



sol-id. adj. Of stable material throughout, reliable, permanent, secure, dependable

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The seacad.ini file

The seacad.ini file, found in the Solid Edge Program directory, is used to store the settings selected from the user interface. When you make a change to a parameter in the options form, a new value is saved to the seacad.ini file.

There are some parameters that are not exposed through the user interface. You can use a text editor, such as notepad, to set these parameters. However, if you edit this file, use extreme caution in setting these parameters. Errors made to this file can adversely affect the quality of the translation. The following list of parameters are not exposed through the interface.

Parameter	Description
Enable Logging=0	Turns on or off the creation of the log file. The default is off.
Read Default Units=64	Stores the import unit value. Possible values for units are: 59=meters, 61=millimeters, 62=centimeters, 63=kilometers, 64=inch, 65=feet, and 66=yards.
Template File =	Not Used.
Write Version=13	Determines what version of AutoCAD to export to.
Seed File =	Stores the path and file name of the AutoCAD ProtoType file used in export. Default value is none. The parameters that it extracts are as follows: <ul style="list-style-type: none"> • Header • Layers • Dimension styles • Line Types • Text Styles
Write Polyline Width Threshold=10.000000	Creates polylines in AutoCAD during the export process. The setting of 10 indicates that all elements with a line width equal to 10 or more will be created as polylines in AutoCAD.
Memory Needed Per Native Element = 100000	Undocumented
Memory Needed Per Foreign Element = 10000	Undocumented
Read Default Width=0.000000	Stores the "Default Width" parameter from the Import Options/Line Width dialog.
Processing Batch Translation = 0	Not used
MFC Application = 0	Not used
Process PaperSpace = 1	Sets flag for translating paper space. If this flag is set to 0, the file will be translated as model space even if it was saved in paper space. Setting this flag to 1 allows paper space to be translated.
Stroke Pattern = 1	Not Used
Use Pattern Name = 1	Not used



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Style Resource Name =	Not used
AutoCAD Extended Data = ACADASE	Undocumented
Delete Symbol Definitions = 1	Not used
Export All Graphics to PaperSpace = 0	Sets flag to export DXF/DWG to model space or paper space. The default is AutoCAD model space.
Process Non-Displayable Reference files = 1	Undocumented
Process Multiple Orientation in Viewports = 0	Controls if multiple viewports are translated or not. The default is on.
Dissolve Symbol to Groups = 1	Not used
Application Text Type = 0	Not used
Bigfont Name =	Not used
Part Layers = 1	Creates AutoCAD layer names that are derived from part file names that make up a draft file from a Solid Edge assembly. The graphics that represent each part are then placed on these layers.
Break Dimensions = 1	Controls how a dimension is created during the import process. With this parameter, dimensions can be created as graphics or as dimensions. The default is as graphics. Dimensions translated as dimensions are no longer supported.
Write Decimal Places = 10	Controls the accuracy of the .DXF file being created. 10 indicates the number of decimal places maintained.
Maximum Number Layer Name Chars = 16	Controls the length of the name used as a layer name in AutoCAD (AutoCAD has a limit of 16 chars for layer names). The Maximum Number of Layer Name Chars is used along with the Part layer name parameter.
Code Page = 0	Enables AutoCAD drawings to be opened both in the language that the drawing was created and in the language of a different country to which the drawing might be sent without any loss of information. The Code Page parameter is used during the export process. This was implemented for the support of the Kanji character set.