Running CADfix in Batch Mode

CADfix 12

Contents

1. Introduction

CADfix can be run in a pure batch mode from the command line environment on both Windows and LINUX systems to provide an automatic point-to-point translator for different data exchange applications.

In batch mode no CADfix graphical user interface is seen and any process output is directed to the command window and session log files.

A CADfix batch run can process either a single file or multiple files.

This document provides instructions for running CADfix in the automatic batch mode.

2. Pre-requisites for Running CADfix in Batch Mode

A valid licensed installation of the CADfix data exchange product is required to run CADfix in batch mode. The CADfix licence file should include the required product import and export options.

Note that if CADfix is being supplied as part of a vendor application integration for an OEM program then a dedicated application specific installation CD may have been supplied.

3. Running CADfix in Batch on Windows

To run CADfix in batch mode from the MS-DOS prompt the following command can be used:

```
"c:\Program Files (x86)\CADfix 12\bin\runcadfconsole.exe" -wait
                -BATCH -config wizard.cwc cube.sat
```

The above command will run the CADfix batch executable 'runcadfconsole.exe' and process the file 'cube.sat' using the CADfix Wizard settings specified in the CADfix Wizard Control (cwc) file 'wizard.cwc'.

In this case a single file 'cube.sat' is processed, but it is also possible to specify multiple CAD files on the command line, for example:

```
"c:\Program Files (x86)\CADfix 12\bin\runcadfconsole.exe" -wait
          -BATCH -config wizard.cwc cube.sat test.igs housing.x_t
```

will process the ACIS SAT file 'cube.sat', the IGES file 'test.igs', and the Parasolid file 'housing.x_t'. It is also possible to use the '-models' option (see section 5) to specify a text file containing a list of CAD files to process.

A full description of the command line options is provided in section 5 of this document.

The CADfix batch executable 'runcadfconsole.exe' is located in the 'bin' directory of the CADfix installation. The path to the CADfix batch executable should be changed according to the specific location of your CADfix installation. Quotation marks must be used as shown above if the path to the CADfix bin directory contains spaces.

The example shown assumes that the cwc file and CAD data file both exist in the directory from which the CADfix batch run is initiated. If this is not the case, then path names should be included in the command string as necessary.

When the CADfix batch command is executed you will see the progress messages displayed in the MS-DOS window from where the application was launched.

3.1. Windows BAT File Integration

For integration within a parent application the CADfix batch process could be initiated from within a Windows BAT file.

The following example is a simple Windows .BAT script called **'batchrun.bat'** with only two required arguments; **%1** is the cwc file name and **%2** is the list of files to process.

The contents of the Windows batch run file are as follows:

```
 rem set top level cadfix directory
 set FEGS_TOP=c:\Program Files (x86)\CADfix 12
  Running CAD translator in batch mode, Please wait...
 "%FEGS_TOP%\bin\runcadfconsole.exe" -wait -BATCH -config %1 –models %2
```

The script would then be run as shown:

```
 D:\users\data>batchrun.bat wizard.cwc files.list
```

The log files created (see section 7) and messages reported by the CADfix batch run can be captured and presented to the user as required by the parent application.

4. Running CADfix in Batch on LINUX

Before running CADfix in batch on a LINUX machine you need to make sure that your CADfix user environment is set up as it would be for a normal CADfix interactive run, i.e. the CADfix set-up script has been run using the command:

      source cadfix_setups   (see CADfix installation guide for more information)

On a LINUX system the command to run CADfix in batch is:

      cadfix -BATCH -config batch.cwc test.stp

This will run CADfix in batch using the Wizard control file 'batch.cwc' and will process the STEP file 'test.stp' in the launch directory. A full description of the command line options is given in section 5 of this document.

The user does not need to specify the path to the CADfix executable because the required environment variables have already been set if the 'cadfix-setups' file has been sourced correctly.

On the LINUX platform, instead of specifying an explicit filename in the command line, the user could use a wildcard character to process all files in the local directory, for example:

      cadfix -BATCH -config batch.cwc *.igs

will process all files with the extension '.igs' in the local directory.

It is also possible to specify multiple CAD files on the command line, for example:

      cadfix –BATCH -config batch.cwc cube.sat test.igs housing.x_t

will process the ACIS SAT file 'cube.sat', the IGES file 'test.igs', and the Parasolid file 'housing.x_t'. It is also possible to use the '**-models**' option (see section 5) to specify a text file containing a list of CAD files to process.

For integration within a parent application the CADfix batch process command can be run from within a LINUX shell script if required.

5. CADfix Batch Input Options

This section provides more information on the batch command line options. Please note that these options are case sensitive and should be used exactly as defined below

-wait	This should appear as the first command line option. The option is only valid on Windows and ensures that the command window DOS prompt will wait for the CADfix run to complete before becoming available for use. We recommend that this option is always used on Windows. This is not required on the LINUX platform because the command window will automatically wait for the process to complete (unless it is manually put into background).
-BATCH	Always required for batch processing and should be specified in upper case as shown. Instructs CADfix to run in a pure batch mode with no user interface displayed. If this option is not included, then the CADfix Wizard GUI will appear on the screen and progress messages will be displayed within it.
-config <cwc_file>	Identifies the CADfix Wizard control (.cwc) file that should be used to control the automatic model processing options and run parameters. The **<cwc_file>** argument is the name of the cwc file, for example **wizard.cwc**. If the file does not exist in the directory location from which the CADfix run was invoked, then the full path name to the cwc file should be specified. More information on the cwc file is provided in [section 6]().
-script <cwi_script_file>	Identifies the CADfix Wizard Interface (.cwi) script file that should be executed as a replacement for the default Wizard process. The script file is written in the cwi scripting language (based on Tcl). Extensive documentation for the cwi scripting language can be found in the separate cwi documentation.

-models <files.txt>	This option allows the user to specify the name of a file containing a list of CAD files for batch processing. In this example **files.txt** is used but any valid file name could be used, for example **models.dat**.
	
	 The contents of **files.txt** should be a list of CAD file names, for example:
	
	     test.igs
	     lid.igs
	     casting.sat
	     bracket.stp
	     housing.igs
	
	 The full path name to the CAD files should be included if they do not exist in the local directory from which the CADfix run was started.

-merge <filename>	This option prepares CADfix for the merging of batch files into a single database on import. The **<filename>** is the name of an existing CADfix database into which all the import files will be merged or the name of a new database.
-exportfile <filename>	Optional argument to specify the name of the file to be exported. If not defined then CADfix will define the export filename from the imported model name with a suffix of _cf (configurable via the .cwc file) added. This option can only be used when processing one model at a time. This option does **NOT** work for CADDS export which creates a folder not a file.
-workingdir <directory>	Optional argument to specify the name of the working directory to be used to export model to. Exported models will be written to this directory and have the same name as the input file but with a suffix of _cf (configurable via the. cwc file) added.
-damemory <number>	Optional argument to specify the memory used by CADfix during the batch processing. The value is given in Words, for example 16777216 for 64MB and 8388608 for 32MB. This is the same as setting the memory buffer using the CADfix Preferences window in interactive mode.

