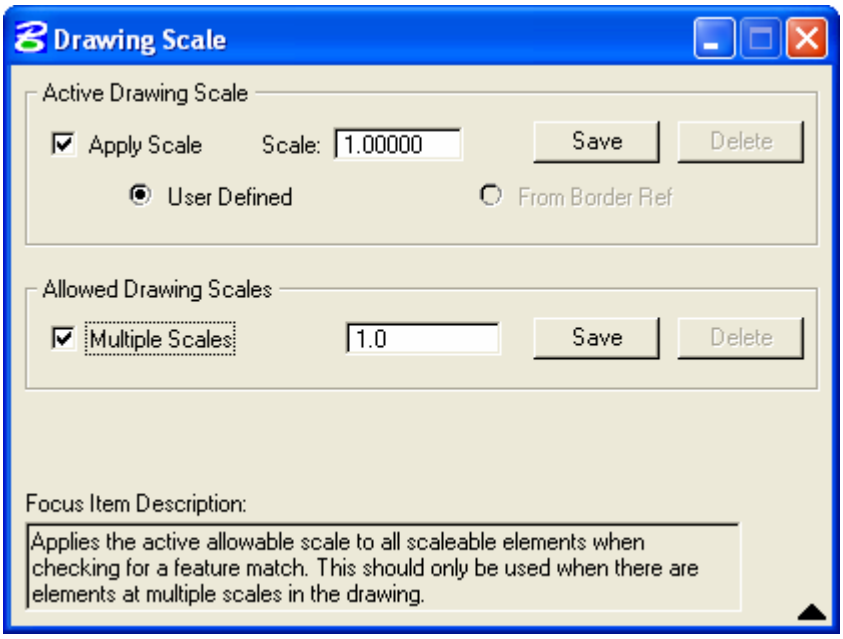
Drawing Scale

This menu item will open the Drawing Scale dialog box, described in the next chapter.

Chapter 8

Drawing Scale

The Drawing Scale dialog allows the user to define the active scale of the current design file / model. The drawing scale can be set in the Feature Table, Conform, Report Generator and Drafting Menu. The drawing scale defines the scale to apply to text sizes, line spacing, custom line-style scale, dimension text sizes and cell scales. It affects both the way features are matched, and the way they are drafted.



Purpose

The Drawing Scale allows the user to define text sizes only once in a text feature, and then apply a different drawing scale to get various text scales. The Drawing Scale is effectively a global scale multiplier, so it can only really be used if a uniform scale multiplier can be applied to text features in the context of one design file / model.

For example, if the scale of the design file was 1:30, then all text sizes would automatically be Conformed, Reported, matched and Drafted at 30 times the scale that they were defined at for 1:1 scale. Thus, a text feature with text size of 1.5mm would only conform if it appeared in the model as 45.0mm in size.

If no global scale multiplier can be applied to a model, then the scale should be set to one (1.00) or Apply Scale should be turned OFF. In this case, individual features will need to be added to the Feature Table for each text size. For example, you would define text features for 1.5mm, 3.0mm, 4.5mm for design files that had scales of 1:1, 1:2 and 1:3 respectively.

Setting the Drawing Scale Automatically

The drawing scale can be set manually and is only applied when the “Apply Scale” toggle is ON. Otherwise a value of “1.00” will be implied.

The drawing scale can be automatically set for every design file by one of two methods:

1) Reading the Drawing Scale last saved to the Design File by CADconform.

2) Reading the reference attachment scale from a standard border sheet.

1 is achieved by a type 66 level 20 element which gets written to the master design file (the active model in MicroStation v8) by CADconform when the user presses the “Save” button. This element contains the drawing scale last saved, and is read whenever a new drawing is opened or CADconform is activated.

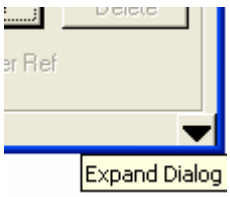
2 is achieved by CADconform comparing all current reference files to a predefined list of standard border sheets. If a match is found, then the reference attachment scale for this model is read and the Drawing Scale is set to this value. Allowable border sheets are defined by the administration configuration file: “CADconform.CFG”.

If both 1 and 2 are set, then 1 will take precedent, which allows the user to override the interpreted Drawing Scale for particular drawings where the scale needs to be overridden.

If neither method 1 or 2 are set, then the drawing scale dialog box will automatically open to allow the user to specify the drawing scale. The radio buttons below the scale options will show which method was used to read the active scale.

The Save and Delete buttons will only be enabled if the scale method is set to “User Defined”. If a saved scale already exists for the active design / model, then the Save button will be disabled and the Delete button will be enabled. Similarly, the Delete button will be disabled and the Save button enabled if no scale is currently saved. If there are no standard borders defined or attached to the current design file / model, then the “From Border Ref” radio-button will be disabled.

Multiple Scales



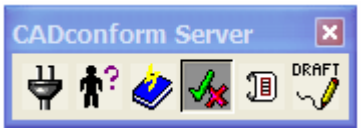
If more than one scale is defined in one design file or model, then each scale can be defined in the “Multiple Scales” definition of the Drawing Scale dialog box, separated by commas. If this option is not visible, then press the down arrow icon on the bottom right of the dialog box to expand the window. The multiple scales can also be saved to the design file, or read from standard border sheets.

Defining multiple scales effectively means that all features in the design file can be interpreted at any of the multiple scales defined. This can create ambiguity if only specific areas of the drawing are at different scales. If this is the case, then it is better to disable multiple scales, restrict Conform or the Report Generator to those areas individually using a fence, and run the tool at the one specific scale for that area.

Although features are allowed to be at any of the multiple scales for matching purposes, they are only ever Drafted or Conformed at the active scale.

Chapter 9

Conform



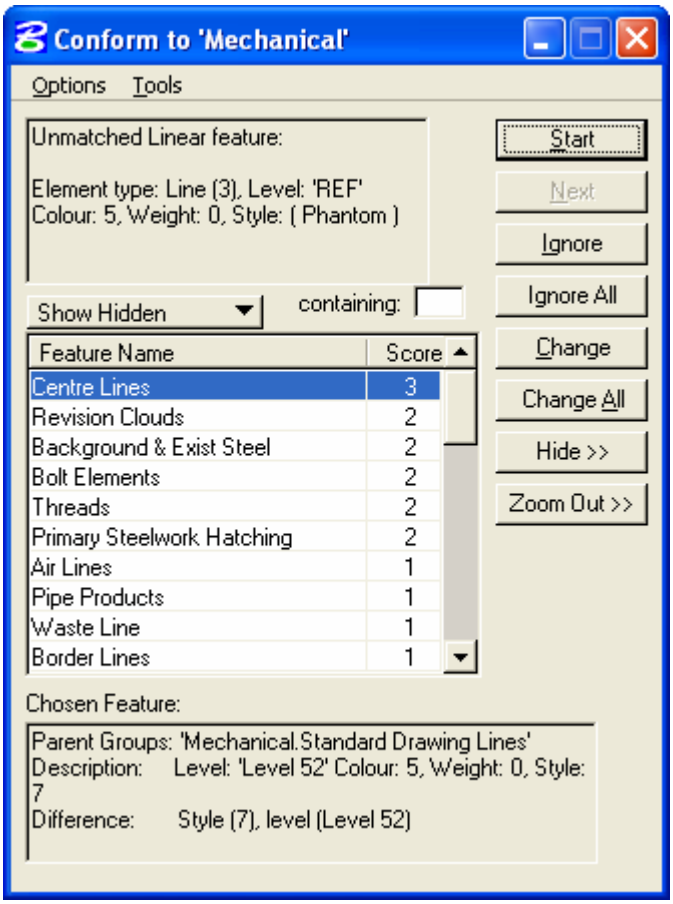
Conform is the feature conformer. This tool allows the user to correct invalid feature symbology in the active design file. After launching the Conform tool, the Feature Table Manager opens to allow the user to select the desired Feature Table. If there is more than one Feature Table in the database, care must be taken that the correct Feature Table is selected for the current design file. If you wish to conform a design file to multiple Feature Tables, then hold down the control key <CTRL> and select each one individually. After selecting the Feature Tables, Conform will import the data from the tables, and the Conform dialog box will open.

Conform will scan every element in the active design file to check if it has a matching feature in the loaded Feature Tables. Conform will only scan and correct elements in the active design file (the master file) and only on levels specified by the Level Map. Where a non-matching feature is found, the scanning stops and the element is identified as an “Unknown Feature”. Unknown features can be ignored or changed to an existing feature in the Feature Table. Optionally, the feature can also be added to the Feature Table if it should be in the current standards table, assuming the user has the “Append to Dictionary” privilege.

Conform has the ability to scan within complex components such as cells, complex chain and text nodes. This functionality is controlled by the administrator and defined in the Administrator’s configuration file. If a component of a complex element is unmatched, then the entire component will be highlighted in the active highlight colour, and the unmatched component will be selected and/or flashing, depending on the display options.

Unknown features will be optionally identified by zooming the view or highlighting, flashing or selecting the element, depending on the Display Options. The user will then be presented with a sorted list of known features to change the feature to, with the best match at the top of the suggestion list box. Sorting the suggestion list can take a while for large Feature Tables (over 1000 entries). To abort the sorting of the Feature List, press the reset button (usually the right mouse button).

When Conform has finished scanning the current design file, a summary of the number of conformed features is displayed, as well as the number of features ignored and added. Conform can abort the scanning operation at any time by pressing the reset button.

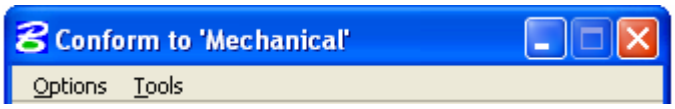


The Conform dialog, after finding an unmatched text element. Note that some of the suggested replacement features may appear in magenta, meaning they are hidden in the Drafting Menu.

Conform Dialog Items

Conform has the following options on the dialog box:

Conform Menu Bar



(See below)

Start

Begins scanning the design file, aborting any previous scan operation.

Next

Finds the next invalid element, after an element is corrected. This allows the user to interrogate the changes made by Conform before continuing to process the rest of the design file.

Ignore

Ignores the current invalid element and continues scanning.

Ignore All

Ignores all instances of this unmatched feature until the end of the design file.

Change

Changes the symbology of the unmatched feature to match the symbology of the selected feature in the Feature Table.

Change All

Changes all features in the design file that match the current unknown feature, to match the symbology of the selected feature in the feature table. Conform will then report how many changes were made to the design file, and then allow the scan to continue from where it left off.

Hide/Show

This button works as a toggling command between “Show” and “Hide”. In “Hide” mode, this command temporarily hides all elements in the design file that do not match the current unmatched element. This allows the user to quickly preview all of the elements that would be affected by a “Change All” or “Ignore All” command. It is also useful when the current element is obscured by other elements in close proximity. The hidden elements will reappear the next time the view is refreshed. In “Show” mode, the view is automatically refreshed to show all elements.

Zoom Out/In

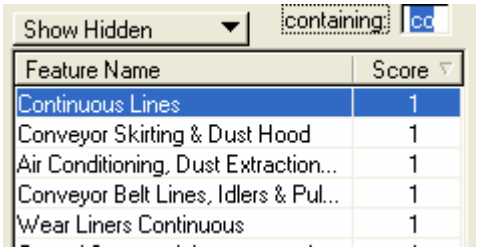
This button works as a toggling command between “Zoom In” and “Zoom Out”. Zoom In occurs automatically when an unmatched feature is found and the “Zoom” option is enabled in the Display Options dialog box. Zoom Out will do a “fit view” of the current scan area. Note that this fitted view is not necessarily the same as the MicroStation fit view command, because the Zoom Out will only show elements in the current design file / model (excluding reference files) and will only show the scan area, which may have been restricted by a fence or boundary shape, or possibly both.

Show All/Normal Features

This option button toggles between displaying all replaceable features in the list-box, or showing only features that are not hidden in the Draft menu. Hidden features appear magenta (instead of black) in the list-box.

Containing

This text-field allows the user to type in a filter string to hide features that do not contain the typed string. The filtering occurs immediately after typing a keystroke, and can thus be very useful when large lists of features are replaced.

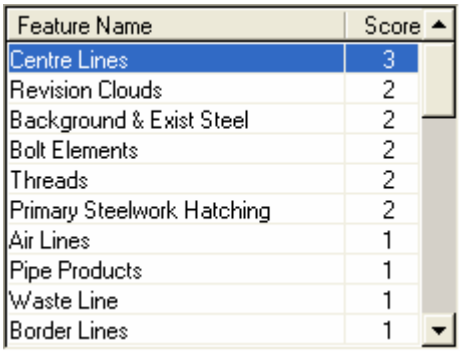


Filtering features by the string "co"

Change To (List-Box)

This list-box summarises all of the features in the Feature Table that are of the same type as the current unknown feature. This list is sorted in descending order of closest match in the Feature Table. Therefore, the top of the list will always be the most likely match for the unknown feature.

The match score of each feature is also displayed in the right-most column of the list box. This score represents how many criteria were successfully matched. For example, if a rebar line was drawn at the correct level, colour and weight, but the style was incorrect, then it would have a score of three.



The selected feature in this list box will represent the feature that the user wishes to change the unknown feature to. Information on this suggested feature is displayed below the list box in the “Group”, “Description” and “Difference” text fields.

Group

The Group text field displays the table name and parent groups of the selected feature in the “Change To” list box. For example, a feature in a group named “Civil & Structural” in a Feature Table called “Cartographic” will have a group name: “Cartographic.Civil & Structural”. Each parent group is separated by a full stop “.”

Description

The “Description” text field summarizes the symbology of the feature currently selected in the “Change To” list box. For example, a Liner Plate may have a description:

Level: 'PIL' Colour: 5, Weight: 2, Style: 0

Difference

Defines the “Difference” between the currently selected feature in the “Change To” list box, and the unmatched feature. Each mismatched property will be listed, for example:

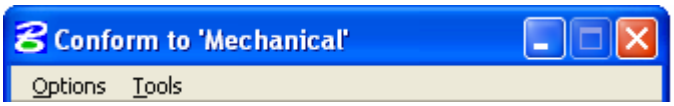
Colour (1), weight (0), level (Main Roads)

means that the chosen feature differs from the unmatched feature because it has a colour of 1, weight of 0 and is on a level called “Main Roads”.

Where a scaleable value is present (such as text size or line spacing), the size is displayed as a percentage scale representing the relative difference. For example, “Text Size (0.5%)” means that the text size is only slightly wrong – by a factor of only 0.005.

Conform Menu Bar

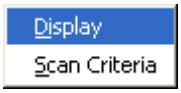
Conform has a menu bar with two sub-menus:



Options
Tools

Options

The options menu allows the user to set various options related to conforming the design file. The options are:



Display
Scan Criteria

Display

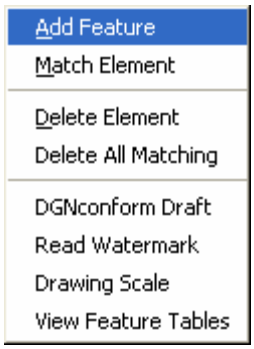
This menu item open the Display Options dialog box.

Scan Criteria

This menu item opens the Scan Criteria dialog box.

Tools

The tools menu currently has the following menu items:



Add Feature
Match Feature
Delete Feature
Delete All Matching
CADconform Draft
Read Watermark
Drawing Scale
View Feature Tables

Add Feature

Adds the current unknown feature to the Feature Table. The Feature Table will then be exported to the database when the Conform dialog box is closed. The added feature will be allocated a default name based on its symbology, and placed in a group called “Added by UserName” where “UserName” is the full name of the user who is currently logged on. This feature can then be renamed and relocated using the Feature Table Editor after the Conform scan is complete.

Note that the “Add” command will be disabled or invisible if one of the following is true:

1) The current user does not have the “Append to Table” privilege.

2) More than one Feature Table is currently open.

3) There is no currently unmatched feature.

Match Feature

Match Element will select the feature in the suggestion list that matches a design file element selected by the user. This is useful when the unknown feature should be changed to a feature that is nearby in the design file. The user must first select the element and accept it with a data-point for the command to work. Additionally, a warning will be displayed if the chosen element does not match an existing feature in the currently loaded feature tables.

Delete Feature

This tool will delete the currently selected element. It will not be enabled if the current element is a component of a complex chain - for example - a line within a cell, or a text element inside a text node.

Delete All Matching

This tool will delete all features like the current feature, from the present position in the active model, to the end of the current model. Caution should be exercised before doing this, as the matching is based on symbology only for simple element types such as line strings.

CADconform Draft

This command opens CADconform Draft (if it is not already open) using the existing feature tables currently loaded by Conform. The only difference between activating this tool from the Conform menu and activating it from the CADconform tool-box is that activation from the tool-box will open the Feature Table Manager.

Read Watermark

This tool allows the user to query an existing watermark cell. More information is available in the Read Watermark chapter.

Drawing Scale

Opens the “Drawing Scale” dialog box.

View Feature Tables

Opens the Feature Table Editor in read-only mode, showing the currently loaded feature tables.

