

Autodesk AutoCAD DWF Reader/Writer

FORMAT NOTES:

This chapter contains information specific to AutoCAD DWF files.

- For general AutoCAD DWG/DXF information and AutoCAD feature types supported by FME, please refer to the documentation on the *Autodesk AutoCAD DWG/DXF Reader/Writer*.

The AutoCAD® Reader and Writer modules allow FME to read and write files used by Autodesk® AutoCAD and compatible systems. AutoCAD drawing files consist of drawing settings and configuration, as well as a series of entities, or graphic elements, organized into layers.

FME provides broad support for many AutoCAD entity types and options and for reading and writing AutoCAD file version up to and including 2007. When AutoCAD data is output, header information may be copied from a supplied template, or prototype, file.

This chapter assumes familiarity with AutoCAD-compatible systems and the entities (features) that are manipulated within these systems.

Note: Throughout this chapter, the AutoCAD file is referred to as a drawing file rather than a DWF file.

Overview

There are three supported formats used by AutoCAD:

- DXF (drawing exchange format) files, which are large ASCII files,
- DWG (drawing) files, which are binary and support the most entity types, and
- DWF (drawing web format) files, which are binary files of reduced size and functionality intended for display on limited-bandwidth mediums such as the Internet.

Logically, both DWG and DXF files are identical and, therefore, FME treats both file types in the same manner. DWG/DXF files are read by the *Autodesk AutoCAD DWG/DXF Reader/Writer*.

DWF files are handled seamlessly but internally they undergo a different series of translation processes. These are read separately by the AutoCAD DWF reader and writer. Currently the DWF reader and writer can only read and write two-dimensional (2D) DWF files.

This document covers information specific to AutoCAD DWF files. For general AutoCAD DWG/DXF information and AutoCAD feature types supported by FME, please refer to the documentation on the AutoCAD DWG/DXF reader and writer.

AutoCAD DWF Quick Facts

Format Type Identifier	DWF
Reader/Writer	Both
Licensing Level	Base
Dependencies	None
Dataset Type	File
Feature Type	Layer name
Typical File Extensions	.dwf
Automated Translation Support	Yes
User-Defined Attributes	Yes
Coordinate System Support	No
Generic Color Support	Yes
Spatial Index	Never
Schema Required	Yes
Transaction Support	No
Geometry Type Attribute	autocad_entity

Geometry Support			
Geometry	Supported?	Geometry	Supported?
aggregate	no	point	yes
circles	yes	polygon	yes
circular arc	yes	raster	yes
donut polygon	yes	solid	yes
elliptical arc	yes	surface	yes
ellipses	yes	text	yes
line	yes	z values	yes
none	no		

Reader Overview

The AutoCAD DWF reader extracts entities, one at a time, from the entity section of the drawing file and passes them on to the rest of the FME for processing. Complex entities such as polylines and inserts are exploded and broken into several individual FME features. If the entity has attribution stored as extended entity data, then this is also read and placed in the feature.

When the AutoCAD reader encounters an entity type it does not know how to process, it simply sets the entity type of the feature and returns it. This feature is then logged by the FME correlation subsystem and the reader moves on to the next entity.

Reader Directives

This section describes the directives that are recognized by the AutoCAD DWF reader. Each directive is prefixed by the current <ReaderKeyword>_ when placed in a mapping file.

DATASET

Required/Optional: *Required*

The dataset into which feature data is to be read.

PASSWORD

Required/Optional: *Optional*

This statement specifies the password to open the DWF file for reading if it is password protected. The statement is of the following form:

```
<WriterKeyword>_PASSWORD <autocad dwf password>
```

The statement below instructs the AutoCAD reader try to open the given dataset with the password "mypass":

```
DWF_PASSWORD mypass
```

Value: <valid password>

Default value: *no password*

PAPER_WIDTH, PAPER_HEIGHT

Required/Optional: *Optional*

These statements specify the maxima of the width and height in millimeters for the sheets read from the input DWF file.. The statements are of the following form:

```
<WriterKeyword>_PAPER_WIDTH <width in mm>  
<WriterKeyword>_PAPER_HEIGHT <height in mm>
```

The statements below instruct the AutoCAD reader limit the extents of the sheets read from the input DWF file to 297mm by 210 mm:

```
DWF_PAPER_WIDTH 297  
DWF_PAPER_HEIGHT 210
```

Paper_Width Value: <Valid positive numeric>

Paper_Width Default Value: 297

Paper_Height Value: <Valid positive numeric>

Paper_Height Default Value: 210

Writer Overview

The AutoCAD writer provides the following capabilities when writing AutoCAD files.