-noresourcedb	Place at the end of the command line to ensure that the .CADfixNNN user configuration file found in the HOME directory is not used

-xmlreport	XML summary report. The command line switch controls whether an additional batch summary is written in XML format. This switch can be used as follows:
	
	   a. '-xmlreport 1' (report will be written to the 'launch' folder, name "cadfixResults.xml")
	
	   b. '-xmlreport <path> (report will be written to the named file [if this file already exists then it will be backed up]

-importsystem <name>	Where <name> is the name of the import system e.g. Step. The command line switch will be followed by a folder path.
-regressiontest 1	This indicates a regression test is being run

6. CADfix Wizard Control File (.cwc)

The cwc file controls the operation of the CADfix Wizard automatic processing during the batch run. Options exist for the CADfix Import, Repair, Transform, Prepare and Export processes.

The cwc file allows you to specify the operations that should be performed, set any fixed tolerances and processing options required, specify export options and export file names.

The individual stages of the CADfix translation process can be turned on or off. For example, the 'Prepare for Export' stage can be set ON:

```
 prepare                          [0, 1] : 1
```

or turned OFF by changing the **1** to a **0**:

```
 prepare                          [0, 1] : 0
```

As well as controlling whether or not an individual process is run, the cwc file also gives access to the Wizard processing options for each stage. The following extract from a cwc file shows the Wizard options that are available for the Repair stage:

```
 repair                                          [0, 1] : 1

   repair,maxTol                         [DEFAULT, value] : DEFAULT
   repair,buildOption [SOLIDS, FACES, SOLIDS+FACES, UNKNOWN] : UNKNOWN
   repair,boundUntrimmed          [AUTO, ALWAYS, NEVER] : AUTO
   repair,approximateSurfaces                   [0, 1] : 0
   repair,allowVoids                            [0, 1] : 0
```

A cwc file can be modified to customise the batch process for specific customer application requirements as necessary. For example, the batch process could be set to stop if a solid body is not constructed during the initial repair stage, or it may be set to continue to prepare and export regardless of the repair result.

It is not recommended that customers manually edit the cwc file.

A cwc file should be supplied by TranscenData to meet the specific model processing requirements as determined during discussions about your specific CADfix application and the desired integration. A series of cwc files may be supplied for use in different usage scenarios.

An example cwc file is included in appendix 1 of this document.

See Appendix 1: CADfix Batch CWC options.

7. Files Generated During a CADfix Batch Run

This section lists the files that are likely to be created during a CADfix batch run.

Batch Run Log - e.g. batch<pid>.log

Overall batch run log file giving a brief summary for each model processed during the run. This is the same text as normally appears in the command window during a batch run. A unique number <pid> is given for every log file generated based upon the machine process number of the batch run. The example in Appendix 3 shows the batch log file for the processing of a single CAD model.

Import Log - e.g. test_IGESimport.log

Log file for the original CAD import process into CADfix. Relevant usually if problems or failures have been encountered during the file import stage. See example in Appendix 4.

Individual Model Processing Log - e.g. test_wizard.log

Detailed log file of the Wizard processing for an individual model. This is useful if a model fails to repair successfully and the user wants to review the processing undertaken. See example in Appendix 5.

CADfix Model Database File - e.g. test.fbm

CADfix binary database file for the model being processed. Could be deleted upon completion of a run or stored for opening in CADfix in a future batch run.

CADfix Model Database Backup File - e.g. test.fbm#1

Intermediate CADfix database backup file. This can usually be deleted after the run has been completed.

CADfix Command Level Log - e.g. test.fbl

CADfix command level log file. Sometimes useful information is contained in this file and would be used by TranscenData for debugging purposes, but in general this file can be deleted after the batch run is complete.

CADfix Export/Translated File - e.g. test_cf.igs

Resulting file exported from CADfix. In this case an IGES file has been created. The file suffix "_cf" indicates that the file was written by CADfix. The suffix used can be controlled by a cwc file option.

8. Return codes generated by a CADfix Batch Run

This section lists the possible return codes that a batch run can return. A return code of 0 means a successful batch run. A non-zero return code has one of the following meanings:

1001	Configuration file not specified
1002	Could not open configuration file
1003	Error reading configuration file
1004	Export system not specified in configuration file
1005	Invalid import system specification
1006	Could not open file
1007	Model name(s) not specified or file does not exist
1008	Value is not an existing file nor refers to an existing directory
1009	Invalid directory specification
1010	Invalid command line argument. Executable for xxx is invalid:
1011	Incorrect value supplied for the -server option
1012	Invalid memory buffer specification
1013	Invalid merge model specification
1014	Model list is empty or resolved to an empty list
1015	Import system is not licensed
1016	Could not read script file
1017	Failed to start CADfix
1018	Failed to checkout license feature
1019	Failed to check-in license feature
1020	Failed to checkout any suitable license features for system
1021	Configuration file xxx contains an error:
1022	Model processing failed

Appendix 1: CADfix Batch CWC[1] options

Item	Batch CWC options	Description
1	cwc,version	batch file format version number
2	preferredUnits	the default units to be used if none are specified
3	maxWizardTol	Processing will not use a tolerance greater than this value
4	workSet	optional working set name
5	workSet,last	++Not used for batch processing++
6	outputDirectory	++discontinued++
7	failDirectory	++discontinued++
8	continueOnError	specify whether the batch process continues when an error occurs
9	saveBetweenOps	save the CADfix database between wizard operations
10	saveOnExport	Save the CADfix database after export
11	pauseBetweenOps	specify if the batch process should pause at each major wizard stage
12	diagnosticCheck	Define at what condition a diagnostic check will be run
13	finish,save	save the CADfix database when the batch process is finished
14	preStartCommand	TCL script to be executed before the Wizard Repair processing starts
15	finish,diagnostics	++Discontinued++
16	qualityStandard	specify the quality standard to be used for a final diagnostic run at the end of the batch process
17	longNames	Will remove the 4-character name limit when on
18	licenceTimeOut	CADfix will check out all the features required at the start of a batch process. If any feature is not accessible, then the user may enter a number of seconds delay to the start of the batch run.
19	batchFeatures	List of additional licence features to be checked out at start of batch run
20	massProps,sagTol	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.
21	massProps,turnTol	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
22	massProps,filename	Specify a filename for capturing the mass property calculations.
23	massProps,append	Specify whether subsequent mass property calculations are appended to the same file
24	massProps,format	A special file format for a specific system
25	import	switch the Wizard **Import** stage ON or OFF
26	import,system	++discontinued++
27	import,labels	if the CAD system has available label information should this be imported
28	import,colours	if the CAD system has available colour information should this be imported
29	import,layers	if the CAD system has available layer information should this be imported
30	import,userAttrs	if user attribute information is available such as material properties these can be input as assignments to geometry
31	import,blanked	Option to import or ignore blanked entities
32	import,groups	Groups parts with matching attributes on import
33	import,shellsAsBodies	Option to import shells as bodies for PPS
34	import,codePage	Code page number for non-unicode characters
35	import,assemblies,mode	choose the mode used to process an assembly: the complete assembly, each component in turn or selected components of the assembly
36	import,assemblies,subpartid	where the option is chosen select the identity of the