Display Options

The display options affect the way an unknown feature is displayed by Conform and the Find Feature tool. The options on this dialog are summarised below:

Highlight

The unknown feature will be highlighted in the active view. The highlight colour will be inherited from MicroStation’s active highlight colour, defined by: “Settings > Design File > Color > Element Highlight Color”.

Select

The unknown feature will be selected. Note that this may appear to have the same visual effect as the “Highlight” option if the following toggle button is ticked in the MicroStation User Preferences: “Workspace > Preferences > Input > Highlight Selected Elements”. Note also that the existing selection set is cleared each time an unknown feature is selected. This makes it easy to use other MicroStation tools in conjunction with Conform, such as “Change Attributes” or “Delete Element”.

Flash

Flash will flash the element in the current highlight colour with a frequency of approximately half a second. If the flash option does not appear to work reliably under MicroStation 2004, check the “CADconform Installation Guide” for graphics card compatibility with this option.

Zoom

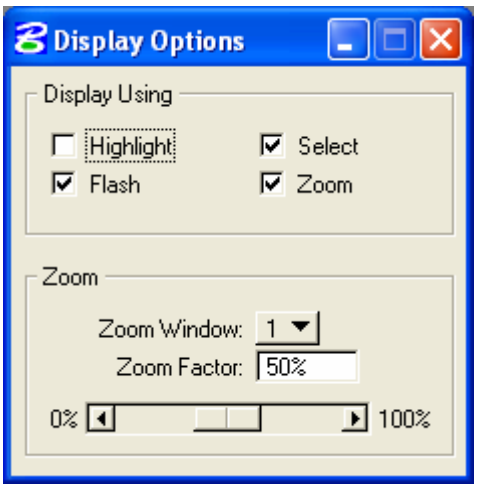
Zoom will modify the chosen view window (see below) extents to centre on the unknown feature. If the zoom feature does not appear to work for some elements, it is because the calculated window extents are smaller than MicroStation allows.

Zoom Window

This option chooses the appropriate MicroStation view window (1 to 8) to perform the zoom function. If the view window is not currently open, then CADconform will open it automatically.

Zoom Factor

The Zoom Factor affects how close the zoom window is to the unknown feature. It is sometimes convenient to have the zoom further out from the element to determine what the feature is supposed to be, in context to the elements around it. The zoom factor represents a proportion of the ratio between the element size and the window extents, with 0% being the minimum zoom and 100% being the maximum.

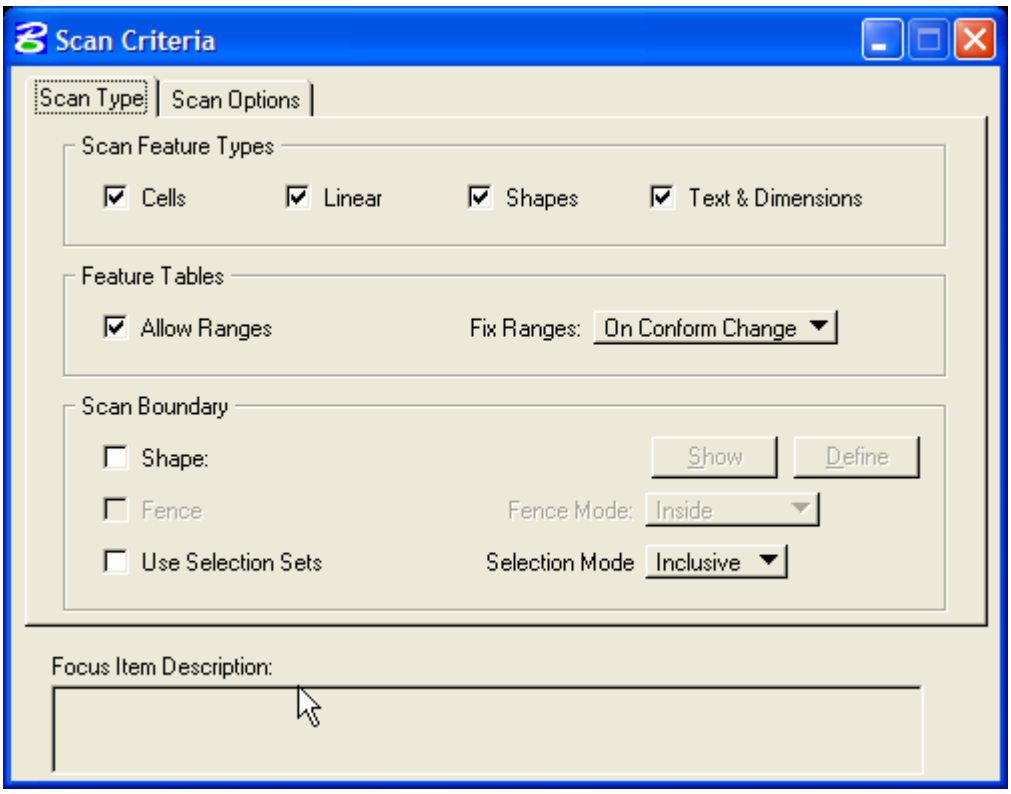


Scan Criteria

The Scan Criteria dialog box is opened from the Conform Window.

Below the tab pages on the Scan Criteria dialog box is a text-field, which explains the purpose of each item in the dialog box, when the item has the input focus.

The Scan Criteria dialog box contains tabs for types and options of scans.



The Scan Type tab page of the Scan Criteria Dialog Box

Scan Type

Scan Type defines the nature of the scan that Conform will carry out. It has the following groups of settings:

Scan Feature Types
Feature Tables
Scan Boundary

Scan Feature Types

These toggle buttons allow basic feature types to be turned ON or OFF mutually exclusively. This allows the drafter to fix specific problems individually. The available toggles are:

Cells (types 2 and 35)

Linear (types 3, 4, 11, 12 and 27-open)

Shapes (types 6, 14, 15 and 27-closed)

Text and Dimensions (types (7, 17, 33 and 37)

Feature Tables

The “Allow Ranges” toggle gives the user the ability to enable or disable symbology ranges globally. This means that a broad feature table specification, which allows many ranges of levels, colours, weights and styles can easily be tightened by turning this toggle off. This can be useful when dealing with legacy design files, which may have been created before tighter DGN standards were enforced. The available options are:

Allow Ranges

Turns ON or OFF Conform’s support of symbology ranges.

Fix Ranges

This option controls what to do with ranges on elements that are conformed. There are presently two options when fixing features with ranges:

1) Never Fix Ranges
2) On Conform Change

“Never fix ranges,” means that the element will be left in its original state when the user hits “Change” or “Change All”. This means that only bad criteria will be replaced with the correct symbology.

“On Conform Change” means that ranged data will be replaced by the proper symbology (the symbology used to draw the feature) on a Conform change. This can include criteria that was allowed because it was in a range.

Consider the following example. If a road centre-line is defined as follows:

Level: “Centre-line”

Colour: 5

Weight: 0 (also allow 1 in the range)

Style: 1

I run Conform on a design file and I find that I have drawn a centre-line with the wrong colour (6 instead of 5). I then press “Change” in Conform with the Centre-line feature highlighted as my replacement feature. The following results would occur:

1) If Fix Ranges was set to “Never Fix Ranges”, then my final centre-line will have a weight of 1.

2) If Fix Ranges was set to “On Conform Change”, then my final centre-line will have a weight of 0.

This example can be summarised by the table below:

———	Original Feature Definition	Unmatched Feature in CAD	Replaced with ‘Never Fix Ranges’	Replaced with ‘On Conform Change’
Level	Centre-line	Centre-line	Centre-line	Centre-line
Colour	5	6	5	5
Weight	0 (also allow 1)	1	1	0
Style	1	1	1	1

Scan Boundary

The “Scan Boundary” options define what region of the current model will be scanned during the conform process. There are three allowed methods for defining a scan area:

1) Shape

2) Fence

3) Use Selection Sets

Shape

The Shape Scan Boundary allows the user to define a boundary element that will restrict the conform process. This allows elements outside the boundary area to be ignored. Elements will only be checked that are inside this element, overlapping elements will only be allowed if the current fence mode is set to “overlap” or “void-overlap”.

To define the boundary shape, press the “Define” button, locate the plot border element and data-point to accept it. The feature name should now display to the right of the Clip by Shape toggle button. The plot border shape must be an existing feature in the current feature table for this to work; otherwise, an error message is displayed.

The shape will be defined as the first matching element for the Boundary Feature in the master design file / model and all attached reference files. The “Show” button allows the user to ensure that the correct element is used for the boundary, in the event that the boundary shape is not unique in all attached files. Note that the Shape Boundary is stored as a feature name, not an element file position. This means that the same definition can be used across differing design files.

Fence

If “Fence” is toggled ON, then Conform will only scan the active design file by the active fence, if a fence is defined. The Fence Mode is similar to the fence mode used during normal MicroStation operation. The “Void” modes will reverse the logic of the Fence, allowing the user to define an area that will not be conformed.

Selection Sets

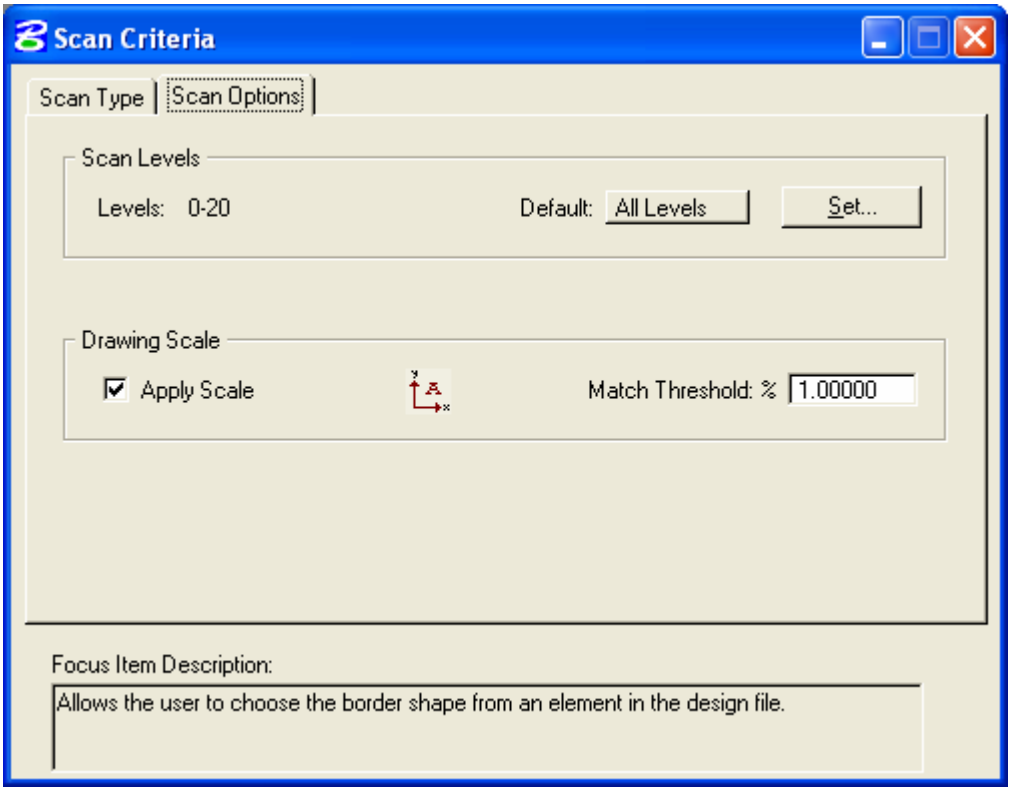
If “Use Selection Sets” is toggled ON and the selection mode is set to “Inclusive”, then Conform will only scan elements that are in the active selection set. If the selection mode is set to “Exclusive” then Conform will scan all elements that are not in the active selection set.

If a combination of fence, boundary shape or selection sets are defined, then the elements scanned are those that satisfy all criteria. For example, if a fence and a boundary shape are defined, then the scan boundary will only include areas that satisfy both criteria, which is the union of both scan areas.

Scan Options

Scan Options define user options specific to an individual run of Conform. This tab page has the following dialog items:

Scan Levels
Drawing Scale



The Scan Options tab page of the Scan Criteria Dialog Box

Scan Levels

Scan Levels defines which levels to scan and which levels to ignore. The current level settings will be summarised by a text string to the right of the “Set…” button. Note that in MicroStation v8, the levels will be summarised by level code, not level name.

Default

This button allows the user to specify which levels will be scanned by default when Conform is opened. This saves the user the bother of checking these settings each time a new design file is opened. The available options are:

From View
All Levels
Last Chosen

“From View” will copy the displayed levels from the active view. Levels turned off in the active view will not be conformed.

“All Levels” will default to Conform to All Levels. This is particularly useful in MicroStation v8, where the number and names of levels can vary between each model.

“Last Chosen” is useful if, for example, the same levels will consistently be ignored in each design file (commonly level 63). This means the user will only have to set this once using the “Set…” command.

Set…

The “Set” button will open a dialog allowing the user to choose levels by name from a Level List item, which will calculate the level codes automatically when the pop up dialog is dismissed. Under MicroStation J and below, the pop up dialog displays a Level Map item.

Drawing Scale

The drawing scale settings define the scale factor for scaleable criteria such as text size and line spacing. The drawing scale will affect both the way Conform interprets a text feature, and the way it replaces an invalid text feature. See the “Drawing Scale” chapter for more information.

Apply Scale

This toggle determines whether to apply the current drawing scale to all elements or not.

Drawing Scale Icon

This icon opens the Drawing Scale dialog box.

Match Threshold

“Match Threshold” is only used by the “Change All” command. If an invalid feature has incorrect scaleable criteria, then the match threshold can be used to also replace all elements that fall within the threshold tolerance.

For example, imagine a text feature has been defined with a text scale of 55mm, and our Match Threshold is set to 1.000%. If a text element is found in the design file that matches the symbology of the feature but has a text size of 55.5mm, then the element will be considered invalid.

If we then do a “Change All”, Conform will prompt us as to whether we want to fix any other text element that may have the same problem – i.e. correct symbology but a text size out by a factor of 1.000% or less. If we choose “Yes”, then all matching text with a text scale between 54.45mm and 55.55mm that doesn’t conform will be replaced with a size of 55mm.

If however, Conform finds a text element with a scale of 56mm and we do a Change All, we will not be prompted to correct text that is within the 1.000% range because the current element is out by 1.818%.

The match threshold is calculated as a proportion of the difference between what the text size is and what it should have been. Thus:

56 – 55 / 55 = 0.0181818

or 1.818%.

Read Watermark

Read Watermark will allow the user to query an existing Watermark or Signature cell. After this tool is activated, the user can choose the Watermark or Signature cell to query, and accept it with a data-point. The Watermark or Signature details are then displayed in the Watermark Info dialog box.

Watermark Cell

This is the name of the cell used for Watermarking or Signing. This cell name may differ for people in different departments of the same organisation.

Feature Table

Lists the Feature Tables used for certifying the design file.

Stamped by User

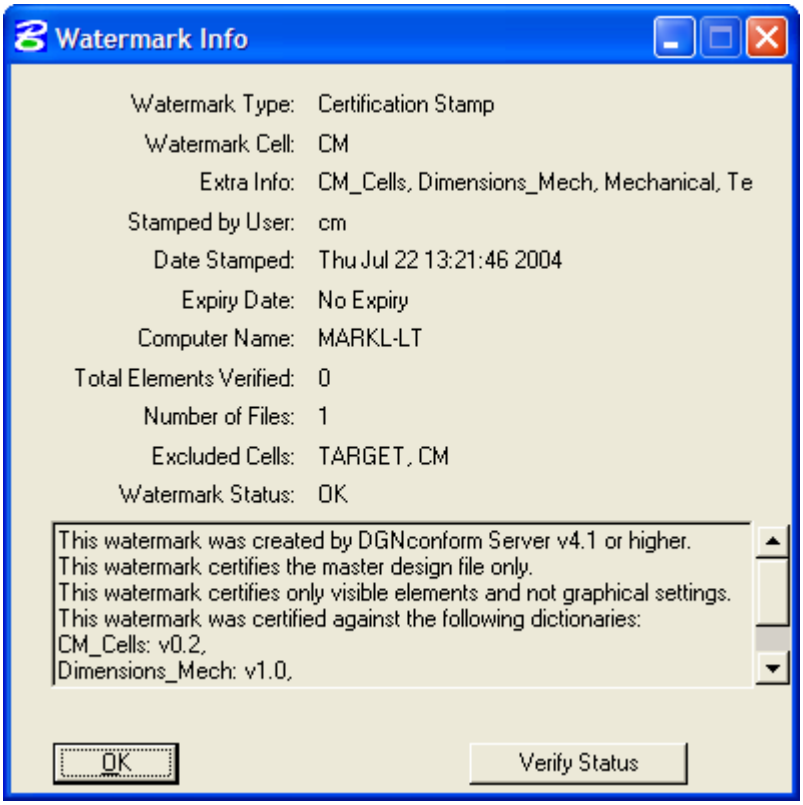
Lists the full name of the user who stamped the design file.

Date Stamped

Lists the time and date when the design file was stamped and certified.

Total Elements Verified

Represents the number of elements that were scanned and verified as correct when the design file was certified.



The Watermark Info Dialog Box

Number of Files

This number represents how many reference files were attached and certified as correct when the watermark was created.