- Password security: Passwords can be created for DWF files written by FME
- Multiple file formats: DWF files can be written as either compressed or uncompressed binary files, or as ASCII files
- Resolution: DWF file resolution can be determined at translation time in the form of the X Size and Y Size writer keywords. These are specified in pixels.
- Color Map Optimization: Colors in the color map that are unused can be eliminated from the output file to reduce space.
- Export Invisible Layers: Invisible Layers can be optionally exported if desired. (Not supported for Write 3D)
- Force View Extents: The initial viewport of the output file can be overwritten to include the entire extents of the drawing instead of the default active viewport.
- Use Inked Area: The inked area of the DWF file can be calculated from the entities in the file to produce a tight bounding rectangle around drawable graphic entities.
- Skip Layer Info: Additional layer information can be omitted from the output DWF file to reduce space.
- Skip Named Views: Named viewports can be omitted from the output DWF file in order to save space.
- Multi-version Support: The AutoCAD DWF writer supports files that are compatible with any current AutoCAD release.

When creating AutoCAD DWF files, the AutoCAD writer first defines the linetypes and layers defined within the FME mapping file. The writer then reads in a template file, if specified, and copies the linetypes, layer definitions, shape file header information, and block information from the template file to the output dataset.

The AutoCAD writer then outputs each feature it is given to the output file in the appropriate entity type.

When writing an AutoCAD DWF file, the format of file output is determined as follows:

- If the file name contains `.dwf` or `.DWF`, then the output dataset is written in DWF format.
- Otherwise, if an error exists in the mapping file, the translation is halted.

Writer Directives

This section describes the directives processed by the AutoCAD DWF writer module. Each of the directives is prefixed by the current `<WriterKeyword>_` when they are placed in a mapping file. By default, the `<WriterKeyword>` for the AutoCAD DWF writer is `DWF`.

DATASET

Required/Optional: *Required*

The dataset into which feature data is to be written.

DWF_VERSION

Required/Optional: *Optional*

The version of the AutoCAD DWF file to be produced.

The value corresponds with the release number of the AutoCAD DWF file that is produced. This statement specifies the version of 2D AutoCAD file to be output. The *WRITE3D* directive offers writing 3D AutoCAD DWF files of a newer version. The statement is of the following form:

```
<WriterKeyword>_DWF_VERSION <autocad dwf version>
```

The example statement below instructs the AutoCAD writer to produce a version 4.2 DWF file:

```
DWF_DWF_VERSION 4.2
```

Values: 4.2 | 5.5 | 6.0

Default value: 5.5

PASSWORD

Required/Optional: *Optional*

This statement specifies the password to open the DWF file for reading if it is password protected. The statement is of the following form:

```
<WriterKeyword>_PASSWORD <autocad dwf password>
```

The statement below instructs the AutoCAD writer try to open the given dataset with the password "mypass":

```
DWF_PASSWORD mypass
```

Value: <valid password>

Default value: no password

TEMPLATEFILE

Required/Optional: *Optional*

The name of an existing AutoCAD DWF file that contains the block definitions and line-type definitions to be used when creating the output dataset.

Value: <valid password>

Default value: no password

TEMPLATEFILE_PASSWORD

Required/Optional: *Optional*

DWF files support an optional password for additional security. If specified, the given password is provided when the file is opened.

Value: <valid password>

Default value: no password

FORMAT

Required/Optional: *Optional*

The format of the AutoCAD DWF file to be produced.

Values: *COMPRESSED_BINARY | UNCOMPRESSED_BINARY | ASCII*

Default value: *COMPRESSED_BINARY*

X_SIZE

Required/Optional: *Optional*

Specifies the horizontal width of the output DWF file in pixels.

Values: <valid positive numeric>

Default value: 36000

Y_SIZE

Required/Optional: *Optional*

Specifies the vertical height of the output DWF file in pixels.