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[1] Include all options to be switched off otherwise the default settings will be used

		assembly subpart to be processed
37	`import,assemblies,explode`	explode the assembly on import
38	`import,assemblies,emptyComponents`	import empty components of the assembly or ignore them
39	`import,assemblies,freeLines`	to control the import of free lines in an assembly
40	`import,assemblies,freeSurfs`	to control the import of free surfaces in an assembly
41	`import,assemblies,freeFaces`	to control the import of free faces in an assembly
42	`import,assemblies,freePartsName`	Sets the naming of free parts
43	`import,autoGroup,colour,apply`	instruct CADfix to create sets of geometry entities based on colour attributes during import
44	`import,autoGroup,colour,type`	specify the geometry entities to be collected into sets
45	`import,autoGroup,colour`	Single option to incorporate 32 and 33
46	`import,autoGroup,layer,apply`	instruct CADfix to create sets of geometry entities based on layer attributes during import
47	`import,autoGroup,layer,type`	specify the geometry entities to be collected into sets
48	`import,autoGroup,layer,values`	specify the layer numbers to be used for collecting geometry into sets
49	`import,autoGroup,layer`	Single option to incorporate 35, 36 and 37
50	`import,autoGroup,label`	Specify a part type, a label and the set name for processing during the transformation stage
51	`import,temporaryFiles`	leave any temporary files used during the wizard process
52	`import,resolveDegenerateLines`	Process degenerate lines during import
53	`import,recentreModel`	A model that was originally part of an assembly may be positioned a relatively large distance from the origin when compared to the size of the model. In order to obtain the highest computational accuracy during processing in CADfix, the model can be optionally re-centred about the origin on import.
54	`import,convertUnits`	convert the units on import
55	`import,scaleModel`	scale the model on import
56	`Import,calcMassProps`	Calculate mass properties after import operation
57	`import,iges,useDefaults`	If you have changed the import and repair options and wish to reset to the defaults
58	`import,iges,definitionEntities`	all entities flagged as "definition" entities are to be imported
59	`import,iges,blankedEntities`	all entities that are flagged as being blanked (e.g. hidden construction data) are to be imported
60	`import,iges,freeLines`	any entities that are not connected to a face, such as centrelines or drawing border entities, are to be imported. The default value of this option is set automatically according to the content of the IGES file. Some wireframe models contain a lot of lines that represent centrelines or dashed lines. Normally the IGES import reads all lines, making it hard to distinguish "real" lines from these construction-type lines. Wizard repair is thus confused by extra unwanted construction lines and manual repair is difficult.
61	`import,iges,freeLinesFont`	Line **Font filter** attribute allows unwanted lines to be filtered out during the IGES read. The IGES line font filter gives a better starting point for repairing wireframe models. Existing CADfix tools can be used to build more faces successfully.
62	`import,iges,constructionGeom`	The data defining an IGES entity may contain a pointer to the geometry used in its construction. For instance, a surface of revolution will be defined with reference to the curves used to sweep the surface. Check **Import construction geometry** if this geometry should also be imported.
63	`import,iges,linesAsNURBS`	By default, all lines will be converted to NURBS on import. If arc lines with very large radius (when compared to the size of the model) are preserved, then this may affect the value of the default tolerance used to repair the model. Uncheck **Import ALL lines as NURBS** if straight lines and arc lines should not be converted to NURBS. Any arc line subtending an angle greater than 90 degrees will then be split into multiple

		arcs.
64	`import,iges,splitCreasedNURBS`	The IGES file may contain NURBS edges and surfaces that need to be split on import. Check **Split discontinuous or creased NURBS edges** to split any edges that are found to be discontinuous
65	`import,iges,turningAngle`	minimum turn angle for the splitting of discontinuous edges is normally set to 20 degrees
66	`import,iges,splitClosedNURBS`	Check **Split closed NURBS surfaces** to split any closed NURBS surfaces that exist in the model.
67	`import,iges,pcurves`	The IGES standard allows trimmed surface entities to be defined with reference to the model space curves and / or the parameter space curves. The entity types to be referenced when the model is imported by default is set to "**Model space curves**" (PREF) to trim all parametric surfaces with the model space curves i.e. use IGES file preference flag to trim each surface according to the preference flag contained in the IGES file. Use GAPS for choosing P-curves for missing model curves if each surface should be trimmed with the model space curves, unless these curves are found to be missing or incorrectly defined, in which case the parameter space curves should be used. Use ON to choose "P-curves" to trim all parametric surfaces with the parameter space curves.
68	`import,iges,calcGoodTol`	The tolerance defined in the IGES file header may often be inaccurate. If the IGES file has been written by an ACIS or Parasolid based system, then the tolerance will often indicate the tolerance of the modelling kernel and may not actually bear any relation to the size of the model. Use to calculate the best tolerance to be used in processing the model. The value will be set with reference to the dimensions of the model and the tolerance specified in the IGES file.
69	`import,iges,mergeDuplicates`	to automatically merge duplicate points and edges at the model processing tolerance.
70	`import,iges,boundUntrimmed`	to create new edges around the boundary of any surfaces that are not currently trimmed
71	`import,iges,completeBoundaries`	to complete any face boundaries that contain gaps
72	`import,iges,stitchFaces`	to stitch the face boundaries together
73	`import,iges,buildBodies`	to automatically make solids from the model topology
74	`import,iges,extraAttributes`	Will assign to each CADfix entity created other attributes from the IGES file such as DE number, Form flag and IGES Entity type.
75	`import,step,freeLines`	**Import free lines/points** if any entities that are not connected to a face, such as centre lines, are to be imported.
76	`import,step,freeSurfs`	**free surfaces** if any unused or free surfaces are to be imported
77	`import,step,freeFaces`	**Import free faces** if any unused faces are to be imported.
78	`import,step,splitDiscontinuous`	The STEP file may contain NURBS edges that need to be split on import. Use to split any G1 discontinuous edges.
79	`import,step,splitAngle`	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the **splitAngle**. This angle defaults to 20 degrees.
80	`import,step,importPFID`	Import the STEP PFID attribute
81	`import,step,importPMI`	Switch on the imposing of Product Manufacturing Information (PMI) from the STEP file. For viewing only
82	`import,vdafs,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
83	`import,vdafs,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported
84	`import,vdafs,freeFaces`	**Import free faces** if any unused faces are to be imported
85	`import,vdafs,splitDiscontinuous`	The VDAFS file may contain NURBS edges that need to be split on import. Check **Split discontinuous**