Watermark Status

Represents the status of the watermark at the time CADconform last checked the watermark – any one of:

OK

INVALID

FILE MODIFIED

CONFORM FAILED

Note that if the design file has been modified since the last time the watermark was checked, then the Watermark Status might say “OK”, even though it isn’t. The “Verify Status” command can be used to ensure watermark integrity (see below).

Multi-line Text-Field

The multi-line text-field contains the following information about the watermark:

The version of CADconform that created it.

How many reference files were attached when the watermark was created.

What settings are certified by the watermark.

Then either:

If a watermark is used then the dictionaries the watermark was certified against eg certification cells

OR if a digital signature has been used the reason for signing will be given.

Verify Status

This command scans the design file to ensure the file has not been modified since it was certified. If the file has been modified, then the watermark will automatically invalidate itself and a single red line will appear through the watermark cell.

Chapter 10

The Report Generator

The Report Generator has three main purposes:

Generating Reports
Certifying Design Files
Reviewing Errors

Generating Reports

The Report Generator is used to create reports on a list of design files that do or do not match any of the features in the chosen Feature Tables. The Report is generated as a database table, either in the CADconform database, or in a separately chosen ODBC data source (as defined by the user). This database table can then be printed out, converted to a text file or other file format, or used to generate a customised report, by using a database application such as Microsoft Access or Oracle.

Several different report types can be generated for the chosen design files. They include:

A Feature Summary
An Error Summary
A Detailed Report of Each Error
A Watermark Report

Additionally, two compulsory reports are generated every time a report is produced:

A Report Information Table
A Design File Summary Table

Note that to overwrite an existing report, the user must be either:

An administrator, or
The creator of the report.

As with Conform, the Feature Tables must be chosen before a report can be generated. The chosen Feature Tables define which features are allowed for each design file in the report. Once the Feature Tables are chosen and imported using the Feature Table Manager, the Report Generator dialog box will open.

The Report Generator allows the user to process one or more design files for each report. Reference files for each design file can also be reported on by using the “Find References” tool. If reference files are processed, then the reference clipping

boundaries defined in each master file can optionally be observed such that only features within the reference clip boundary will be analysed. Similarly, if the reference files have been moved and scaled, then the coordinates displayed in the Report Table can also be transformed by these settings. Report Generation can be aborted at any time by pressing the reset button (usually the right mouse button).

Certifying Design Files

The Report Generator also has a “Certify” command. This command allows the user to flag each file as being approved, as long as it passes through the report with no errors. Reports are generated by both the “Report” and the “Certify” commands. Certification involves two automatic procedures:

WatermarkingTurning off unused levels

Watermarking

If a design file passes certification, then CADconform will attempt to place a Watermark cell in the design file. The watermark cell name is defined in the user’s profile, and must be a cell either found in the defined cell library, or in a library defined in the configuration variable “MS_CELLLIST”.

The first thing the CADconform Administrator needs to do (if Watermarking is to be used) is to create a “Marker” and a “Watermark” cell. This can be done using the usual MicroStation methods for cell creation. An example of a marker and watermark cell is given below. The origin and scale of the placed watermark cell will be picked up from one of two existing cells in the design file: a marker cell, or a watermark cell. This means that in order to watermark a design file, the user has to first place a marker cell into the design file. This cell will usually be placed in the title block area of the design file. Additionally, the administrator should make sure that both the marker cell and the watermark cell symbology are defined in the Feature Tables; otherwise, the file cannot be certified.

To avoid having to place the marker cell in every design file, the user can place the marker cell in a standard border sheet that is referenced in. If no marker cell is found in the active model, then CADconform will search all attached border sheets, and use the location and scale of the marker cell in the border sheet. In this case, the watermark cell with be placed in the master model on top of the referenced marker cell, and the marker will not be deleted from the reference file.

CADconform can automatically prompt the user to place a marker cell if one isn’t found in the current model during a report. See the Administrator’s Guide for more information.

If a file passes a certify operation, then a fresh watermark will replace the existing Marker cell. If the design file already has an existing watermark, then this cell will also be replaced, but only if the “Update Existing Watermarks” toggle is ON in the Advanced Options dialog box. If this option is OFF, then existing watermarks will be ignored, and new watermarks will only be placed on top of marker cells. If a file does not pass certification, then the watermark will not be placed. Additionally, if a valid watermark already exists in the file, it will become invalidated. Invalidation of a watermark can occur during three possible operations:

1) The user generates a report.

2) The user performs a “certify”.

3) The user runs the “Verify Watermark” command on the Watermark Info dialog box.

A watermark will only be invalidated if one of the following occurs:

1) An error occurs during a report or certify.

2) The design file has been modified since it was last certified.

If an error occurs during the report or certify, then the error will be logged in both the “Error Summary” and “All Errors” report. Additionally, if an existing watermark is invalidated, then the reason will be logged in the Watermark Report. Watermarks invalidated due to symbology errors will be represented by a cross over the watermark cell. Watermarks invalidated due to design file modification will be represented by a single diagonal line.

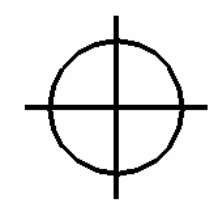


Figure 1: A Marker Cell

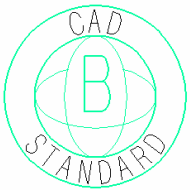


Figure 2: A Watermark Cell

Figure 3: A Failed Watermark

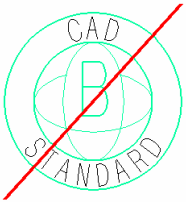


Figure 4: A File Modified Watermark

Text Substitution within Watermarks

CADconform supports the substitution of text elements at the time of certification by replacing user defined strings (as set out in configuration variables) with desired text strings. There are two text substitution options:

Dictionary/Table Version numberDate/Time

Dictionary/Table *Version Number*

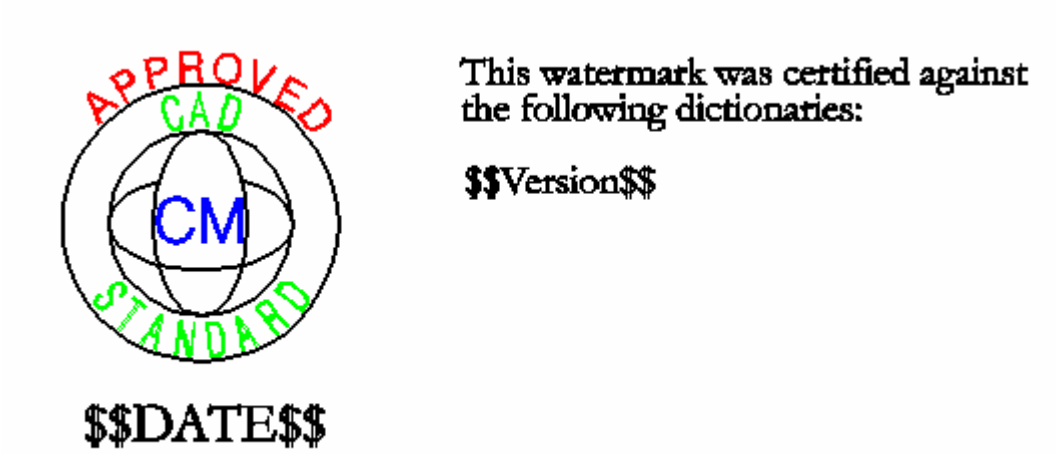
The string defined in _CADconform_REPLACETEXT_DICTVERSION_STRING will *be replaced by CADconform with the name and version numbers of the dictionaries the watermark was certified against. If the watermark was certified using multiple dictionaries, then the resultant element will be a text node that’s line spacing is defined by _CADconform_REPLACETEXT_LINESPACING.*

Date/Time

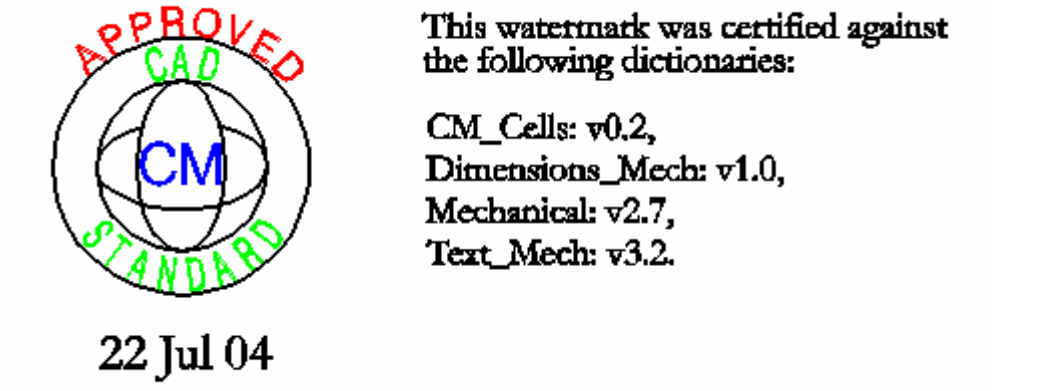
CADconform will replace the string defined in _CADconform_REPLACETEXT_DATE_STRING with a string that is formatted as d*efined in _CADconform_REPLACETEXT_DATE_FORMAT*. This time or date is calculated using the system time and date at the time of certification.

Text Substitution by Example

The following example uses both dictionary version and date substitution to show how CADconform processes text substitution. The first step in using text substitution is to create a cell that contains the strings the user wishes to replace. This cell then gets defined in the user manager dialog like any other watermark cell. An example of a watermark cell that would use text substation is:



This cell will then be modified by CADconform at the time of certification and all the user defined strings will be replaced with the relevant replacement text, as shown below:



Turning off Levels

The second part of the certification approval process is the automatic turning off unwanted levels in each view in the design file. The levels to be turned off are defined in the administrator configuration file "CADconform.cfg" by the variable "_CADconform_DISPLAY_LEVELS_OFF". If no levels are defined by this variable, then this operation is effectively skipped.

Reviewing Errors

The Report Manager can be used to review errors by viewing the error reports. By double-clicking on a report row, CADconform will find the first matching feature in the relevant design file. See the Report Viewer chapter for more information.

Dialog Options

The Report Generator dialog consists of three dialog components:

The Menu Bar
IconsDialog Items

The Menu Bar

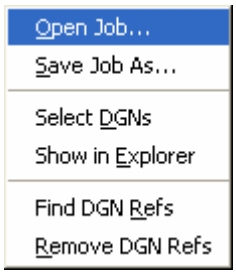
There are four menus in the menu bar:

FileEditToolsOptions

The File menu

The File menu has the following commands:

Open Job…Save Job As…Select CADsShow in ExplorerFind DGN RefsRemove DGN Refs



Open Job

Opens a File Selector dialog to allow the user to open an existing Report Job File. Report Job files are basically settings files for batch reporting, they allow individual report runs to be saved to a text file for later reuse.

Save Job As

Allows the current Report settings (including Advanced Options, levels and design files) to be saved to a file for later reuse.

Select *DGNs*

Allows the user to edit the list of design files to report on. Note that the user must choose "Add" to add design files to the list. The current list of design files to report on is displayed in the "Design Files" list box on the Report Generator dialog box.

Show in Explorer

This command will open Windows Explorer and select the design file currently selected in the Report Generator list-box.

Find *DGN Refs*

This command will find all of the reference files for the currently selected design files. Multiple design files can be selected by holding down <SHIFT> or <CTRL>. These reference files will be added to the Design Files list box, underneath their respective master design files and indented once for each level of nesting.

By default, the nesting level is inherited from the selected design file. To find the next level of nested reference files, select the required reference files and start the "Find DGN Refs" command again.

Remove *DGN Refs*

This command removes all reference files from the selected master files. If the selected row is a reference file, it will also be removed.

The Edit Menu

The Edit Menu consist of the following items:

Select
Remove
Set Levels

Select

Select facilitates the selection of design files in the list box. There are three options:

All
None
Invert

Remove

This command removes currently selected design files from the list of design files to report on. This can be useful when used in conjunction with the "Find Refs" command, since commonly the standard border sheet does not need to be checked for every drawing.

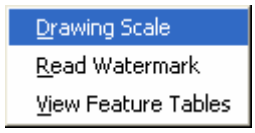
Set Levels

Defines the levels to scan for each design file during a Report or Certify. These levels can be chosen individually for each design file, via a Level List item in MicroStation v8, or a Level Map in MicroStation J or below.

The Tools Menu

The Tools Menu provides short-cuts to the following tools:

Set Drawing Scale
Read Watermark
View Feature Tables



The Options Menu

There is only one item in this menu: "Advanced Options".

Icons

Most of the icons on the Report Generator dialog box are shortcuts for the menu commands. The icons are described below:

Open Job

Opens a Report Job file. See the "File > Open Job…" menu command.

Save Job

Saves a Report Job file. See the "File > Save Job…" menu command.

Add Design Files

Opens the File Selector to add and remove design files from the report. See the "File > Select CADs…" menu command.

Remove Design Files

Removes the selected design files from the list. See the "Edit > Remove" menu command.

 **Find Reference Files**

Finds the reference files of the selected design files. See the "File > Find DGN Refs" menu command.

 ***Set Levels***

Sets the reported levels of the selected design files. See the "Edit > Set Levels" menu command.

 ***Drawing Scale***

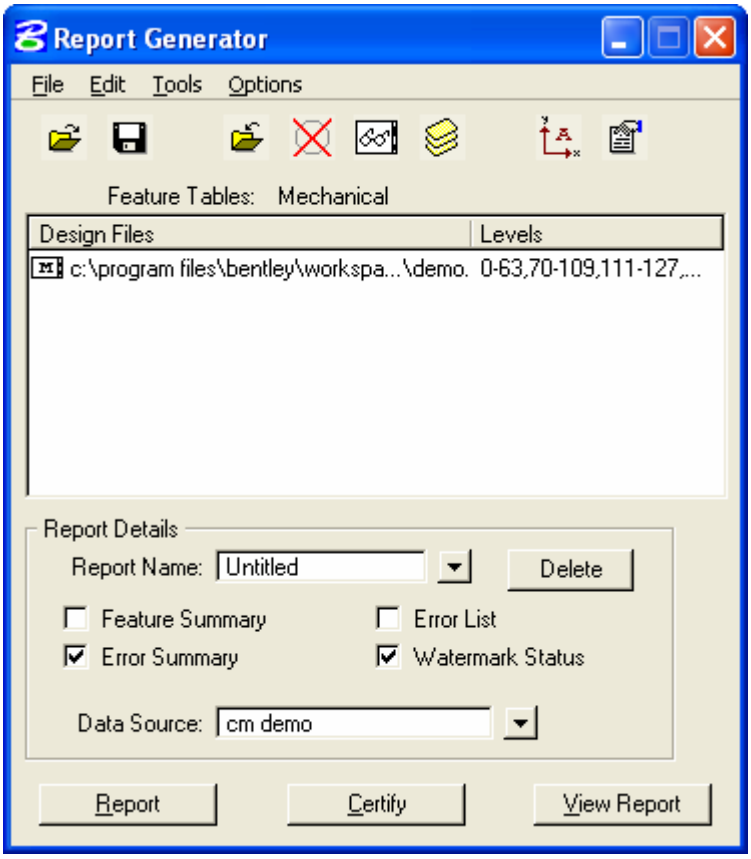
Opens the Drawing Scale dialog box.

 ***Advanced Options***

Opens the "Advanced Options" dialog box.

Dialog Items

Below is a summary of the remaining items on the Report Generator dialog box.

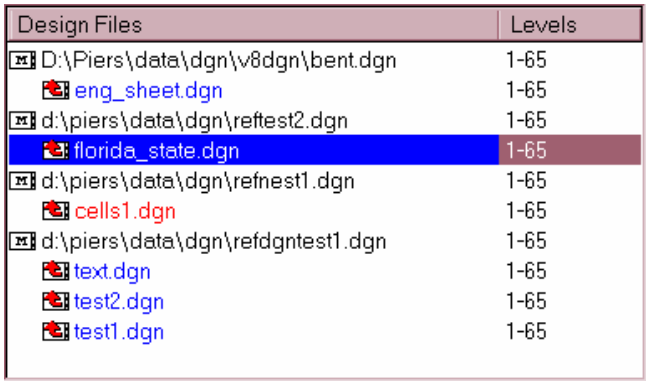


The Report Generator Dialog Box

Design Files List Box

Displays the chosen design files to generate the report on. This list can be edited by clicking the "Add / Remove" button. Master design files are displayed in black text; referenced design files are displayed in blue, and missing reference files are displayed in red. Reference files are indented once for each level of nesting.

The right-most column of this list box also contains the levels to report on for each design file. To change the level for a specific design file, double-click the corresponding row / cell.



The Design Files List Box showing Master models in black, reference models in blue and missing references in red.

Report Name

Lists the name of the report to generate. This combo box contains the names of all existing reports so that the user can view or overwrite an existing report, or change the name to create a new report.

Feature Summary

Generates a report listing each unique feature and how many instances of each were found in every design file.

All Errors

Lists the symbology, coordinates and file position of every element that did not match any features in the Feature Tables. Note that this report can be very time consuming to produce in files with many errors.

Error Summary

Produces a summary of each error in the design file and how many times it occurred. This is usually the most useful of the report types.