Values: <valid positive numeric>

Default value: 24000

OPTIMIZE_COLOR_MAP

Required/Optional: *Optional*

This directive, if set, prevents unused colors in the color map from being stored in the DWF file.

Value: YES | NO

Default value: NO

Example:

```
DWF_OPTIMIZE_COLOR_MAP YES
```

EXPORT_INVISIBLE_LAYERS

Required/Optional: *Optional*

This statement exports invisible layers in an AutoCAD file to be output to the DWF file.

Value: YES | NO

Default value: NO

Example:

```
DWF_EXPORT_INVISIBLE_LAYERS YES
```

FORCE_VIEW_TO_EXTENTS

Required/Optional: *Optional*

This directive, if set, sets the initial viewport of the DWF file to the entire extents instead of the last actively seen viewport.

Value: YES | NO

Default value: NO

Example:

```
DWF_FORCE_VIEW_TO_EXTENTS YES
```

USE_INKED_AREA

Required/Optional: *Optional*

This directive, if set, calculates a tight bounding area around the graphic elements of a drawing.

Value: YES | NO

Default value: NO

Example:

```
DWF_USE_INKED_AREA YES
```

SKIP_LAYER_INFO

Required/Optional: *Optional*

This directive, if set, prevents additional layer information from being stored in the DWF file.

Value: YES | NO

Default value: NO

Example:

```
DWF_SKIP_LAYER_INFO YES
```

SKIP_NAMED_VIEWS**Required/Optional:** *Optional*

This directive, if set, prevents named views from being stored in the DWF file.

Value: *YES | NO***Default value:** *NO***Example:**

DWF_SKIP_NAMED_VIEWS YES

DEF**Required/Optional:** *Optional*

The AutoCAD DWF writer requires that every feature written to the AutoCAD file be stored within a predefined AutoCAD layer. In AutoCAD, the layers are used to store collections of logically related attributes. Within the FME, the AutoCAD layer and the type of the feature are treated synonymously as there is a one-to-one correspondence between FME feature type and AutoCAD layer.¹ The order of properties in the layer statement is required as shown, though additional attribute name and type pairs may be in any order. The layer statement is of the following form:

```
<WriterKeyword>_DEF <layer name> \
  autocad_color <default color> \
  autocad_linetype <default linetype> \
  [autocad_layer_type frozen] \
  [<attribute name> <attribute type>]
```

where:

- `<layer name>` is the name of the layer being defined. This is the name that is used throughout the remainder of the FME mapping files.
- `<default color>` is the color number used for all features stored within the layer unless explicitly overridden on the correlation lines below. Valid values are between 1 and 255.
- `<default linetype>` is the name of the linetype to use for the layer if no linetype is specified on the correlation line. The linetype specified must either be:
 - defined in the mapping file,
 - copied from a specified template file, or
 - the predefined linetype named `CONTINUOUS`.
- `<autocad_layer_type>` is the type of layer to create. Currently, only the value `frozen` is supported. If specified, then the created layer is frozen; otherwise, the layer is not frozen.
- `<attribute name> <attribute type>` is the definition of an attribute to be stored within the extended entity data of features for the layer. If no attributes are

1. Layers can also be defined through the use of a `TEMPLATEFILE`.

defined, then all feature attributes (except those that start with `autocad_`) are stored. The storing of attributes can be turned off by specifying a value of `external_attributes` for the `autocad_attributes` feature attribute on the correlation line. The values for `<attribute type>` are the same as those for ESRI Shapefiles.

The example below defines a layer called `boundary` in which entities are drawn using color 13 (unless otherwise specified) and a linetype called `dash-dot` (unless otherwise specified). The feature also has several attributes specified that will be written to the extended entity data of each feature within the layer.

```
DWF_DEF boundary \
  autocad_color 13 \
  autocad_linetype dash-dot \
  FEATCODE char(12) \
  PPID char(10) \
  DATECHNG date \
  SURVEYDIST number(8,2)
```

Feature Representation

Special FME feature attributes are used to hold AutoCAD entity attributes. The AutoCAD writer uses these attribute values as it fills in an entity structure during output. The AutoCAD reader sets these attributes in the FME feature it creates for each entity it reads.

For more information on general AutoCAD entities and their representations inside FME, please see the documentation on the AutoCAD DWG/DXF reader and writer.