		NURBS edges to split any G1 discontinuous edges
86	`import,vdafs,splitAngle`	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the **Discontinuity split angle**. This angle defaults to 20 degrees
87	`import,stl,mergeVertices`	By default, all duplicate vertices will be merged as the model is imported. Switch OFF if duplicate vertices should not be merged.
88	`import,stl,facettedSurfaces`	For quick viewing of the STL model this converts the STL facets to a CADfix entity type that appears and handles like a CAD surface. This reduces the volume of data transferred and increases the speed of import. The facetted representation subsequently held in CADfix can be roughly converted to geometry using an **experimental** interactive tool. The "Facetted surface" tool under the Integrity class of diagnostic problems can be used to identify areas of the model that can be subsequently trimmed (using Topology "Untrimmed surfaces") and converted to NURBS.
89	`import,stl,traceShells`	To trace shells in the model
90	`import,stl,reorient`	Option to re-orient all imported STL facets to be consistent, where possible
91	`import,stl,units`	Read the STL units as: MM, CM, M, INCH, FOOT
92	`import,3ds,facettedSurfaces`	For quick viewing of the 3DS model this converts facets to a CADfix entity type that appears and handles like a CAD surface.
93	`import,3ds,units`	Read the 3DS units as: MM, CM, M, INCH, FOOT
94	`import,acis,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported
95	`import,acis,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
96	`import,acis,freeFaces`	**free faces** if any unused faces are to be imported
97	`import,acis,splitDiscontinuous`	The ACIS file may contain NURBS edges that need to be split on import. Use to split any G1 discontinuous edges.
98	`import,acis,splitAngle`	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the **splitAngle**. This angle defaults to 20 degrees.
99	`import,acis,labelAttributeName`	Name of CADfix XASG to be used to define the ACIS entity label value, default is LABEL
100	`import,acis,layerAttributeName`	Name of CADfix XASG to be used to define the ACIS entity layer value, default is LAYE|R
101	`import,acis,splitPeriodics`	Option to split periodic surfaces
102	`import,dxf,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
103	`import,dxf,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
104	`import,dxf,freeFaces`	**free faces** if any unused faces are to be imported
105	`import,dxf,importBlanked`	Import or ignore blanked entities
106	`import,dxf,importProxy`	Option to control import of DXF proxy entities
107	`import,dxf,facetsToFaces`	Option to import DXF facets as BRep faces
108	`import,dxf,splitDiscontinuous`	The DXF/DWG file may contain NURBS edges that need to be split on import. Use to split any G1 discontinuous edges.
109	`import,dxf,splitAngle`	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the **splitAngle**. This angle defaults to 20 degrees.
110	`import,dxf,splitPeriodics`	Option to split periodic surfaces
111	`import,dxf,blocksAsAssembly`	**Import** DXF blocks as assemblies
112	`import,parasolid,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported
113	`import,parasolid,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
114	`import,parasolid,freeFaces`	**Import free faces** if any unused faces are to be imported.
115	`import,parasolid,units`	The Parasolid unit of length is meters. If it is known that the original design intent was to define the model using

116	`import,parasolid,splitClosedNURBS`	to control the splitting of closed NURBS during the import of a Parasolid file
117	`import,parasolid,Seamless`	to control the splitting of seamless faces during the import of a Parasolid file
118	`import,parasolid,splitDiscontinuous`	The Parasolid file may contain NURBS edges that need to be split on import. Use to split any G1 discontinuous edges.
119	`import,parasolid,splitAngle`	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the **splitAngle**. This angle defaults to 20 degrees.
120	`import,parasolid,explodeAssembly`	Parasolid assemblies can be exploded on import
121	`import,cadds,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
122	`import,cadds,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
123	`import,cadds,importBlanked`	Import or ignore blanked entities
124	`import,cadds,pointTol`	The CADDS file may contain coincident points and to distinguish these from points to be merged a **Point coincidence tolerance** may be applied.
125	`import,cadds,nameByPart`	assembly components labelled by part name
126	`import,catia,importNoShows`	A model can be constructed in CATIA with model entities on a SHOW or NOSHOW layer. Typically, the NOSHOW layer will be used for model construction entities. The use of this option depends largely on the source of the native CATIA files, and the CAD practices adopted by the designer. It has been found in some cases that the NOSHOW data is required to enable a valid model to be built within CADfix. Uncheck this option if it is found that a large amount of unwanted construction data is imported which is impairing the repair process.
127	`import,catia,importNoPicks`	The decision to classify modelling entities as either PICK or NOPICK also depends on the practices adopted by the CAD designer. By default, entities classified as NOPICK entities are not imported into CADfix.
128	`import,catia,importOnlyRoot`	use this option if problems are encountered when importing models from a particular supplier.
129	`import,catia,importOnlyMasterWorkspace`	Entities in a CATIA model can reside in different workspaces, but normally only the master workspace is required. Uncheck this option if problems are being encountered when importing models from a particular supplier.
130	`import,catia,dittosAsAssembly`	Import CATIA DITTO assemblies. This requires a special version of the translator based on Spatial's InterOp for which there is a royalty payment
131	`import,catia,solidmAsFacets`	Process CATIA mock-up solids or SOLIDM parts as facets
132	`import,catia,importPipes`	Import CATIA V4 piping entities (*PIP)
133	`import,catia,layerFilter`	option controls whether to use the user-specified layer filter while translating CATIA V4 file
134	`import,catia,layerLayerFilter`	Import the CATIA V4 layer filter attribute
135	`import,catia,useAnalyticSurfs`	The CATIA model can contain both NURBS and analytic representations of the same surface. This option is used to control which of these surface forms are imported into CADfix. If analytic surface representations are imported in preference to NURBS then this will result in a smaller CADfix database, but this often affects the reliability of the import process.
136	`import,catia,3DCurveRegen`	option controls the regeneration of 3D curves
137	`import,catia,3DCurvePref`	option sets the preference to be used while generating trimmed curves for a surface
138	`import,catia,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
139	`import,catia,freeSurfs`	**Import free surfaces** if any unused or free surfaces

"a different unit of length, then this parameter can be used to scale the model on import as appropriate."