Watermarks

Lists the status of existing watermarks in every design file.

Data Source

The name of the ODBC data source to write the report to. By default, this will be the CADconform database, but it can be set to any data source defined in the ODBC Data Sources in "Windows > Control Panel". It may be quicker and easier for users to define their own report database to use locally, if reports are not required to be shared amongst users. This can also be useful to ensure that the CADconform database does not get too large.

Report

This button begins generation of the report. A completion bar will open to display the progress of the Report Generator for each design file (and reference file) in the design file list. Files that do not pass with zero errors will have their watermarks invalidated if they exist.

Certify

Begins certification of the design files. This process is identical to the Report command, but in addition it will:

Turn off specified levels in all views (see "_CADconform_DISPLAY_LEVELS_OFF").

Place or update watermark cells if there are zero errors.

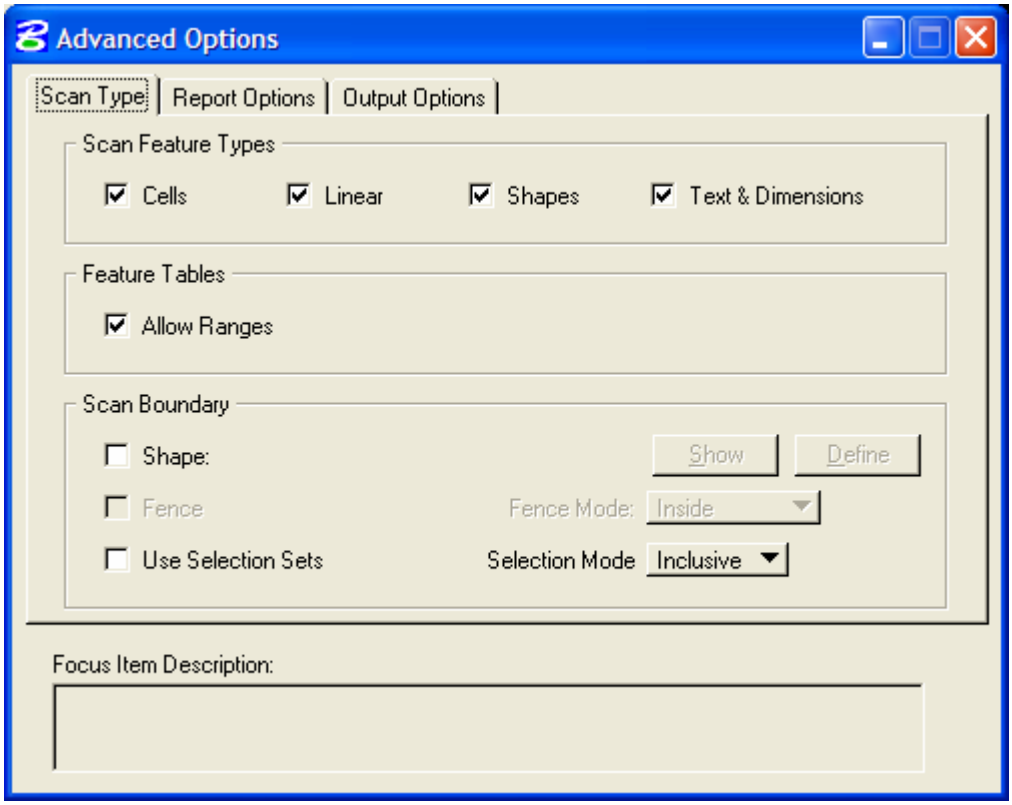
View Report

Opens the Report Viewer dialog box.

Advanced Options

The Advanced Options dialog box is used to control how each design file is scanned for the report, which features are being scanned and what the scan area is. There are three tab pages on the Advanced Options dialog box:

Scan Type
Report Options
Output Options

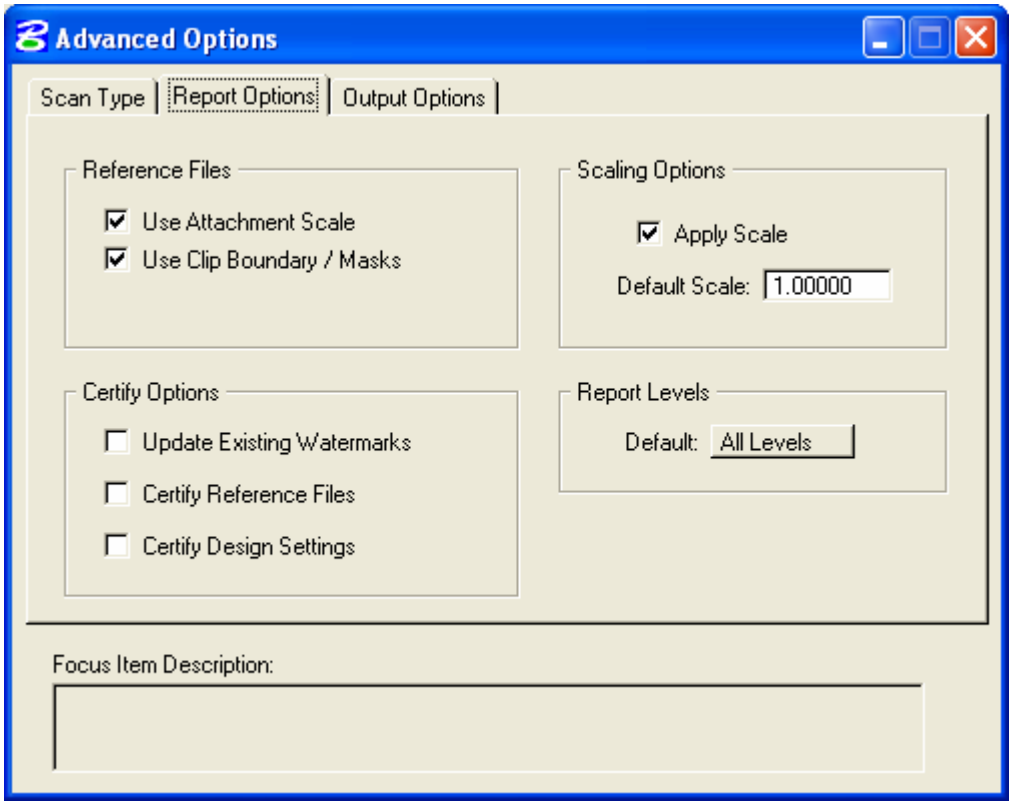


Scan Type Tab Page

The Scan Type options are analogous to the Scan Type options on Conform*'s Scan Criteria dialog box. See the Scan Criteria* chapter for more information.

The Report Options Tab Page

This tab page contains options for interpreting reference files, applying a default scale and treating existing watermark cells. These options are described below:



Reference Files

These options only affect design files / models that are attached to other files that are being reported on (i.e. displayed in blue in the list box).

Use Attachment Scale

If this toggle is ON, then the scale of text, cells and dimensions in reference files will be read scaled by the attachment scale and working units of the master file. This toggle should usually be ON.

Use Clip Boundary / Masks

If this toggle is ON, then clip boundaries and clip masks are observed when reporting on reference files. This means that only the portions of the reference file that are visible in the master file will be reported on, and areas clipped out will be ignored.

Scaling Options

These options affect the interpreted scale of master design files. Note that unlike the FTE, Conform and Draft tools, there is no overall scale. This is because the multiple design files reported on could have numerous different scales, which should be interpreted automatically from either the border sheet or the saved scale.

Apply Scale

If turned ON, this toggle will scale text features by the Drawing Scale as defined by the standard border sheet scale or saved scale for each drawing. If this toggle is ON and no saved scale is detected or no standard border sheets are attac*hed, then the default scale is used.*

Default Scale

This option defines the scale to use if no Drawing Scale could be read from the standard border sheets, and Apply Scale is ON.

Certify Options

These options only affect the certification process, assuming that the user has been granted this privilege.

Update Existing Watermarks

If the user is certifying design files, this toggle will allow existing valid watermarks to be rewritten with the current time, date, user and feature tables. If the toggle is ON and the file has no errors, then invalidated stamps will be updated to valid status. If this toggle is OFF, then existing valid and invalid watermarks will be left alone.

Certify Reference Files

This option makes the watermark include the reference file information for each design file certified. This means that the watermark will invalidate with the "File Modified" message if the master model is altered or ANY of the reference files attached to it.

Certify Design Settings

This option adds the design settings information to the watermark, such that the watermark will become invalidated if any of these settings are subsequently changed after the watermark is created. These settings include:

Shared cell Definitions (type 34)
Raster References (type 90)
Group Data (type 5)
Colour table data
Reference file settings
View Settings: displayed levels, saved views, current view, displayed view

Pattern information
Auxiliary Coordinate System data
MicroStation v8/2004 table info (type 96 elements): level tables, text style, level filters, line style names, line style definitions, etc.

Note: In MicroStation v8/2004 font tables, dimension styles and multi-line styles (type 96 elements) can not checked as they are re-written every time a DGN is closed.

If this option is not enabled, then the watermark will only invalidate when changes are made to visible (graphic) elements in the design file.

Under MicroStation v8, the automatic process of placing a certification cell may invalidate the file the first time it is placed. This can happen if any of the components of the certification cell contain information not already contained in the design file. For example, if the cell contains a piece of text on a level named "Watermark Text", and this level did not exist prior to certification, then the level table will become rewritten, which would cause the watermark to become invalid upon next verifying the watermark status.

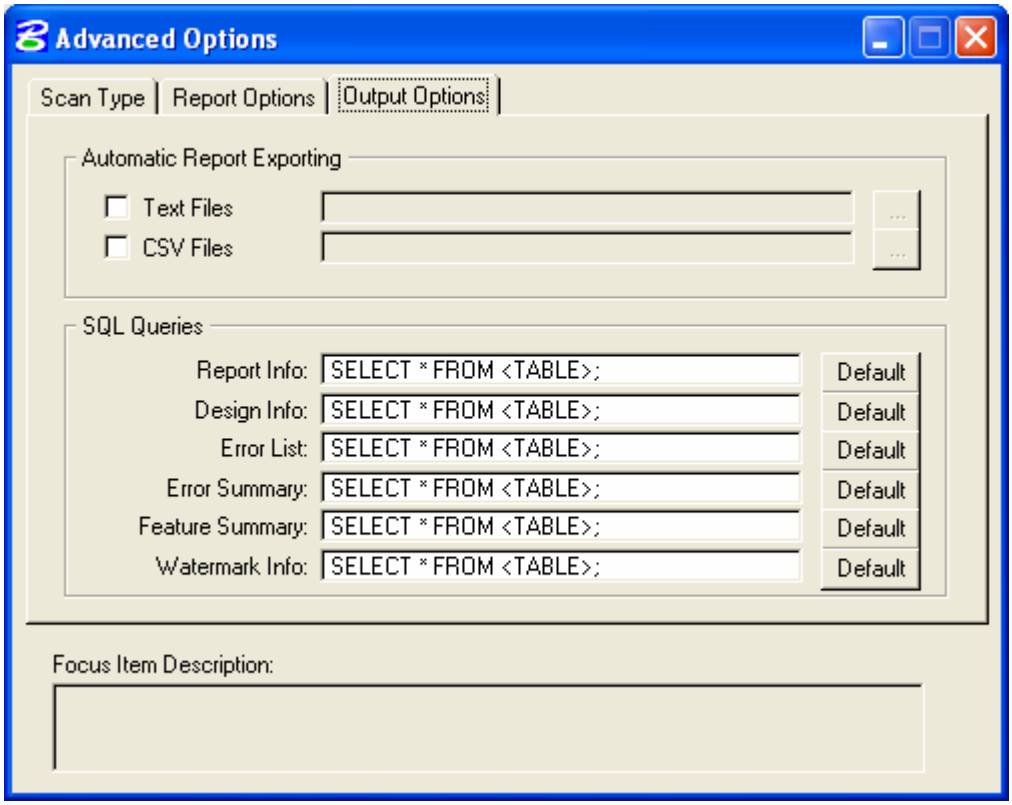
The solution to this problem is to place the watermark cell manually and then delete it, as this will copy the necessary unique features to the current model. If the various settings such as Text Styles and Levels (see list above) are already in the model or an attached CADLIB file, then this problem will not occur.

Report Levels

This option defines what the default will be for reported levels when a design file is added to a batch report. The options are analogous to the Default Levels options on the Scan Criteria dialog box in Conform.

The Output Options Tab Page

The Output Option tab page allows the user to define an output directory for automatically exporting text file reports. This is an optional extra step performed after the reports have been written to the database. The options on this page are described below.



Automatic Report Exporting

The options in this group-box define whether automatic report exporting is turned ON, and if so, what format they are in and where they are created.

Text Files

If this toggle is ON, then plain ASCII text files will be created in the specified directory. The required path can be chosen using the "…" browse button. Text file reports are human-readable formatted text files suitable for printing or viewing. Text files contain a header describing each column at the top of the file.

CSV Files

If this toggle is ON, then ASCII CSV (Comma Separated Values) files will be created in the specified directory. The required path can be chosen using the "…" browse button. CSV file reports are suitable for importing into 3rd party applications (such as spreadsheets). They do not contain a header describing each column, and thus should only be used for data sharing between applications.

SQL Queries

This group-box contains settings for post-processing of created report tables. SQL Queries can be customised to display information in certain ways. The SQL Query is called:

1) When a report is viewed in the Report Viewer

2) When a report is exported automatically to a text file

If you do not understand SQL Syntax, then it is probably safer to leave these settings at their default values. The dialog items within this group-box have the following purpose:

The keyword <TABLE> in the SQL Query is replaced at run-time with the actual table name for the desired report type. For example, <TABLE> would be replaced with 'Report_Sum_Untitled' for the Error Summary report for a report named 'Untitled'.

Report Info

This text-field defines the SQL query to perform to retrieve the Information Report.

Design Info

This text-field defines the SQL query to perform to retrieve the Design File Information Report.

Error List

This text-field defines the SQL query to perform to retrieve the Individual Error List Report.

Error Summary

This text-field defines the SQL query to perform to retrieve the Error Summary Report.

Feature Summary

This text-field defines the SQL query to perform to retrieve the Feature Summary Report.

Watermark Info

This text-field defines the SQL query to perform to retrieve the Watermark Status Report.

Default

This push-button returns the associated text-field to the default SQL Query, which simply queries all columns in the table.

A few examples of customised SQL queries follow.

To make the Design Info report only display the design file name, levels used and number of errors:

```
SELECT Design_File, MSLevel, Num_Errors  FROM <TABLE>;
```

To make the Feature Summary report only show features on the level 'REBAR':

```
SELECT * FROM <TABLE> WHERE Symbology LIKE '%lv='REBAR'%';
```

To sort the Error Summary in order of most errors at the top:

```
SELECT * FROM <TABLE> ORDER BY Num_Errors DESC;
```

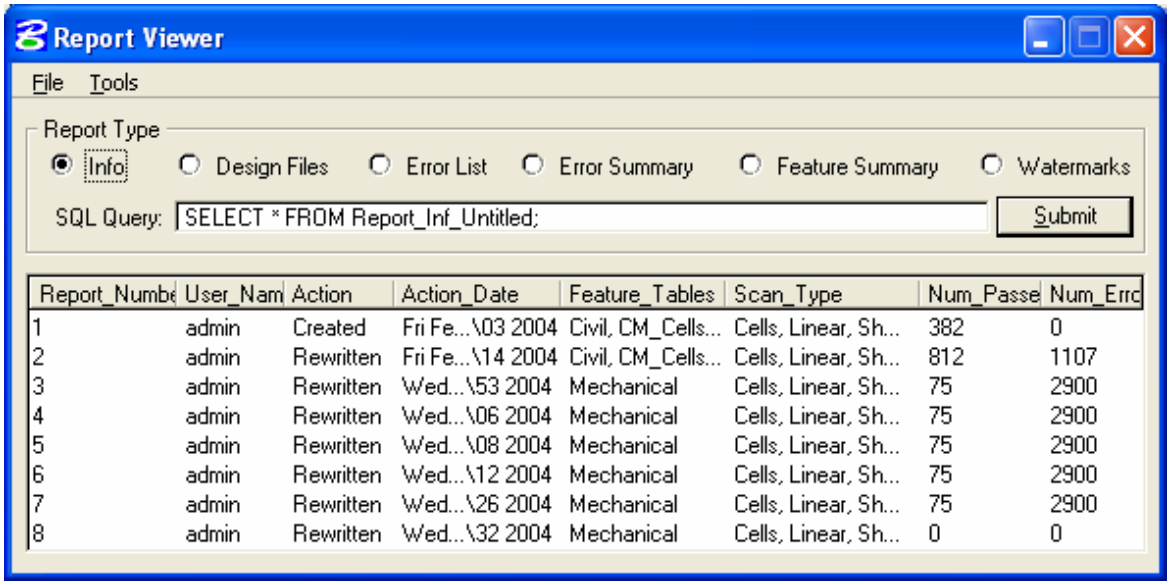
The Report Viewer

The Report Viewer allows a simple way for the user to view the contents of a report without resorting to opening the database from within a database application. This is particularly advantageous for users who do not have a database application loaded on their machine.

The Report Viewer also provides a method of interactively reviewing errors in the report, and exporting reports to text files or comma separated value files (CSV) for importing into Spreadsheet applications such as Microsoft Excel.

The Report Viewer dialog and its columns are fully resizable, and have the following dialog components:

Menu Bar
Dialog Items



The Feature Summary Report displayed on the Report Viewer Dialog Box.

Menu Bar

There are two menus on the menu bar:

File
Tools

File Menu

The File menu has the following menu items:

Edit
Export

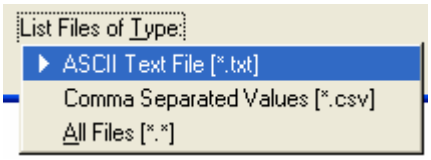
Edit

Edit will open a file requestor dialog, allowing the specified file to be opened using the Windows default application associated with the file extension. This is useful for viewing reports after they have been exported to an external file. If, for example, NotePad is associated with "TXT" files, then the specified file will be opened in NotePad.

Export

Export can be used to export the data currently displayed in the Report Viewer to an external file. There are two formats currently supported by the Export command, which can be chosen by setting the File Extension option button on the File Requestor dialog box:

CSV (Comma Separated Values)
TXT (ASCII Text File)



Tools Menu

The Tools menu has two menu items:

Find Feature
Read Watermark

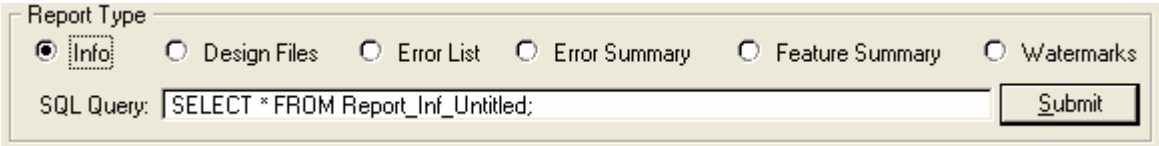
Find Feature

The only currently supported mode for the Locate Tool is "Find Feature", which will locate the corresponding feature for the selected row in the Report Viewer. For the Feature or Error Summary reports – *where one row may refer to multiple features – the first match is displayed. Subsequent matches can then be found by pressing the "Next" button on the Tool Settings window. Features in the "Individual Error" report will always be found correctly (assuming the file hasn't been modified since the report was generated), because CADconform uses the element's file position to uniquely locate the element.*

The Find Feature tool can also be activated by double-clicking on a list box row in the Report Viewer dialog box.

Dialog Items

The Report Viewer dialog has the following dialog items:



Report Type

Allows the user to choose which report to view. These radio buttons automatically set the SQL Query to show the corresponding report type and display the query results in the list box. The SQL Query used by default can be defined in the "Output Options" tab page of the Advanced Options dialog box.

SQL Query

Allows the user to enter an SQL query to create their own data sets. All standard ANSI SQL commands are supported, for example:

Example 1

SELECT Feature, Num_Features FROM Report_Fea_new2 ORDER BY Num_Features ASC

Will display only the "Feature" and "Num_Features" columns from a Feature Summary report named "New2", sorted in order of most common features at the top.

Example 2

SELECT Xcoord, YCoord FROM Report_All_new2 WHERE Symbology LIKE '%lv=6%co=5%'

Will display the X and Y coordinates of all mismatching features on Level 6 that have a colour of 5.

For more information on SQL statements, consult an SQL manual or search on the Internet for:

ANSI + SQL + TUTORIAL

Submit

Submits the current SQL Query. If the SQL query does not result in any database rows being returned, then the column display name will change to "SQL Query returned no rows".

List Box

The list box displays the results of the last SQL Query. The number of columns displayed in the list box depends on the number of columns resulting from the last query. All columns are resizable, and will display (in MicroStation v8) truncated with ellipses ("…") if more information is contained in a cell than can be displayed. Double-clicking on a list box row will activate the "Find Feature" tool and display the feature in the current display view.

Under MicroStation v8, the following enhancements are available:

Hovering the mouse over a truncated cell will result in the full contents appearing in a balloon help window.

Columns can be automatically sorted by clicking on the column name.

Chapter 11

The Draft Menu

The Drafting Menu (also simply "Draft") allows the user to use CADconform to create design files to specification from the beginning of the drafting phase. It works by creating a pull down menu for every selected Feature Table, with a command to place every feature simply by selecting it. For example, selecting a linear feature from the menu might start the PLACE SMARTLINE command. A default command is associated with every feature, when a command isn't already defined by the user. The default commands for each feature are as follows:

Linear: PLACE SMARTLINE
Cell: PLACE CELL ICON
Shape: PLACE SHAPE ICON
Text: PLACE DIALOG TEXT ICON
Dimension: DIMENSION ELEMENT

The other two feature types have the following effect on the drafting menu:

1) Command: Defines any menu command that is not associated with any feature

2) Separator: Defines a horizontal separator bar to place in the menu.

An example of a command that is not associated with a feature would be a command that simply turned a particular level on or off, e.g. "on=62". Although this isn't considered as a feature by the Conformer or the Report Generator, we may still want this command to appear in our drafting menu. Similarly, a separator will not be considered a feature for validation purposes, but can still be defined for placement in a menu.

User Interface

The Draft tool is made to be as flexible as possible, since it is the tool used most often in CADconform. The Draft dialog is resizable, dockable, context sensitive and supports tear-off menus. Various components of the Draft window can be enabled and disabled by resizing the window larger or smaller, or docking the window to the top or bottom of the MicroStation window. Additionally, the behaviour of Draft will change depending on what other windows are open. For example, Draft can act as a suggestion list chooser for Conform if a feature is activated in Draft when the Conform window is open. Draft itself can also act as a feature conformer if a selection set exists before a feature is activated.

Note that because various components of the Draft interface can be disabled, there are equivalent commands duplicated in the list-box, tool-bar, pop-up menu and menu-bar.

Key-in Commands

A key-in command is created for every criteria enabled for that feature. This means that if a feature is on level 20, then the key-in "lv=20" is appended to the key-in. CADconform adds key-ins to MicroStation's command table where an equivalent key-in doesn't already exist. For example, there is no standard key-in to define the dimension line style, so CADconform adds a key-in (while it is loaded) to the command table: "dimension style". This means that when this feature is activated by Draft, the correct dimension style is set prior to the user placing the dimension.

Tools and Settings

The final menu in the Drafting Menu is always the "Tools" menu, which allows the user to control various settings and commands available to the Draft tool. Note that many aspects of the Drafting menu can be controlled through the Tools menu, as well as interactively through the mouse. For example, the Draft menu can be docked and undocked by either dragging it to or from the top or bottom docking position, or by using the "Docking" menu items in the Tools menu.

Various components of the Draft menu can be disabled or turned off. For example, the menu bar can be turned off if the list box is active, or vice versa. These components can be reactivated by either resizing the dialog large enough to accommodate them, or activating them in either the "Tools" menu or the popup "Settings" menu. The popup settings menu can be opened by right-clicking on the list box, if it is displayed. If neither the Tools menu nor the list box is displayed, then the Tools menu can be accessed by undocking and resizing the Draft menu large enough that the list box automatically appears.

A feature in the Draft menu can be activated one of two ways:

1) By choosing it in the menu

2) By double-clicking the list-box row

Draft menus can also be torn off into separate menus with independent settings. This can help to break up large Feature Tables into separate focus areas or component groups for easy access when drafting.

Note that there is a current limitation of:

250 sub menus per Draft dialog
30 tear-off Draft dialogs open at any time

In practise, it would be unlikely that these limitations would be reached by any Feature Table. As a comparison, MicroStation's main menu contains only around 35 sub-menus.

Some of the Drafting menu settings are remembered between runs automatically. Settings automatically stored include only global settings that affect all Feature Tables. This includes:

Window size
Docking position and height
Menu Length (short or long)
Interface options (list box, menu bar, tool box)

Options that are specific to individual Feature Tables are not automatically saved, but can be written back to the local cache on the user's machine using the "Save Settings" command. These options include:

Tear off menus
Hidden features
Open feature groups
Menus Across (ON or OFF)

These Feature Table settings will be lost when CADconform needs to update the local cache due to a modification of the Feature Table on the server. To make these changes global to all users, these same settings can be applied in the Feature Table Editor.

Modes of Operation

CADconform Draft effectively has three modes of operation:

1) Draft by Feature

2) Conform to Feature

3) Conform Change to Feature

Draft by Feature

Draft by Feature is the method discussed earlier, where the key-in associated with a feature is activated to allow the feature to be drawn. This works by starting the placement command (e.g. "PLACE SMARTLINE") associated with that feature.

To draw a given feature, follow these steps:

1) Clear the current selection set, if there is one.

2) Choose the feature by clicking once on the menu, or by double-clicking on the list box.

The Draft window title will change to represent the current feature name, eg: "Place Rebar Line".

Conform by Feature

Conform by Feature allows an existing element to be conformed to match a chosen feature. This could be used to change a feature from one type to another (e.g. "Concrete Object Line" to "Concrete Hidden Line") or to fix an incorrectly drawn feature (e.g. changing an unknown feature to a known feature – "Concrete Object Line").

To conform an existing element (or selection of elements), follow these steps:

1) Clear the current selection set, if there is one.

2) Select the elements you wish to conform.

3) Choose the feature by clicking once on the menu, or by double-clicking on the list box.

4) Enter a data-point to confirm the change.

Prior to entering the confirmation data-point, the Draft window title will change to represent the current feature name, eg: "Conform to Rebar Line".

Conform Change to Feature

Conform Change by Feature is the default Draft command that occurs when the Conform window is open. Both "Conform to Feature" and "Draft by Feature" are effectively disabled when Conform is open. Conform Change by Feature allows the user to choose the "Change To" feature in Conform from the Draft window. This can be useful when the suggested features list does not contain the correct feature near the top of the suggestion list. Rather than scroll through potentially many features to find the correct one, it can be quicker to simply choose it from the Draft menu. For example:

1) Open both Conform and Draft

2) Start a scan and find an unknown feature

3) Choose a feature in Draft

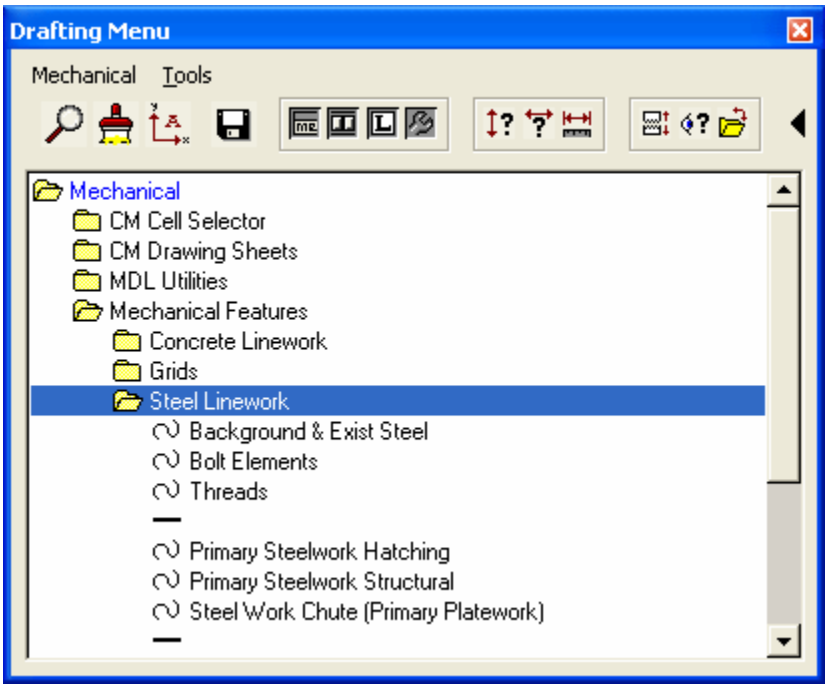
4) Conform will now automatically select that feature in the Change To suggestion list

Note that the Draft window title will change to: "Conform Change to Rebar Line", if a feature called "Rebar Line" is chosen.

Interface

The components of the Draft menu that are not built from the Feature Tables are described below:

Menu Bar
Pop-up Menu
Tool-Bar
List-Box


Drafting Menu
Mechanical
Tools
Mechanical
CM Cell Selector
CM Drawing Sheets
MDL Utilities
Mechanical Features
Concrete Linework
Grids
Steel Linework
Background & Exist Steel
Bolt Elements
Threads
Primary Steelwork Hatching
Primary Steelwork Structural
Steel Work Chute (Primary Platework)

Menu Bar

A new menu is created across the Draft menu for every feature group in the chosen Feature Tables when:

1) The group is the root level of a Feature Table.

2) The group is a child of the root level of a Feature Table and the "Menu Width > Menus Across" setting is ON.

These settings are discussed further on in this chapter.

Groups and features hierarchically below these parent menus are displayed as sub-menus and menu items inside these main menus.

Tear Off Menus

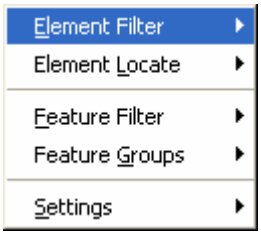
Each menu and sub-menu in the menu bar has a "Tear Off" option as the last item in the menu. The Tear Off command will flag the parent feature group as being a new menu. If there is a tick next to this option, then it means the menu has already been torn off. Tear off menus can be defined by either:

The "New Menu" option in the Feature Table Editor
The "Tear Off" option in the menu bar
The "Tear Off" toggle button in the tool box

The Tools Menu

The "Tools" menu is automatically added to the Draft dialog as the last menu across the dialog. The tools menu has the following menu items:

Element Filter
Element Locate
Feature Filter
Feature Groups
Settings



Element Filter

This setting controls whether the Element Filter is currently ON or OFF.

Element Locate

This tool starts the Locate Tool with one of the following modes:

Identify Feature
Add Features
Find Feature
Read Watermark

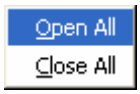
These settings are identical to the Locate Tools in the *Feature Table Editor*, apart from the *"Add Feature" command, which only temporarily adds features to the Feature Table. Temporary features appear in red in the list-box, and will be automatically deleted when any other tool in open.*

Feature Filter

This option can be used to hide features from the Drafting Menu that do not currently exist in the active model. These settings are very similar to the Feature Filter in the *Feature Table Editor*.

Feature Groups

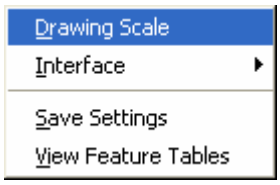
This menu allows global control of the open status of all feature groups. Groups can be either all opened or all closed.



Settings

The Settings menu comprises various options that effect the display and behaviour of the Draft menu. The items in this sub-menu are:

Drawing Scale
Interface
Save Settings
View Feature Tables



Drawing Scale

As with the Drawing Scale defined in the Feature Table Editor, Conform and the Report Generator, the Drawing Scale will act as a scale multiplier for text size, line spacing, cell scale and line style scale. See the Drawing Scale chapter for more information.

Interface

The interface sub-menu contains options controlling the look and feel of the Draft menu. The sub-menus are:

List-Box
Menu Bar
Tool-Bar
Tools Menu
Fit Dialog Width
Menus Across
Menu Length
Dock at Top
Dock at Bottom
Reorganise Docking
Dock Height

List-Box

This toggle item defines whether the list box is displayed below the menu or not. It can also be activated or deactivated by resizing the Draft dialog along the Y-axis, if the menu bar is not displayed.

Menu Bar

Toggles the display of the menu bar. If the menu bar is toggled OFF, then features can be activated by double-clicking the list box row. Note that either the menu bar or the list box can be turned OFF, but never both.

Tool-Bar

Toggles the display of the icon toolbox. This can also be activated by resizing the dialog in the Y-axis.

Tools Menu

The toggle controls display of the Tools menu in the menu bar. This is particularly useful where multiple docked Draft dialog boxes are open, and optimising "screen real estate" is important. Once deactivated, the Tools Menu can be activated again by right-clicking on the list box to access the Popup menu.

Fit Dialog Width

This command automatically fits the width of the Draft dialog to the smallest size that displays all icons of the tool bar, all menus of the menu bar, and the longest feature name of the displayed list-box.

Menus Across

This toggle controls whether the second level in the group hierarchy (the child groups of the root level) display across the Draft menu to form a new menu for each group, or whether they are all put in one menu under the root level.

Menu Length

Menu Length allows the user to toggle between "Short" and "Long" menus. Short Menus are menus that have some features hidden from view (features or groups that were flagged with "Hide"), whereas Long Menus display every feature in the Feature Table, regardless of the Hide status.

This is useful to reduce clutter on menus for features that are a necessary part of your DGN standards but either:

1) Never manually drafted (e.g. Watermark cells)

2) Rarely used

In Short Menu mode, hidden features and groups do not appear in either the list box or the menu bar. In Long Menu mode, hidden features will display in the menu and the list box, although the list box row will appear grey to notify the user that the feature is not normally displayed.

Dock at Top

This menu item will dock the selected Draft dialog at the top of the MicroStation window.

Dock at Bottom

This menu item will dock the selected Draft dialog at the bottom of the MicroStation window.

Reorganise Docking

This command reorganises the application area of MicroStation, undocking and re-docking all docked toolboxes. This option is only occasionally necessary when docked windows overlap incorrectly.