		are to be imported.
140	`import,catia,freeFaces`	**Import free faces** if any unused faces are to be imported.
141	`import,catia,splitDiscontinuous`	The CATIA file may contain NURBS edges that need to be split on import. Check Use to split any G1 discontinuous edges.
142	`import,catia,splitAngle`	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the **splitAngle**. This angle defaults to 20 degrees.
143	`import,catia,importer`	Refers to different CATIA translator options: GSSL, INTEROP_GENERIC and INTEROP_ACIS. Some of these require extra royalty payments.
144	`import,catia5,importHidden`	**Import hidden entities** if entities tagged as hidden are to be imported.
145	`import,catia5,importCGR`	Import of CATIA V5 .cgr facet data
146	`import,catia5,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
147	`import,catia5,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
148	`import,catia5,freeFaces`	**Import free faces** if any unused faces are to be imported.
149	`import,catia5,combinedFaces`	**Import combined faces** if unused shells are to be imported
150	`import,catia5,importer`	Refers to different CATIA translator options: INTEROP_GENERIC and INTEROP_ACIS. These require extra royalty payments.
151	`import,catia5,labelType`	When reading labels from a CATIA V5 file the type can be set to **ELEMENT** or **PUBLICATION**. Alternatively, **BOTH** types of labels can be read.
152	`import,catia5,units`	Specify the CATIA V5 units
153	`import,catia5,combinedFaces`	Import combined surfaces from CATIA V5
154	`import,ug,importHidden`	**Import hidden entities** if entities tagged as hidden are to be imported.
155	`import,ug,importNonVisibleLayers`	**Import non-visible layers** if layer entities tagged as non-visible are to be imported.
156	`import,ug,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
157	`import,ug,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
158	`import,ug,freeFaces`	**Import free faces** if any unused faces are to be imported.
159	`import,ug,units`	If it is known that the original design intent was to define the model using a different unit of length, then this parameter can be used to scale the model on import as appropriate.
160	`import,ug,importer`	Refers to different CATIA translator options: INTEROP_GENERIC and INTEROP_ACIS. These require extra royalty payments.
161	`import,ug,splitClosedNURBS`	to control whether closed NURBS from a NX file are split during import
162	`import,ug,splitSeamless`	To control whether seamless faces from a NX file are split during import
163	`import,ug,assemblyPathPref`	Assembly path preference
164	`import,inventor,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
165	`import,inventor,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
166	`import,inventor,freeFaces`	**Import free faces** if any unused faces are to be imported.
167	`import,inventor,assemblyPathPref`	Select assembly path preference
168	`import,inventor,importUnfolded`	Import unfolded bodies
169	`import,inventor,importer`	Fix/stitch polyshells
170	`import,sw,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
171	`import,sw,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
172	`import,sw,freeFaces`	**Import free faces** if any unused faces are to be

		imported.
173	`import,sw,importHidden`	**Import hidden instances** if any hidden components are to be imported
174	`import,sw,importSuppressed`	**Import suppressed instances** if any suppressed components are to be imported
175	`import,sw,splitClosedNURBS`	Splitting of closed NURBS during import of SolidWorks data
176	`import,sw,splitSeamless`	To control whether seamless faces from a SolidWorks file are split during import
177	`import,sw,assemblyPathPref`	Select assembly path preference
178	`import,sw,configName`	To control whether SolidWorks configuration names are to be imported
179	`import,sw,importer`	Refers to different Inventor translator options: GSSL, INTEROP_GENERIC and INTEROP_ACIS. Some of these require extra royalty payments.
180	`import,proe,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
181	`import,proe,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
182	`import,proe,freeFaces`	**Import free faces** if any unused faces are to be imported.
183	`import,proe,facets`	Import of Pro/E facet data
184	`import,proe,importBlanked`	Import of Pro/E blanked data
185	`import,proe,vertexTol`	this is used to overcome the problem of unclosed loops in the Pro/Engineer data and permits the merging of vertices to be attempted during import to a tolerance specified
186	`import,proe,importer`	Refers to different Inventor translator options: GSSL, INTEROP_GENERIC and INTEROP_ACIS. Some of these require extra royalty payments.
187	`import,ansys,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
188	`import,ansys,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
189	`import,ansys,splitDiscontinuous`	The ANSYS file may contain NURBS edges that need to be split on import. Use to split any G1 discontinuous edges.
190	`import,ansys,splitAngle`	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the **splitAngle**. This angle defaults to 20 degrees.
191	`import,agps,freeLines`	**Import free lines** if any entities that are not connected to a face, such as centrelines, are to be imported.
192	`import,agps,freeSurfs`	**Import free surfaces** if any unused or free surfaces are to be imported.
193	`import,agps,splitDiscontinuous`	The AGPS file may contain NURBS edges that need to be split on import. Use to split any G1 discontinuous edges.
194	`import,agps,splitAngle`	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the **splitAngle**. This angle defaults to 20 degrees.
195	`import,agps,keepTempEntities`	Import construction geometry
196	`import,agps,pcurveToXYZTol`	tolerance used to map pcurves to model space curves
197	`import,sc03,userAttrs`	if user attribute information is available these can be input as assignments to geometry
198	`import,sc03,strictPMPLUS`	Choose between the formats SC03 PM or PM+
199	`import,sc03,buildFacesIf2D`	Where the SC03 import only contains 2D data faces will be built
200	`import,sc03,flatten2D`	CADfix will flatten the SC03 data to a 2D plane at the zero axis point
201	`import,sc03,ignoreCurveTrimming`	Option to ignore the SC03 curve trimming values and use full extent of curve
202	`import,sc03,importCL`	Import hole centre lines
203	`import,sc03,vertexTol`	Tolerance used to determine when two SC03 vertices are coincident. Used when building faces from 2D data.
204	`import,jt,freeLines`	Import of JTOpen free lines
205	`import,jt,freeSurfs`	Import of JTOpen free surfaces
206	`import,jt,freeFaces`	Import of JTOpen free faces

207	`import,jt,splitClosedNURBS`	Splitting of closed NURBS during import of JTOpen data
208	`import,jt,splitSeamless`	Splitting of seamless faces during import of JTOpen data
209	`import,jt,onlyFacets`	Import of only JTOpen facet data
210	`import,jt,facetLOD`	LOD control for imported JTOpen facetted data
211	`import,jt,importPMI`	Switch on the imposing of Product Manufacturing Information (PMI) from the JTOpen file.
212	`import,jt,stitchPolyshells`	Stitches the facets on import when selected
213	`import,prc,freeLines`	Import of Hoops PRC free lines
214	`import,prc,freeSurfs`	Import of Hoops PRC free surfaces
215	`import,prc,freeFaces`	Import of Hoops PRC free faces
216	`import,prc,onlyFacets`	Import of only Hoops PRC facet data
217	`import,pdf,freeLines`	Import of 3D PDF free lines
218	`import,pdf,freeSurfs`	Import of 3D PDF free surfaces
219	`import,pdf,freeFaces`	Import of 3D PDF free faces
220	`import,pdf,onlyFacets`	Import of only 3D PDF facet data
221	`import,flite,kntred`	Option to enable the removal of knots from Flite NURBS
222	`import,flite,kntredtol`	Tolerance used to reduce the complexity of Flite NURBS surfaces by removing knots.
223	`Import,solar,kntred`	Option to enable the removal of knots from solar NURBS
224	`import,solar,kntredtol`	Tolerance used to reduce the complexity of solar NURBS surfaces by removing knots.
225	`repair`	Switch the Wizard **Repair** stage ON or OFF
226	`repair,maxTol`	The Wizard will attempt to resolve any model connectivity problems by merging duplicate points and edges at successively higher tolerances. The maximum tolerance at which entities will be merged can be set using this parameter. The default maximum merge tolerance is dependent on the model dimensions and the original source of the model. Prior to importing the model these factors are unknown and so the maximum merge tolerance is set to the word DEFAULT, implying that the actual value used will be the default value for the model.
227	`repair,buildOption`	The type of model required at the end of the fixing process can be specified. The value of this option is also used to judge the success or failure of the fixing operation. If the selected option is not achieved, then the Wizard processing will end and an interactive session will be started. Choose **SOLIDS** if one or more bodies are expected and any unused faces that remain signify a failure of the repair process. Choose **SOLIDS+FACES** if one or more bodies are expected together with any unused faces. Choose **FACES** if no bodies are expected to exist after the repair stage. Choose **UNKNOWN** if the type of model required is not known. In this case no unused entities will be deleted from the model.
228	`repair,boundUntrimmed`	If any surfaces remain untrimmed after the model has been imported, then new edges can be created around the boundaries of these surfaces. Choose **Auto** if the decision to bound untrimmed surfaces is to be taken automatically by the Wizard depending on the original source of the CAD data and the poorly repaired state of the model. For instance, CADfix will choose not to bound untrimmed surfaces in IGES files from Pro/Engineer or CATIA but will if the IGES data is from NX. Choose **Always** if untrimmed surfaces should always be bounded. Choose **Never** if untrimmed surfaces should never be bounded.
229	`repair,wireframe`	Make allowance for wireframe within the imported geometry and repair
230	`repair,approximateSurfaces`	In those cases, where the model has missing surface geometry it may be possible to repair the model by