Dock Height

This option is only available if the selected Draft dialog is currently docked. It allows the pixel height of the docked window to be set, between 12 and 32 pixels. The correct size is usually either 25 or 31 pixels, but can depend on various factors such as:

Font Size
Tool Size (Small or Large as set in Preferences)
Dock Position (top or bottom)
The height of other docked windows

Due to these various unpredictable influences on the correct docking height, it is left up to the user to specify an ideal size based on their work environment.

Save Settings

Save Settings will save the current Feature Table back to the user's local cache directory. The settings saved by this operation are:

Opened / Closed status of feature groups
Show / Hide status of features and groups
The "Tear Off" status of feature groups

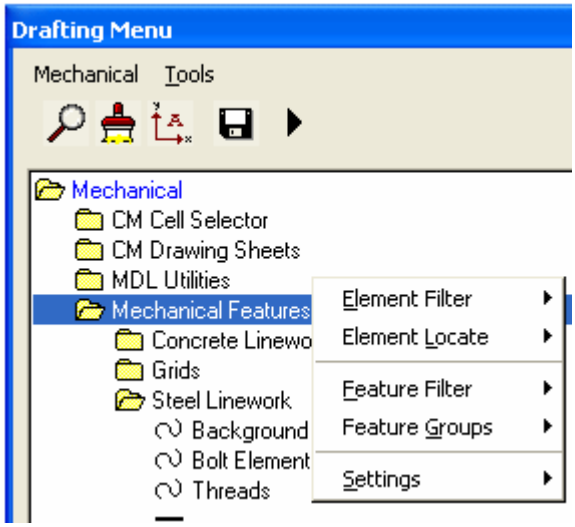
Settings can only be saved for Feature Tables opened one at a time. This command is not available if more than one Feature Table is opened.

View Feature Tables

Opens the Feature Table Editor in read-only mode.

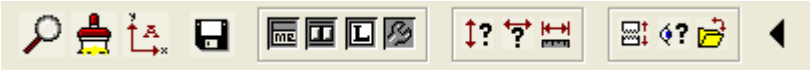
Pop-Up Menu

The pop-up menu duplicates most of the settings in the "Tools" pull-down menu that is part of the menu bar on the Draft dialog. The pop-up menu is only available if the list box mode is ON, and can be accessed by right-clicking on the list box.



The Draft dialog with the pop-up menu appearing over the list box

Tool-Bar



The tool-bar provides a set of tools that control the Drafting interface. Some of these tools only affect features in the list-box, and will therefore be disabled if the list-box is not displayed. These tools are described from left to right below:

Locate Tools

Locate Tools are similar to the Locate Tools in the Feature Table Editor. The only difference is the *"Add Features from CAD" tool mode, which will only add features temporarily to the Draft menu's list box. Temporary features can be useful when using the other tools, such as the Display Filter and the Conform by Feature* tool, since they allow unmatched features to be filtered, selected and replaced. Temporary features will display in the list box in magenta, and w*ill not be added to the feature table when saved.*

Filtering Tools

The Display Tools function identically to the Filtering Tools in the Feature Table Editor. They allow filtering of the view display or selection set by choosing features in the list-box. Multiple features can be selected in the list box by using <SHIFT>, <CTRL> or by drag selecting.

Drawing Scale

This icon opens the Drawing Scale dialog box. This is useful for changing the scale of text, cells and linestyle scales during the drafting process.

Save Settings

This command saves the current Feature Table settings to the local cache file. This allows individual users to customise the Feature Table for their own personal preferences as well as to their specific discipline. For example, an electrical engineer may be working on a P&ID and want the electrical symbols displayed in a separate window. The EE could tear off the "Electrical Symbols" sub-menu and then save the settings, so that every time he/she opens the Feature Table, the Electrical Symbols menu appears separately.

Settings that are stored in the Feature Table include:

Tear off menus
Hidden features
Open feature groups
Menus Across (ON or OFF)

The Save Settings command only allows one Feature Table to be opened at any time. If multiple Feature Tables are open, then the Save Settings command is unavailable.

Interface Toggles

The Interface Toggles control whether certain components of the Draft dialog are displayed or not. They display as clicked "in" (backfilled in grey) when they are ON, and clicked out when they are OFF.

Menu-Bar

Toggles whether the menu bar is displayed or not. This option is duplicated in the "Tools > Settings > Interface > Menu Bar" menu.

Tool-Bar

Toggles whether the tool-bar is displayed or not. This option is duplicated in the "Tools > Settings > Interface > Tool Bar" menu.

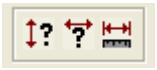
List-Box

Toggles whether the list-bar is displayed or not. This option is duplicated in the "Tools > Settings > Interface > List Box

 ***Tools Menu***

Toggles whether the "Tools" menu is displayed or not. This option is duplicated in the "Tools > Settings > Interface > Tools Menu" menu.

Menu Control Tools



The Menu Control tools control how the menu is displayed. The first two icons act as toggles, but the last icon is a one-time command icon.

 ***Long/Short Menus***

Toggles between Short Menu mode and Long Menu mode. In Short Menu mode, features that are flagged as hidden will not be displayed in either the pull-down menus or the list-box. In Long Menu mode, features flagged as hidden will still be displayed, but they will appear in magenta in the list-box. This option is duplicated in the "Tools > Settings > Interface > Menu Length" menu.

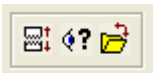
 ***Menus Across/Down***

Toggles between Menus Across and Menus Down mode. In Menus Across mode, children of the root feature table will be displayed in a separate menu across the width of the Draft dialog box. In Menus Down mode, they will be placed under the parent menu.

 ***Fit Width***

This options sets the Draft dialog width to the optimal size in order to display all interface components, including the menu-bar, tool-bar and list-box.

Feature Control Tools



The Feature Control Tools control the display of features in the list-box. They will be disabled if the list-box is not currently displayed, or if more than on list-box row is selected. The first two icons act as toggles, but the last icon is a one-time command icon.

Tear-off Menu

This toggle button denotes whether the selected row in the list box is flagged as a tear off menu. Torn-off menus appear in a separate Draft dialog, and display as a green row in the list-box. Only feature groups can be torn off, not individual features themselves. A feature group can also be designated as a tear off menu by the following options:

The "New Menu" option in the FTE
The "Tear Off" option in the menu bar

Hide Feature

This toggle controls whether a feature or feature group is hidden or not. Hidden features will not be displayed in either the menu-bar or the list-box, when the current menu length is set to *"Short Menus". This option is disabled in Short Menu mode, since you would never see a hidden feature to be able to turn it back on. Note that features can also be set as hidden by the administrator in the Feature Table Editor.*

Open/Close All Groups

This option toggles between opening or closing all feature groups.

List-Box

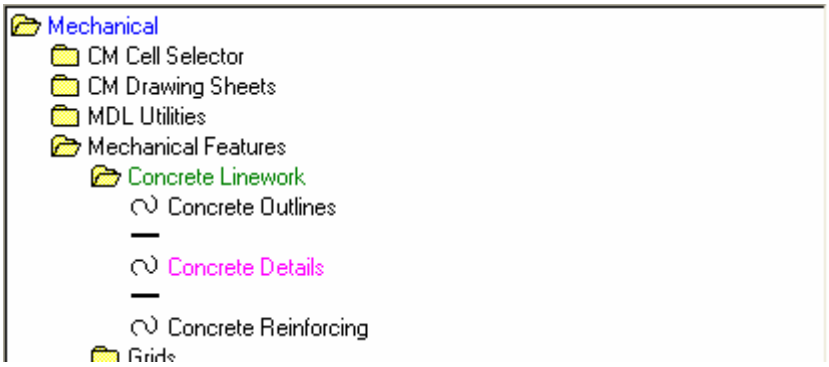
The list box displays all open Feature Tables concatenated together, similar in appearance to the Feature Table Editor. Groups are displayed hierarchically, and can be opened and closed by double-clicking on the group. Each feature is represented by a symbol, described below:

Cells	
Linear	
Shapes	
Text	
Dimension	
Command	
Separator	
Group	

The list box rows may also be coloured. Coloured rows have the following meaning:

Blue – The root level of a Feature Table
Red – A temporary feature
Magenta – A hidden feature
Green – A torn-off menu

Features can be activated by double-clicking in the list box row. If a selection set is active, then Draft will start the "Conform by Feature" command. Otherwise, it will start the "Draft by Feature" command.



A Feature Table named "Mechanical" with a torn off group "Concrete Linework" and a hidden feature "Concrete Details".

Right-clicking on the list box will open the Pop-up Menu. This is useful if the Menu Bar has been turned Off.

Chapter 12

The Watermarking Toolbox

The Watermarking Toolbox

The Watermarking Toolbox provides a number of Watermark related tools. The toolbox can be opened via the main CADconform menu by pressing "Watermarking Toolbox".

From left to right, the tools are:

Place Certify MarkerPlace Signature Marker Sign DrawingRead Watermark Cell

Place Certify Marker

The role of marker cells within the watermarking workflow is described in detail in the Report Generator (see chapter 10). This tool provides an easy way to place the watermarking marker cell.

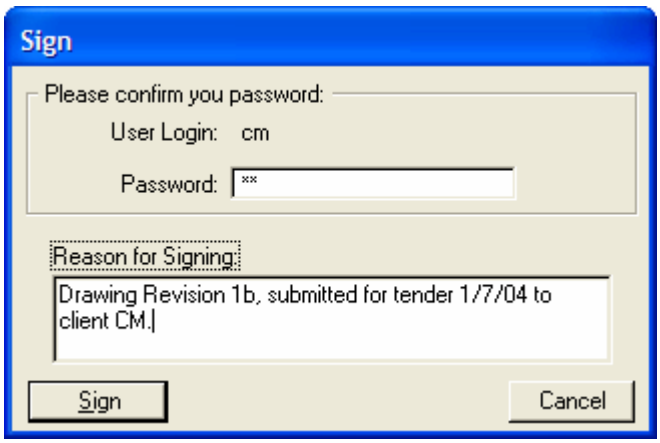
Place Signature Marker

The first thing the CADconform Administrator needs to do in order to digitally sign drawings is to create a "Signature Marker" and a "Signature" cell. This can be done using the usual MicroStation methods for cell creation. The origin and scale of the placed signature cell will be picked up from one of two existing cells in the design file: a marker cell, or a signature cell. This means that in order to sign a design file, the user has to first place a marker cell into the design file. This cell will usually be placed in the title block area of the design file. This tool will allow the user to place the signature marker cell in the drawing provided the cell has been defined and the user has the correct privileges.

Sign Drawing

Digital signatures are a useful tool used by CADconform to ensure the integrity of a drawing. Once a drawing has been signed then CADconform can determine if any changes have been made to that drawing. Digital signatures will check for any changes made to visible elements as well as the design settings (as described in Report Generator section).

When starting the tool, CADconform will check if the user has the correct privileges and then pop up the dialog below, that will prompt the user to enter their password and reason for signing.



Assuming the correct password for the logged in user is entered, CADconform will then replace the signature marker cell with the signature cell. To test if the drawing has been changed at any time after this point, simply verify the watermark (as discussed in the Conform section).

The concept of digital signatures and certification are very similar. Both use marker cells, watermarks and can check if a file has been changed. The major difference between them is that a drawing can be digitally signed even if it does not adhere to any standards, whereas a certification watermark can only be placed if the drawing is certified against a set of standards (dictionaries). This explains why users do not need to specify a dictionary when digitally signing a drawing.

Read Watermark Cell

This tool provides a quick method to launch the read watermark cell tool described in the conform section.

Chapter 13

Technical Reference

The *CADconform Database*

The CADconform database (herein referred to as "the database") is communicated to via the ODBC drivers under the Windows Operating System. The appropriate ODBC drivers will need to be loaded before using CADconform, depending on the preferred database application. Most ODBC drivers should be available either:

Preinstalled by Windows
From the database vendor
Downloadable from Microsoft's website: (http://www.microsoft.com/)

Availability and compatibility of various ODBC drivers is covered in the CADconform Installation Guide, which is part of the README file delivered with CADconform.

Note that the ODBC interface has had several name changes with various releases of Windows 95, 98, NT and 2000. This chapter refers to the ODBC interface generically, as the "ODBC Control Panel". On various systems, it may be called "ODBC", "32-bit ODBC", or "Data Sources (ODBC)". In Windows 2000/XP, the ODBC Control Panel may be in a sub-directory of the Control Panel called "Administrative Tools". In all cases, the ODBC data sources should be available from the Windows Control Panel.

Creating the CADconform Database

CADconform is delivered with a tutorial database ("CADconform TUTORIAL.MDB" - a Microsoft Access database) and an ODBC data source to connect to this data source ("CADconform TUTORIAL.DSN"). These files are located in the following directory, where $(CADconform) represents the install directory that the configuration variable $(CADconform_SERVER_DIR) points to:

$(CADconform)Database\CADconform Tutorial.mdb
$(CADconform)File DSN\CADconform Tutorial.dsn

This tutorial is intended to get the administrator familiar with using CADconform without having to configure everything beforehand. However, the tutorial database should not be used in a production environment. It is preferable to create a separate database for specific DGN standards, retaining the tutorial database simply for testing or training purposes.

The database should only be created by the administrator. This will usually be done only once at the beginning of the deployment of CADconform. Multiple databases can also be created if required for separate disciplines or sites.

The CADconform database can be created in one of two ways:

1) Through the ODBC control panel

2) Through a database application

Creating ODBC Data Sources

Once a new database has been created, an ODBC data source needs to be created which points to this database. CADconform supports all three types of ODBC Data Sources:

System DSNs
User DSNs
File DSNs

A brief description of each type is listed below. Note that because ODBC data sources are a standard Windows component, more information can be found at Microsoft's website.

System DSN

Stored in the Windows registry of the local computer, a system Data Source Name is visible for every user who logs on to this machine.

User DSN

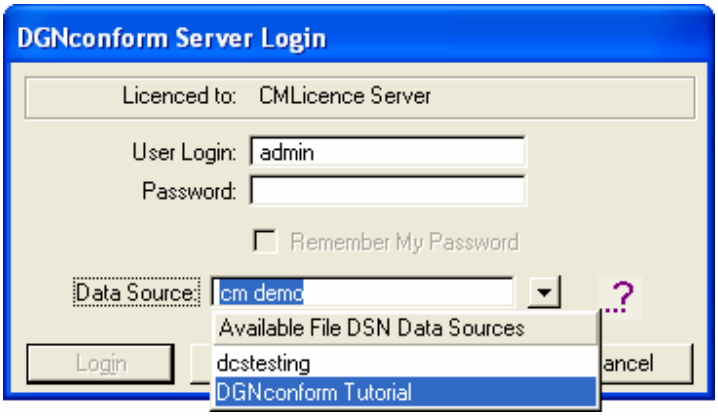
Also stored in the Windows registry, this type of DSN is only available to the person currently logged on to the machine. If another user logs on to the same machine, then they will not be able to use the DSN.

File DSN

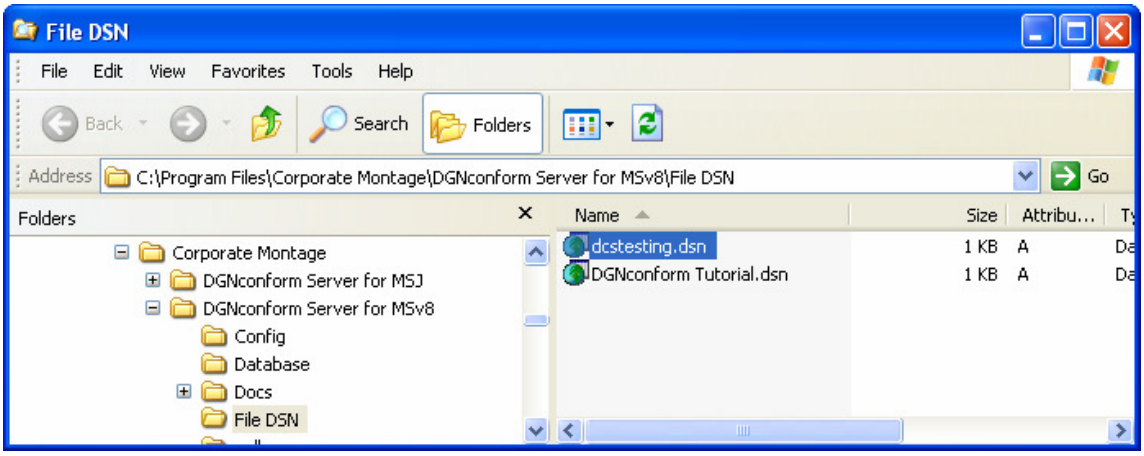
Unlike the User and System DSNs, the File DSN stores its information in a plain ASCII text file instead of the Windows Registry. The main advantage of this type of DSN is that this one text file can be copied to the server, making it visible to all CADconform clients.

Since the File DSN is the only type of DSN that doesn't require setting up on each client machine, it is the one most recommended to use. There is a special directory reserved for File DSNs in the root CADconform installation directory named "File DSNs". Any files in this directory will be automatically listed on the CADconform login dialog box.

Existing DSN files can either be copied to this directory, or created from scratch in this directory using the ODBC Data Source Administrator.