		creating new surface geometry. Use to create new surface geometry wherever possible. If the surfaces created are not planar then the new surfaces will only be an approximation of the original design intent.
231	`repair,allowVoids`	By default, any voids that exist in a model will be automatically removed. If the removal of these voids is found to be impairing the repair process or changing the original design intent, then use this so that the faces defining the voids are retained in the model. **This feature is now redundant and has been replaced with the option below.**
232	`repair,voids`	The user has the option to **Delete**, Remove or **Keep** any voids found in the solid model during the repair process
233	`repair,deleteWireframe`	Delete redundant wireframe geometry
234	`repair,keepUnused`	During the repair process any remaining unused entities would normally be removed as part of the cleaning and tidying up process unless switched on here
235	`repair,preserveKernels`	Preserve kernels in cellular bodies with shared topology
236	`repair,deleteUnusedFaces`	Under some circumstances you may wish to **Delete unused faces** during the repair process that may only require bodies.
237	`repair,calcMassProps`	Calculate the mass properties after the Repair process is complete
238	`repair,customCommand`	TCL script to be executed instead of the standard Wizard Repair
239	`transform`	Switch the Wizard **Transform** stage ON or OFF
240	`transform,joinBodies,apply`	to automatically search and replace groups of bodies with a single body
241	`transform,joinBodies,tol`	Tolerance used for touching bodies
242	`transform,joinBodies,mode`	Create a manifold or non-manifold bodies
243	`transform,joinBodies,allowVoids`	Allow voids in the final joined body
244	`transform,joinFragmented,apply`	Join faces with fragmented surfaces
245	`transform,joinFragmented,tol`	Tolerance used for joining faces with fragmented surfaces
246	`transform,joinFaces,apply`	to automatically search and replace groups of faces with a single face.
247	`transform,joinFaces,type`	Choose to join all faces or just slivers to their neighbours
248	`transform,joinFaces,width`	Set the maximum width of the sliver
249	`transform,joinFaces,quadlar`	Set a value which is used to specify the aspect ratio of 4-sided faces respectively when assessing which faces are to be joined.
250	`transform,joinFaces,trilar`	Set a value which is used to specify the aspect ratio of 3-sided faces respectively when assessing which faces are to be joined.
251	`transform,joinFaces,splits`	Allow narrow face splits
252	`transform,joinFaces,quality`	The user can set a bias or **Quality** control for the join decision making process and weight the joining of faces towards; **Any**, Mesh or **Geom**etry
253	`transform,joinFaces,faceangle`	Set maximum face angle deviation
254	`transform,joinFaces,edgeangle`	Set maximum edge angle deviation
255	`transform,joinFaces,angle`	The tangency **angle** may be used to control how faces are collected into groups
256	`transform,joinFaces,filtermultiloop`	Exclude multi-loop faces from the face joining
257	`transform,joinFaces,filterlarge`	Exclude relatively large faces from the face joining
258	`transform,joinFaces,filtermultisided`	Exclude multi-sided faces from the face joining
259	`transform,joinFaces,rejectthreesided`	Reject from the face joining three-sided faces
260	`transform,joinFaces,rejectmultisided`	Reject from the face joining multi-sided faces or those greater than four-sided
261	`transform,joinFaces,matchlayers`	Make use of matching layer in forming groups
262	`transform,joinFaces,matchcolours`	Make use of matching colours when joining groups
263	`transform,joinFaces,resolution`	**The Resolution** option controls the density of the sampling points used to calculate the new embedding surface. The higher the resolution, the closer the new surface will match the original surfaces.

264	`transform,joinFaces,change`	the **Maximum change** in displacement between the original surfaces and the final surface may be used as the acceptance criteria.
265	`transform,joinEdges,apply`	to automatically search for chains of edges and replace them with a single NURB edge
266	`transform,joinEdges,angle`	The tangency **angle** between any two edges may be used to control the process of joining chains of edges. Default angle = 1.0
267	`transform,collapse,apply`	to automatically collapse or remove any parts that are less than the **Minimum feature size – see below**
268	`transform,collapse,type`	Choose **EDGES** to only remove those edges that are shorter than the minimum feature size. Choose **FACES** if both faces and edges should be removed from the model. In this case, any `sliver' faces whose vertices and edges intersect according to the T-junction principle will be collapsed.
269	`transform,collapseOption`	Collapse FACES or EDGES
270	`transform,collapse,length`	**The Minimum feature size** specifies the size of the smallest feature that will be retained in the model. The default value is dependent on the model dimensions and the original source of the model. Prior to importing the model these factors are unknown and so the minimum feature size is set to the word DEFAULT, implying that the actual value used will be the default value for the model.
271	`transform,pinch,apply`	Check for and pinch sharp edge or corner angles in the model
272	`transform,pinch,angle`	Set the angle for the sharp edge angle test. Default = 5 degrees
273	`transform,pinch,tol`	Set the bridging length at which to pinch.
274	`transform,pinch,filletsonly`	Set the operation only for fillets
275	`transform,pinch,fixaction`	Specify the type of pinch action to take i.e.: MERGE, IMPRINT, PINCH, PINCH2
276	`transform,narrow,apply`	Check for narrow regions
277	`transform,narrow,width`	Minimum width defining a narrow region can be given as an embedded tolerance or a user supplied value
278	`transform,narrow,type`	The type of narrow region can be **Necks**, **Tapers** or **Both**
279	`transform,narrow,fixaction`	**Split** or **Pinch** are the type of fix choices
280	`transform,holes,removeHoles`	to automatically remove any holes whose radius is smaller than the **Max. diameter** – see below
281	`transform,holes,plugHoles`	Create bodies from holes
282	`transform,holes,drillTips`	Look for conical drill tip features in the model and remove them
283	`transform,holes,centrelines`	Generate centre lines for holes
284	`transform,holes,type`	Choose **BLIND** to remove only those holes that pass part way through the model. Choose **THROUGH** to remove only those holes that pass through the model. Choose **COMPLEX** to remove counter-sunk or counter-bored holes that pass through the model. Choose **ALL** to remove all types of hole.
285	`transform,holes,blind`	**Blind holes** remove only those holes that pass part way through the model.
286	`transform,holes,through`	**Through holes** remove only those holes that pass through the model
287	`transform,holes,complex`	**Complex holes (three or more openings)** remove holes that have three or more openings.
288	`transform,holes,searchMode`	The Cross-section mode
289	`transform,holes,check2dSize`	**Check exit size** parameter is used to check the 2D bounding box (or diameter) of the edges defining the hole exit.
290	`transform,holes,check3dSize`	**Check feature size** defines the 3D bounding box of the whole hole.
291	`transform,holes,checkSize`	this is a **No size limit** option to look for all the selected hole types in the model regardless of size.
292	`transform,holes,size2d`	The actual check exit size value
293	`transform,holes,size3d`	The actual check feature size value