DSN files listed on the Login Dialog Box above, and the corresponding DSN files shown in Windows Explorer below.



Tables Created by CADconform

CADconform keeps extensive records of all changes made by users to Feature Tables, Reports and design files modified by the Feature Conformer - Conform. As well as these tracking records, CADconform also stores the Feature Tables and Reports in the database. A list of all of the tables created by CADconform follows:

CADconform

"CADconform_Version" – The versions and install date of CADconform used on each database.

The Report Generator

"Report_All_XXX" - a detailed error report called "XXX"
"Report_Inf_ XXX" – a tracking log of changes to the report "XXX"
"Report_CAD_ XXX" – a list of design file information and levels used in "XXX".
"Report_Fea_ XXX" – a Feature Summary Report for "XXX".
"Report_Sum_ XXX" – an Error Summary Report for "XXX".
"Report_Wat_ XXX" – a Watermark Report for "XXX".
"Log_BatchReport" – a log file of reports run using the Batch Reporting API.

The Feature Table Editor

"Table_XXX" – a Feature Table called "XXX".
"*iTable_XXX" – a tracking log of changes to the Feature Table "XXX".*

Conform

"Track_ParentDir_DesignFile" – a tracking log of changes to "ParentDir\DesignFile.CAD" using Conform.

The Tracking Report is done in the background when the user runs Conform, and is only available if the configuration variable: "_CADconform_REPORT_TRACKING" is set to 1.

The structure of the tracking log tables may be of interest to document management administrators wishing to create customised reports. The structure of the Feature Table may be of interest to administrators wishing to import data from another database.

Table Structure

A summary of the columns of each of these tables is listed below. A "Boolean" value is represented as TRUE = -1, FALSE = 0.

The Information Report Table

This is a compulsory report written out every time the Report Generator is used.

Column Name	Description
Report_Number	The Report Number
User_Name	The creator of the report's Full Name
Action	Action – either "created" or "rewritten"
Action_Date	Action Date – when the action occurred
Feature_Tables	Feature Tables used in creating the report
Scan_Type	Scan type – which features were reported on
Num_Passed	Total elements that matched an existing feature
Num_Errors	Total errors produced in this report

The Error Summary Report

This optional report contains one row for each unique error in a design file. Multiple instances of the same error are summarised.

Column Name	Description
Design_File	The design file in which an error was found
Symbology	The description of the error or the symbology of an unmatched feature
Num_Errors	The number of times this error occurred

The Detailed Error Report

This optional report contains one row for every error in a design file. The same error may appear multiple times.

Column Name	Description
Design_File	The design file in which an error was found
Symbology	Error description (unmatched symbology or other problem)
XCoord	X coordinate of unknown feature (in master units)
YCoord	Y coordinate of unknown feature (in master units)
FilePos	File position of unknown feature

The Feature Summary Report

This optional report contains one row for each unique feature (known or otherwise) in a design file.

Column Name	Description
Design_File	The design file in which an error was found
Feature	The full name and parent groups of the feature
Symbology	The symbology of this feature
Num_Features	The number of instances of that feature in the named file

The Design File Summary Report

This compulsory report contains information about every design file opened by the Report Generator.

Column Name	Description
Design_File	The design file included in the report
MSLevel	The levels reported on in this design file
Master_Units	The Master Units per Sub Unit in this file
Sub_Units	The Position Units per Sub Unit in this file
MU_Name	The name of the Master Units, e.g.: “ft”
SU_Name	The name of the Sub Units, e.g. “in”
Origin_X/Y/Z	The global origin of the model, or the attachment origin of a reference
Scale	The Drawing scale of the model, or the attachment scale of a reference
Num_Passed	Total elements that matched an existing feature
Num_Errors	Total errors produced in this report

The Watermark Status Report

This optional report contains one row summarising the watermark status of every design file.

Column Name	Description
Design_File	The design file reported on
User_Name	The full name of the user who created the watermark
Action_Date	The time and date that the Watermark was placed
Feature_Tables	The Feature Tables used when the watermark was created
Watermark_Status	The validity status of the watermark, or the reason it couldn’t be certified

The Feature Table (CADconform v4)

This is the Feature Table structure used by version 4.

Column Name	Description
Feature	The Feature Name
Description	The Feature Description5
Flags	Feature Flags (group, hidden, tear off, etc)
Type	The Element Type (e.g. cell = 2, line = 3, line string = 4, etc)
Id	ID (a unique number)
ParentId	Parent ID (to identify the parent group)
Match_Types	Match Types (element types 1-128)
Special_Flags	Special Flags (to represent FTE options)
ElmType_Info	Element Type Info (specific to some element types)
Special_Info	Extra Info (additional info, such as symbology ranges)
UserKeyin1-4	User Key-in (4 columns of 256 characters each)

The Feature Table (CADconform v3)

This is the legacy feature table structure for CADconform. This table format can be imported into the current version, and may be useful for importing data from other sources.

Column Name	Description
Feature	The Feature Name
Flags	Feature Flags (group, hidden, tear off, etc)
Type	The Element Type (e.g. cell = 2, line = 3, line string = 4, etc)
Id	ID (a unique number)
ParentId	Parent ID (to identify the parent group)
Match_Colour	Boolean – match on element colour
Match_Weight	Boolean – match on element weight
Match_Style	Boolean – match on element style
Match_Level	Boolean – match on element level
Colour	Element colour (0-255)
Weight	Element weight (0-31)

[5] Reserved for future expansion

Style	Element style name (string or number)
Level	Element level (1-63)
Cell_Match_Normal	Boolean – match on normal cell (type 2)
Cell_Match_Shared	Boolean – match on shared cell (type 35)
Cell_Match_Name	Boolean – match on cell name
Cell_Name	Cell name string
Linear_Match_Line	Boolean – match on line (type 3)
Linear_Match_LineString	Boolean – match on line string (type 4)
Linear_Match_Arc	Boolean – match on arc (type 16)
Linear_Match_Curve	Boolean – match on curve (type 11)
Linear_Match_Chain	Boolean – match on complec chain (type 12)
Linear_Match_Bspline	Boolean – match on open B-spline (type 27)
Shape_Match_Normal	Boolean – match on shape (type 6)
Shape_Match_Complex	Boolean – match on complex shape (type 14)
Shape_Match_Ellipse	Boolean – match on ellipse (type 15)
Shape_Match_Bspline	Boolean – match on closed B-spline (type 27)
Shape_Match_FillType	Boolean – match on shape fill type
Shape_Match_FillColour	Boolean – match on shape fill colour
Shape_FillType	Shape fill type (0 = not filled, 1 = filled)
Shape_FillColour	Shape fill colour (0-255)
Text_Match_Normal	Boolean – match on text (type 17)
Text_Match_Node	Boolean – match on text node (type 7)
Text_Match_Dimension	Boolean – match on dimension (type 33)
Text_Match_Tag	Boolean – match on text (type 37)
Text_Match_Width	Boolean – match on text width
Text_Match_Height	Boolean – match on text height
Text_Match_FontName	Boolean – match on font name
Text_FontWidth	Text Width (in units of resolution)
Text_FontHeight	Text Height (in units of resolution)
Text_FontName	Font Name (“0” to “255” or the font name)
Special_Flags	Special Flags (to represent FTE options)
UserKeyin	The user key-in associated with this feature

Feature Table Tracking Log

This table is written out whenever a Feature Table is created or exported.

Column Name	Description
Report_Number	The event number
User_Name	The user's full name
Action_Date	Action Date – when the action occurred
Action	Action – either "exported" or table description when created
Num_Features	The total number of features in the Feature Table

The Conform Tracking Log

This table is created for every design file operated on by Conform when Design Tracking is enabled by the administrator.

Column Name	Description
Design_File	Design file path and name
User_Name	The user's full name
Action_Date	Action Date – when the action occurred
Action	Action performed from Conform (Change / Change All / Ignore / Add / Delete)
Original_Symbology	Original Symbology of unknown feature
New_Symbology	New symbology of feature if changed
Feature	Feature Table, Group name and Feature name if changed

The CADconform Version Table

This table is written to every time a new version of CADconform is loaded.

Column Name	Description
Report_Number	The Report Number
User_Name	The logged in user's Full Name
Action_Date	Date when the new version was detected
Installed_Version	CADconform version detected
MS_Version	The MicroStation Version the table was created within.

Configuration Files

There are two configuration files used by CADconform:

Administrator's Configuration File
Client Configuration File

One is stored on the server and read by all users. This is called the "Administrators Configuration File". The other file is stored locally on each client machine and is created by CADconform automatically. This file is called the "Client Configuration File".

The Administrator*'s Configuration File*

The Administrators Configuration File is called "CADconform.CFG" and is usually located in the "Config" directory of the server. This file controls global settings for all users of CADconform who run this executable. If your site has different departments with different configuration needs, then you can create separate copies of the file for each department with a different name or path. CADconform will use the CFG file pointed to by the local configuration variable: "_CADconform_CONFIG_FILE".

Note: The Configuration file gets loaded every time CADconform is loaded. If you change variables and reload CADconform, then the changes should take effect. However, this is not the case for variables that are deleted or commented out. Because MicroStation does not have a new value to overwrite the old one, the old definition for the variable will still be loaded until MicroStation is exited. For this reason, it is better to reload MicroStation each time you change a variable in the Configuration file.

The Administrators Configuration File defines the following variables. These variables are loaded in as standard MicroStation configuration variables at the Application (Appl) level, but will not be visible to users because they begin with an underscore "_" character. They can still be overridden at the workspace "user" level if necessary, or locked using the standard MicroStation "%lock" keyword to disable users overriding the administrators value.

A value of "0" means a particular option is OFF, and a value of "1" means the option is "ON". Note that most configuration variables expect a zero or one value, but some require a path to a file, directory or URL (internet address). As a general rule, these are differentiated by the suffix, either "FILE", "DIR" or "URL", unless otherwise stated.

The available variables are:

_CADconform_REPORT_TRACKING

_CADconform_REPORT_PROMPTMAXLOG

_CADconform_REPORT_MAXLOGGABLE

_CADconform_CHECK_CELLS

_CADconform_CHECK_SHAREDCELLDEF

_CADconform_CHECK_TEXT

_CADconform_CHECK_TAGS

_CADconform_CHECK_SHAPESASLINEAR

_CADconform_CHECK_CELLCOMPONENTS

_CADconform_ALLOW_UNKNOWN_CONFORMING_CELLS

_CADconform_CHECK_COMPLEXCOMPONENTS

_CADconform_BORDERSHEETS

_CADconform_DISPLAY_LEVELS_OFF

_CADconform_WATERMARK_CROSS

_CADconform_MAX_ERRORSCALE

_CADconform_DICT_DIR

_CADconform_DOCS_DIR

_CADconform_FILEDSN_DIR

_CADconform_MENU_FILE

_CADconform_README_FILE

_CADconform_MARKERCELL

_CADconform_SCRIPTS

_CADconform_TECHSUPPORT_URL

_CADconform_USERS_FILE

_CADconform_WEBSITE_URL

_CADconform_USEDICTVERSIONING

_CADconform_REPLACETEXT_DATE_STRING

_CADconform_REPLACETEXT_DATE_FORMAT

_CADconform_REPLACETEXT_DICTVERSION_STRING

_CADconform_REPLACETEXT_LINESPACING

These variables are described below:

_CADconform_REPORT_TRACKING

If ON, then Conform will track all commands to the Conform tracking table. Default is OFF (0).

_CADconform_REPORT_PROMPTMAXLOG

If ON, then the Report Generator will prompt the user to continue when a report attempts to write a defined number of entries. Default is ON (1).

_CADconform_REPORT_MAXLOGGABLE

Defines the maximum number of allowed entries in a report before the user is warned (see also CADconform_REPORT_PROMPTMAXLOG). Default is 1000.

_CADconform_CHECK_CELLS

If ON, then Conform will check cell headers, otherwise they will be skipped. Cell headers are usually checked to ensure that only cells of a known cell name are allowed in the drawing, and that they are on the correct level. If a particular cell is placed as a point cell, then the usual symbology options (e.g. colour, weight, style, etc) can also be specified. Default is ON (1).

_CADconform_CHECK_SHAREDCELLDEF

If ON, then Conform and the Report Generator will check shared cell definitions (element type 34). This means that an error will be generated for shared cell definitions if they do not match any features in the feature tables. Note that shared cell definitions are usually invisible elements, but Conform will display them at position (0,0,0) in the design file if they are unmatched. Fixing a shared cell definition has the advantage of automatically fixing all of the shared cell instances (type 35) in the design file / model. Note that Conform and Report will not pick up problems with shared cell definitions if either a scan boundary or a fence are defined. This is because shared cell definitions have no real geographical location in a model, and hence specifying an area causes them to never fall within that search area. Default is OFF (0).

_CADconform_CHECK_TEXT

If ON, then Conform will check text and text nodes, otherwise they will be skipped. Default is ON (1).

_CADconform_CHECK_TAGS

If ON, then CADconform will check tag (type 37) elements, otherwise they will be ignored. Default is ON (1).

_CADconform_CHECK_SHAPESASLINEAR

If ON, then shape types (i.e. ellipses, shapes, etc) will be available in linear feature types, otherwise they will only be available for features of type "Shape". Default is ON (1).

_CADconform_CHECK_CELLCOMPONENTS

If ON, then CADconform will check all components of cells. If this variable is not set to "1", then the cell component elements will be ignored. Note that this setting is independent of the _CADconform_CHECK_CELLS setting, such that components of cells can be checked even if the header is ignored, and vice versa. Default is OFF (0).

_CADconform_ALLOW_UNKNOWN_CONFORMING_CELLS

If ON, then CADconform will automatically check the components of unmatched cells, regardless of the _CADconform_CHECK_CELLCOMPONENTS setting. If all of the components of an unmatched cell conform to the DGN standard, then the unknown cell will be allowed. If the components of an unmatched cell do not conform, then errors will be generated for each non-conforming component. This option has no effect on cells that do conform. This can be very handy to save the administrator having to define every possible cell in every cell library. Using this option, as long as the components of a cell conform, then it is not necessary for the cell to exist in any feature table. Default is OFF (0).

_CADconform_CHECK_COMPLEXCOMPONENTS

If ON, then CADconform will check all components of complex elements (other than cells). This includes complex chains, complex shapes and text nodes. This is useful in MicroStation v8 where individual text elements within a text node may have overriding symbology (such as font) that differs from the parent text node. Default is OFF (0).

_CADconform_BORDERSHEETS

This variable defines a semi-colon separated list of standard border sheets. CADconform will generate a warning if a design file is opened that does not have one of these border sheets attached and it will open the "Drawing Scale" dialog box (see Drafting Menu). If one of the defined border sheets is attached, then the attachment *scale of this reference file will be used as the drawing scale. If multiple border sheets are attached, then a warning will be issued and the Drawing Scale dialog will appear so that the user can set the scale manually. If the MicroStation "Save Full Path" option is used for reference file, then each border sheet must include the full path, otherwise it can simply be the reference file name.*

If no border sheets are defined, e.g.:

_CADconform_BORDERSHEETS =

Then no warning will appear, and CADconform will consider the automatic scale reading functionality to be disabled. Default is an empty string (no border sheets).

_CADconform_DISPLAY_LEVELS_OFF

This setting defines the levels that are automatically turned off when a file is certified. This list is comma separated, with a dash "-" representing a range, e.g. "41-48, 52, 54, 60-63". In MicroStation v8, this is a range of level codes. Default is ON (1).

_CADconform_WATERMARK_CROSS

This setting defines the symbology of the cross that is automatically drawn through an invalid Watermark cell. The symbology is represented by a key-in string in the same format used by a MicroStation key-in, e.g.:

lv=63;co=3;wt=4;lc=0

Where a symbology setting isn't specified (for example, no colour is defined) the cross will use the active symbology at the time the Watermark cross is drawn. Watermark crosses are drawn automatically by CADconform when generating a report, certifying design files, or rescanning the design file using the "Read Watermark" tool in conform. There is no default value.

_CADconform_MAX_ERRORSCALE

This value is a floating-point number representing the error tolerance of floating point values. Since MicroStation stores floating-point numbers as rounded integers, two numbers are rarely exactly equal. This value represents the percentage of error tolerated in the range 0% to 100%. Default is 0.0001.

_CADconform_DICT_DIR

This directory represents the default search path for imported and exported DICT files.

Default: $(CADconform_SERVER_DIR) Dictionary Files/

_CADconform_DOCS_DIR

This directory represents the default search path for documentation.

Default: $(CADconform_SERVER_DIR)Docs/

_CADconform_FILEDSN_DIR

This directory represents the default search path for File DSNs. It also supports wildcards, such as "" to match any number of characters, or "?" to match any one character.*

Default: $(CADconform_SERVER_DIR)File DSN/.**

_CADconform_README_FILE

This variable points to the README file opened when the user clicks on the "Release Notes" button in the "About CADconform" dialog box. It should not normally be changed.

Default: $(_CADconform_DOCS_DIR)readme.html

_CADconform_MARKERCELL

If ON, this option makes CADconform prompt the user to place a marker cell in the current model if one isn't already present. This prompt will occur during one of the following times:

The Drafting Tool is open
A new design file is open
The Report Generator opens a model

Default is ON (1).