294	`transform,holes,size`	setting the **Maximum Diameter** specifies the diameter above which holes will not be found.
295	`transform,holes,tol`	The loops defining the start and end of the hole do not have to be defined by circular arc lines. If only circular holes are to be removed, then this tolerance is used to judge whether any NURBS lines meet this criterion.
296	`transform,holes,xsect`	By default, the Holes tool will search for simple holes that are circular in **NON-CIRC**. Uncheck **CIRC** by choosing **ANY** if this restriction should not be applied.
297	`transform,holes,delete`	when creating bodies from holes delete the original body
298	`transform,protrusions,remove`	to automatically remove any protrusions whose radius is smaller than the **Max. diameter** – see below
299	`transform,protrusions,check2dSize`	Check exit size parameter is used to check the 2D bounding box (or diameter) of the edges defining the protrusion.
300	`transform,protrusions,check3dSize`	Check feature size defines the 3D bounding box of the whole protrusion.
301	`transform,protrusions,size2d`	The actual check exit size value
302	`transform,protrusions,size3d`	The actual check feature size value
303	`transform,protrusions,tol`	The loops defining the start and end of the protrusion do not have to be defined by circular arc lines. If only circular protrusions are to be removed, then this tolerance is used to judge whether any NURBS lines meet this criterion.
304	`transform,protrusions,checkSize`	this is a **No size limit** option to look for all the selected protrusions types in the model regardless of size.
305	`transform,protrusions,size`	setting the **Maximum Diameter** specifies the diameter above which protrusions will not be found.
306	`transform,protrusions,xsect`	By default, the Protrusions tool will search for simple protrusions that are circular in **NON-CIRC**. Uncheck **CIRC** by choosing **ANY** if this restriction should not be applied.
307	`transform,removeSeams,apply`	A seamless face is a face that is embedded in a cylindrical or conical surface, for instance, and is defined by just one or two circular edge loops without a seam running along its length. Use to make a seamless face.
308	`transform,removeSeams,tol`	tolerance used for evaluating the seamless face
309	`transform,removeSeams,pln`	applies to planes
310	`transform,removeSeams,cyl`	applies to cylinders
311	`transform,removeSeams,con`	applies to cones
312	`transform,removeSeams,sph`	applies to spheres
313	`transform,removeSeams,tor`	applies to toroids
314	`transform,deleteFillets,apply`	to automatically remove fillets
315	`transform,deleteFillets,radmax`	A maximum radius may be set to define the radius
316	`transform,deleteFillets,radmin`	If a minimum radius is used only groups of fillets falling between the two figures will be chosen. For variable fillets the program tests for radii along the fillet and only fillets that fall between the two values will be selected.
317	`transform,deleteFillets,radtol`	This value is used to control how accurately the fillet face chosen is defined. In other words, whether or not the radius around the curve of the face is constant or to within what value can it be defined as constant.
318	`transform,deleteFillets,tol`	This value is used to control accuracy of the operation.
319	`transform,deleteFillets,type`	The check button for **CONST** defines whether the fillet radius remains constant from one end of the face to the other. If **VARY** is chosen, then the radius may vary from one end of the face to the other.
320	`transform,deleteFillets,angtol`	Is used to determine when adjacent surfaces are considered parallel (default = 3 degrees).
321	`transform,deleteFillets,joinedges`	After removing fillets join edges to remove old vertices
322	`transform,deleteFillets,facemod`	*Redundant as scars are always left*
323	`transform,deleteChamfers,apply`	To automatically remove chamfers from the model
324	`transform,deleteChamfers,d1`	The upper distance from the dihedral angle corner to the start of the chamfer. Use this setting to avoid large slopping faces being picked up. The default upper

		corner angle (**A1**) of the chamfer will be 90 degrees.
325	`transform,deleteChamfers,ang1`	The upper corner angle of the chamfer.
326	`transform,deleteChamfers,d2`	The distance from the dihedral angle corner to down the side of the chamfer.
327	`transform,deleteChamfers,ang2`	The smallest dihedral angle between chamfer faces.
328	`transform,deleteChamfers,tol`	This value is used to control accuracy of the operation.
329	`transform,deleteChamfers,angtol`	Is used to determine when adjacent surfaces are considered parallel (default = 3 degrees).
330	`transform,deleteChamfers,joinedges`	Chains of edges left after the process will also be removed.
331	`transform,deleteChamfers,facemod`	++Not used for batch processing++
332	`transform,deleteFeatures,apply`	to automatically remove features
333	`transform,deleteFeatures,tol`	tolerance used for evaluating the features
334	`transform,deleteFeatures,joinedges`	Scan for multiple continuous edges between vertices and join into a single edge
335	`transform,deleteFeatures,facemod`	*Redundant as scars are always left*
336	`transform,deleteFeatures,set`	to specify a set name
337	`transform,imprintEdges,apply`	Set the program to imprint edges that coincide or overlap
338	`transform,imprintEdges,tol`	Set tolerance for edge imprinting
339	`transform,imprintBodies,apply`	Set the program to imprint faces of bodies. The bodies must touch along co-incident/overlapping faces. Intersecting bodies will not be imprinted. Imprinting bodies separated by large gaps can introduce accuracy problems
340	`transform,imprintBodies,tol`	Tolerance used for body imprinting
341	`transform,deleteBodies,apply`	Automatically delete bodies from the model using either a volume or a bounding box dimension
342	`transform,deleteBodies,method`	Specify whether volume or bounding box is to be used
343	`transform,deleteBodies,size`	Specify a value for the volume or bounding box diagonal dimension
344	`transform,externalFilter,apply`	Selection of only external model data activated
345	`transform,externalFilter,tol`	Set the tolerance for the removal of internal detail
346	`transform,externalFilter,maxgap`	Maximum gap to be ignored during the internal detail removal
347	`transform,externalFilter,mode`	Option to switch on aggressive filtering
348	`transform,externalFilter,perf`	Option controlling performance cost of aggressive filtering,1 is fast/low accuracy, 10 is fast/accuracy
349	`transform,externalFilter,level`	Set the level of data removal to faces, bodies or instances
350	`transform,wrap,apply`	Automatically create a shrink wrap of the model
351	`transform,wrap,facetSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.
352	`transform,wrap,facetTurn`	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
353	`transform,wrap,facetSize`	Specify the typical facet size to be used for shrink wrapping
354	`transform,decimate,apply`	Decimate the shrink wrap model by making planar areas coarse
355	`transform,decimate,tol`	Specify the tolerance to which the decimate function will work
356	`transform,unfacet,apply`	Option to enable/disable the unfacet action
357	`transform,unfacet,edge_vertextol`	Max expected difference between a vertex of a facetted edge and the original edge the facets were made from
358	`transform,unfacet,edge_sagtol`	Max expected difference between a centre of a facetted edge and the original curve the facets were made from
359	`transform,unfacet,face_vertextol`	Max expected difference between a vertex of a facetted face and the original face the facets were made from
360	`transform,unfacet,face_sagtol`	Max expected difference between centroid of a facetted face and the original face the facets were made from
361	`transform,unfacet,face_normtol`	Max expected difference between facetted face normal and the original face the facets were made from
362	`transform,unfacet,no_unclosed`	Option to reject any faceted bodies that are not closed shells after unfaceting