_CADconform_SCRIPTS

This variable points to a directory containing scripts to be used with the "CADconform RUNSCRIPT" command. It is not currently used.

Default: $(CADconform_SERVER_DIR)scripts/

_CADconform_TECHSUPPORT_URL

This variable points to the website that will be opened in the system default web browser when the user clicks the "Website" button in the "About CADconform" dialog box. It should not normally be changed unless the Corporate Montage Technical Support page is moved. If the user wants to define their own web pages that can be added to the CADconform menu, they can use the "CADconform START" command, as demonstrated in the menu file. Default:

http://www.corporatemontage.com.au/contact/techsupport.htm

_CADconform_USERS_FILE

This variable points to the users password file. It is strongly recommended that this variable be locked (if it is not already) in the configuration file. Default:

$(CADconform_SERVER_DIR)Users/CADconform.pwl

_CADconform_WEBSITE_URL

This variable points to Corporate Montage on the web.

Default: http://www.corporatemontage.com.au

_CADconform_USEDICTVERSIONING

If ON, then CADconform will store a version number for feature tables and prompt users for the version number when saving a feature table. This version number will also be displayed in the feature table manager and can be used as replacement text within watermarks (see Watermarks Section). Default is 1.

_CADconform_REPLACETEXT_DATE_STRING

When defined this is the string within watermarks that will be replaced with the date. The string can either be a text element or part of a text node element (see Watermarks Section). The default string is $$DATE$$.

_CADconform_REPLACETEXT_DATE_FORMAT

This variable defines the format of the date or time string according to the table below. The times and dates are calculated by CADconform using the system time and date. By default this string is set to "%d %b %y" which for example for result in "01 Jul 04".

CODE	DESCRIPTION
%a	Abbreviated weekday name
%A	Full weekday name
%b	Abbreviated month name
%B	Full month name
%c	Date and time representation appropriate for the locale
%d	Day of month as a decimal number (01- 31)

%H	Hour in 24-hour format (00-23)
%I	Hour in 12-hour format (01-12)
%j	Day of year in decimal format (001-366)
%m	Month as a decimal number (01-12)
%M	Minute as a decimal number (00-59)
%p	Current locale's A.M./ P.M. indicator for a 12 hour clock
%S	Second as a decimal number (00-59)
%U	Week of year as a decimal number (00-51) with Sunday as the first day of the week.
%w	Weekday as a decimal number (0-6; Sunday is 0)
%W	Week of year as a decimal number (00-51) with Monday as the first day of the week.
%x	Date representation of current locale
%X	Time representation of current locale
%y	Year without century as a decimal number (00-99)
%Y	Year with the century as a decimal number
%z	Time zone name or abbreviation; No character if unknown.
%%	Percent sign.

Examples:

FORMAT	REPLACEMENT TEXT
%d %b %y	01 Jul 04
%A, %d %B, %Y	Tuesday, 27 August, 2002
$$:FTIME:Copyright %Y.	Copyright 2002.

_CADconform_REPLACETEXT_DICTVERSION_STRING

When defined this is the string within watermarks that will be replaced with the dictionary version(s) that the watermark was certified against. If there is more than one dictionary then the resultant replacement string will be a text node. This string can either be a text element or part of a text node element (see Watermarks Section). The default string is $$VERSION$$.

_CADconform_REPLACETEXT_LINESPACING

If the replacement string is over one line then the replacement element in a text node. This configuration variable defines the line spacing for this text node. An example of when this configuration variable would be used is if a DGN is certified against multiple dictionaries and the dictionary version string is replaced within a watermark. By default this variable is set to 1 and if it is undefined then the active line spacing of the DGN will be used.

Client Configuration File

This file is written out automatically by the CADconform Client Configuration Dialog. This dialog opens whenever CADconform cannot find a necessary file, such as the menu file, or the path to the server machine. This allows CADconform to be reconfigured on the fly if; for example, the administrator is working on a laptop that is removed from the network. See also: CADconform Client Configuration Dialog Box.

This file will not usually be edited manually by either the user or the administrator. An explanation of the variables stored in this configuration file follows:

CADconform_CLIENT_DIR

This variable contains the path to the local installation of CADconform Client. All files in this path are created automatically.

CADconform_SERVER_DIR

This variable contains the path to the standards database. This path is used for many things, such as checking the application version, reading the File directory and the administrators configuration file. Most other variables inherit this path.

_CADconform_CONFIG_FILE

This variable points to the Administrators Configuration File. It is preceded by an underscore to hide it from the Workspace Configuration dialog box.

CADconform*_CACHE_DIR*

This path points to the location in which to store cached dictionary files. By default, this is a local path.

CADconform_LOG_FILE

This variable points to the name of the log file to create during batch reporting. It is only used when CADconform reports are run outside of MicroStation.

CADconform*_MENU_FILE*

This variable points to the menu file to build the "CADconform" menu from. If users want to customise this menu, they can enter a path different to the default one which is shared by all users.

Installation FAQ

Q: I have migrated users from MicroStation v8 to 2004. Can I upgrade the CADconform database to 2004 standards automatically?

A: Yes. CADconform will automatically re-map level numbers to level names using the level tables active at the time that a feature table is open. Level numbers from a MicroStation v8 feature table (1-63) will be remapped to level codes in 2004, and the corresponding level names associated with these codes will be substituted. All key-ins supported by MicroStation v8 should be supported by 2004, assuming macros and 3rd party MDL applications are also available.

Q: I installed the CADconform database and client on one machine that works okay, but on another machine it can't connect to the database. What have I done wrong?

A: The most likely cause of this problem is that the server path was set incorrectly during the installation process. When CADconform is first installed, it writes the full path of the server (specified by the user) to the File DSN. This path must be a full UNC path to be visible to other machines on the network; it should not be a local drive or a mapped drive. Some examples of the server path are below:

Example 1:
C:\Program Files\Altiva\CADconform for MSv8

This is an example of a bad path. This would work on the machine it was installed on, but not any other networked machine; since CADconform Server was installed on the local drive of the server only.

Example 2:
M:\Altiva\CADconform for MSv8

This is another example of a bad path. This would work on any machine that had the "M:" drive mapped the same as the server machine, but not any other networked machine.

Example 3:
\\CADServer1\Shared Folders\CADconform for MSv8
This is an example of a good path. It should work for any machine that can see the shared folder "CADServer1\Shared Folders".

If the server path was entered incorrectly during the installation phase, you have two choices of how to fix it:

Uninstall the software and then reinstall it.

Using a text editor, correct the path to the database file in the File DSN directory on the server. By default, this file is located at:

"C:\Program Files\Corporate Montage\CADconform Server for MSv8\File DSN\CADconform Tutorial.dsn"

The required change to this file is the final line beginning with "DBQ=", followed by the database path. The contents of an example File DSN is displayed below:

```
[ODBC]
DRIVER=Microsoft Access Driver (*.mdb)
UID=admin
UserCommitSync=Yes
Threads=3
SafeTransactions=0
PageTimeout=5
MaxScanRows=8
MaxBufferSize=2048
FIL=MS Access
DriverId=25
DefaultDir=\\PIERS\CADconform Server for MSv8\Database
DBQ=\\PIERS\CADconform Server for MSv8\Database\CADconform Tutorial.mdb
```

Administration FAQ

Q: How much network bandwidth does CADconform use in day-to-day operation?

A: Very little. Configuration files (such as the administrators configuration file and the menu file) are read by CADconform at start-up from the network, but both of these files are around 2KB. The largest files in CADconform are the executable "CADconform.MA" and the database, which is usually around a couple of Megabytes, depending on the database application and the amount of data stored. Both of these are cached locally, and will only be downloaded from the network when they have been updated on the server. Note that the actual size on disk of the database is much greater than the individual data transmitted when a Feature Table is downloaded.

Q: Is it necessary to make backups of the database?

A: Yes. Since the database is read from (and possibly written to) by all users, it is essential that the data in the database remains intact. It is preferable that some form of automatic backup or archiving of the server machine is in place, but this is often not the case. At the very least, it is important to make occasional backups of the database file (or files, depending on the DB application) in the event that something becomes corrupt.

DB Applications often have repair tools, but these should not be relied upon to fix every problem with database corruption.

Q: What is the easiest way to make a backup of a Feature Table?

A: The easiest way to backup a Feature Table is to export it to a dictionary (".DICT") file. This allows the administrator to reimport the data to a new database; or over the top of an existing Feature Table if a mistake is made. See the Feature Table Editor* chapter for more information on exporting dictionary files.

Q: I have made a mistake in my Feature Table and exported it back to the database. Is there any way to undo this?

A: Currently there is no "Undo" operation for exported Feature Tables. However, there is always a copy of the Feature Table stored as a dictionary file available in the local cache directory on each client machine (pointed to by the Client Configuration variable "CADconform_CACHE_DIR"). This file can be reimported using the Feature Table Editor's "Import" command.

Q: How do I allow different menus to be used by different users?

A: All configuration is done through the two main configuration files – the Client Configuration and the *Administrators Configuration* files. To make changes specific to certain users, you need to change the Client configuration file on each machine. The Client configuration variable *"CADconform_MENU_FILE" sets the full path to the user's menu file.*

Trouble Shooting FAQ

This chapter lists a series of common problems and how to avoid them. It is broken down into the following sections:

Database Errors
Performance Issues
Conform Problems
Generating Reports
Draft Problems

It is also worth consulting the CADconform Installation Manual for problems with installing and running the software, or configuring the database. See the CADconform User Manual for "How To" style FAQs.

Database Errors

CADconform generates detailed error messages from both the MicroStation database server and the ODBC database server. Generally, if something goes wrong with the database, a helpful message will appear explaining how to resolve it. The questions below represent cases where a detailed error message does not appear.

Q: When I create a new database using the ODBC Control Panel, I cannot open it in my database application. Why?

A: Care must be taken when creating new databases using the ODBC Control Panel. If the ODBC drivers for the database application are newer than the database application itself, then the application may not recognize the format of the database file. For example, Microsoft Access 97 users will not be able to open a database (.MDB) file created by the Microsoft Access driver greater than version 4. To get around this, create the new database from within the database application itself.*

Q: When I try to log on to CADconform, I get a message "Unsuccessful CONNECT statement". What am I doing wrong?

A: This message will appear if:

The ODBC data source has been misspelled or has not been configured. Double check the data source name in the ODBC Control Panel.

The ODBC drivers are not supported by MicroStation. See the CADconform Installation Guide for driver compatibility.

The MicroStation Database Components have not been installed. This can be verified by clicking the MicroStation menu "Settings > Database > Connect". If the connect dialog appears, then the necessary components have been installed, otherwise a warning will appear.

Q: I keep getting an error "Couldn't update the information table" or something similar. Why?

A: This error can appear when the user selects a table in the Feature Table Manager that does not have a corresponding information table. This usually happens because the user has accidentally deleted the table, or imported the table from another package. This table can be created manually by creating a new Feature Table and copying the "iTable_TableName" file using your database application, and renaming it to match the Feature Table that causes the problem.

Performance Issues

Q: I have upgraded my Microsoft Access driver to version 4, and now my Feature Tables take longer to import. What has happened ?

A: According to information on Microsoft's Knowledge Base, some users have reported a 400% decrease in performance between version 3.5 and version 4.0 of Microsoft's Access driver. Check the Microsoft website (www.microsoft.com) for information on how to revert to the older driver if this is an issue.

Q: Importing and exporting Feature Tables to the database takes too long. Can I do anything to improve the speed?

A: There are a couple of ways to speed up creation of Feature Tables. Firstly, if the Feature Table export is slow because the network is very busy, you can copy the database file locally and work from there. This should only be done by the administrator, and only if no one else is likely to edit the Feature Tables. Once the editing is complete, the database file can be recopied to the network server.

Secondly, if the Feature Tables are quite large (over 3000 entries), you may consider breaking them up into smaller tables. This can be done by saving and inserting DICT files through the File menu. It is also possible that if the Feature Tables were created automatically, they contain many redundant entries. Try matching on as many possible element types for each feature and only specifying symbology match criteria (colour, weight, etc) where necessary. The "Delete Duplicates" tool in the Feature Table Editor can help reduce duplicate entries once these changes are made.

Large feature tables are often a result of importing cell libraries. If this is the case, you may wish to consider using the "_CADconform_ALLOW_UNKNOWN_CONFORMING_CELLS" option instead of defining each cell individually. With this option on, you will only need *to define cells that are allowed that contain components that do not conform. Cell components should usually conform, so this should require a minimum number of entries, if any.*

Conform Problems

Q: Why doesn't the "Flash" display option work on my machine?

A: Some graphics card drivers have problems with MicroStation OpenGL functionality. When this occurs, MicroStation cannot generate a proper timing interrupt; and the flashing may appear intermittent or not at all. Check the CADconform Installation Guide on ways to get around this problem.

Q: I have the Zoom toggle on, but I can't see the unknown feature in the view window. Why not?

A: There are three reasons why this can occur:

The view level for the feature is turned off.

The element has a corrupt element range. Try using "FIXRANGE" or EdG.

The element range is too small to zoom in to, e.g. the element is a line of zero length or a text element has a size of zero. In this case, the error status field in MicroStation should contain a message: "Element is too small to zoom in to."

Q: When Conform finds an unknown feature, I don't get any suggestions in the "Change To" list box. Why not?

A: The suggestions in the "Change To" list box will only display features of the same type as the unknown element, either: cell, linear, shape or text. If there are no suggestions in the list box, then there are no features of this type in the Feature Table.

Q: Why does Conform find an error with an element that appears to have correct symbology?

A: Conform will not match an element if the element type is not supported by the feature, regardless of the symbology. For example, if a feature called "Concrete Object Line" does not have the "Arc" toggle turned on in its Feature Type options, then an Arc element will be flagged as an error, even if the symbology of the arc exactly matches the symbology of a Concrete Object line. If this is indeed the reason why the element did not conform, then changing the element to a Concrete Object Line will not solve the problem, because Conform changes element symbology, not element type (it can not convert an Arc to a Line). If the user attempts to change an element to a feature type that doesn't support the element type, then a warning to this effect will be displayed in a pop-up dialog box.

Q: Why does Conform find an error with a cell that is not visible?

A: If the "_CADconform_CHECK_SHAREDCELLDEF" option is enabled, then Conform will check shared cell definitions (type 34). These elements are not normally displayed by MicroStation, but they do control the symbology of shared cell instances (type 35). During this process, Conform will temporarily display the invisible shared cell definition at origin (0,0,0) in the current highlight colour. Once the element is conformed or ignored, the cell will once again disappear. If you do not wish to check shared cell definitions, then this option should be turned OFF.

Report Generator Problems

Q: The Report Generation is very slow. Is there any way to speed it up?

A: The Report Generator can become very slow when there are many invalid features in the design file. You may wish to try doing only a minimum report with no Report Types toggled ON, to see how many errors there are. You can then fix some of these errors in Conform before trying the report again. Additionally, you may wish to do reports on only a few levels at a time, or only one feature type at a time. The largest report is always the "Individual Error Report", so turning off this report type should give the best speed increase.

Q: I ran the Report Generator but nothing happened. What's going on?

A: You may have forgotten to use the "Add" button when defining which design files to report on. Go back into the Report Generator and check if there are any files in the Design File List. If there are files in the list, then it is possible that you have opened the wrong database file. Check the "ODBC Data Source" on the Report Generator dialog box to see where the report went. Then go the ODBC Control Panel to see which database file the data source is pointing to. Additionally, it is possible that:

1) Not all the levels of each design file were reported on.

2) The report was restricted by either a fence or *border shape*.

3) Certain element types were disabled in the "Scan Type" options.

Q: I often get database error dialog boxes popping up when viewing reports. Why?

A: You will get "Table not found" type errors when you click on a report type if the corresponding report was not created. For example, to view a Feature Summary report, you must ensure the report was created with "Feature Summary" toggled ON.

Draft Problems

Q: When I click on a feature, it does not start the Drafting command. Why not?

A: There are two main reasons why this would occur:

1) An element selection set exists prior to starting the command

2) The Conform dialog box is open.

In the first case, this can easily be solved by clearing the selection set with a data-point before restarting the draft command. When a selection set exists, Draft automatically goes into Conform by Feature mode.

In the second case, you will need to close the Conform window before you can start drafting. When Conform is open, Draft automatically goes into Conform Change to Feature mode.

Q: My Draft tool-bar, menu-bar or tools menu has disappeared. How do I get it back?

A: Various components of the Draft interface can be disabled via the tool-bar and menu-bar, or by resizing or docking the Draft window. In order to get these components back, try any of these options:

Undocking and resizing the Draft window larger

Right-clicking on the list-box to get the pop-up menu

Using the "Tools > Settings > Interface" options

Using the tool-bar interface settings icons

FAQ Contact Information

If you have any questions regarding this FAQ or any suggestions to add to it, simply contact Corporate Montage at support@corporatemontage.com.au or visit the website for an office near you:

http://www.CADconform.com - CADconform*'s main webpage*

http://www.altivasoftware.com – Altiva Software's Main webpage.

http://www.corporatemontage.com.au - Corporate Montage Australia

http://www.corporatemontage.com - Corporate Montage North America

http://www.corporatemontage.de - Corporate Montage Germany