363	`transform,deform,apply`	Switch on deforming of CAD geometry to match deformed mesh
364	`transform,deform,fitMethod`	Select a fit method from either position only or position plus derivatives
365	`transform,deform,edgeTol`	Target tolerance when fitting new NURBS to deformed edges. Default is smaller than interior fit tolerance so as to bias the fit to produce minimal edge sloppiness. A successful fit will produce a NURBS that is within this tolerance of the deformed edges.
366	`transform,deform,interiorTol`	Target tolerance for fitting the interior of a deformed face. A successful fit will produce a NURBS that is within this tolerance of the deformed mesh nodal positions.
367	`transform,deform,normalTol`	When fitting with derivatives this option controls the quality of the fit to the deformed normal's. A successful fit will produce a NURBS with surface normal's within this angular tolerance of the normal's from the deformed mesh.
368	`transform,deform,keepBad`	Enable this option to keep fits that do not meet the target tolerances. A warning will be displayed if any fits triggering this condition. If disabled, then out-of-fit results will be discarded and the deformed surface will not be embedded in a NURBS surface.
369	`transform,deform,delete`	Enable this option to automatically delete the original surfaces following an acceptable refit.
370	`Transform,calcMassProps`	Calculate the mass properties after the transform operations finish
371	`transform,customCommand`	TCL script to be executed instead of the standard Wizard Transform
372	`prepare`	Switch the Wizard **Prepare** stage ON or OFF
373	`prepare,system`	specifies the prepare target system if different from the export system
374	`prepare,debug`	Use to print values without performing prepare for export
375	`prepare,customCommand`	TCL script to be executed instead of the standard Wizard Prepare
376	`'Overlapping curves,R-CU-EM:{{apply value}{tol value}}`	Check for whether a curve lies wholly within or partially overlaps another curve
377	`'Short edges',R-ED-TI:{{apply value}{tol value}}`	to automatically remove any short edges that occur in a chain of edges. Any edge that forms part of a chain and whose length is shorter than the user-specified value will be removed, either by collapsing the end-points to their average position, or by collapsing one end-point to the other, as appropriate.
378	`'Tiny faces',R-FA-TI:{{apply value}{areamin value}}`	faces smaller than 1.0e-03 times the model diameter or whatever value is provided by the user are removed
379	`'Narrow faces',R-FA-NA:{{apply value} {widthmin value:mm} {onlysimple value} {twosided value}}`	faces that are smaller than the value given along their whole length are removed
380	`'Surface simplification',R-SU-SIM :{{apply value}{tol value}{type value}}`	Check **Surfaces** to find and replace any NURBS entities that can be represented by a simple (primitive) entity within the specified tolerance. For instance, an analytic plane will replace any NURBS surface that can be considered to be planar within the specified tolerance.
381	`'Edge simplification',R-CU-SIM:{{apply value}{tol value}}`	Check **Edges** to find and replace any NURBS entities that can be represented by a simple (primitive) entity within the specified tolerance. For instance, any NURBS edge that is effectively straight within the specified tolerance will be replaced by a straight edge.
382	`'Non-NURBS surfaces',R-FA-AN:{{apply 1} {tor 1} {sph 1} {con 0} {cyl 0} {pln 0}}`	Check surfaces to convert any non-NURBS surfaces to NURBS entities
383	`'Non-NURBS edges',R-ED-AN:{{apply value}}`	Check edges to convert any non-NURBS edges to NURBS entities
384	`'Rational NURBS surfaces',R-SU-RAT :{{apply value}{tol value}}`	Some target systems do not support rational NURBS surfaces. Use to convert any rational NURBS surface to

		an equivalent degree 3 non-rational NURBS within the specified tolerance.
385	`'Rational NURBS curves',R-CU-RAT:{{apply value}{tol value}}`	Some target systems do not support rational NURBS edges. Use to convert any rational NURBS edges to an equivalent degree 3 non-rational NURBS within the specified tolerance.
386	`'Low-degree surfaces',R-SU-LDG:{{apply value}{degree value}{ignoredegr 1}}`	Increase the NURBS degree of surfaces to a user defined value. By default, surfaces of degree 1 are ignored
387	`'Low-degree curves',R-CU-LDG:{{apply value}{degree value}{ignoredegr 1}}`	Increase the NURBS degree of curves to a user defined value. By default, curves of degree 1 are ignored
388	`'High-degree surfaces',R-SU-HD:{{apply value}{tol value}{degree value}}`	Some target systems, such as I-DEAS and Pro/ENGINEER, do not support high degree NURBS. Use to reduce the maximum NURBS order to the value specified.
389	`'High-degree curves',R-CU-HD:{{apply value}{tol value}{degree value}}`	Some target systems, such as I-DEAS and Pro/ENGINEER, do not support high degree NURBS. Use to reduce the maximum NURBS order to the value specified.
390	`'Dense NURBS surface definition',R-SU-KNT:{{apply value}{tol value}}`	Some CAD systems create overly complex NURBS definitions with a high number of knots. Use to find any NURBS entities that can be redefined using a lower number of knots (and control points) such that the replacement NURBS is within the specified tolerance of the original NURBS.
391	`'Dense NURBS curve definition',R-CU-KNT:{{apply value}{tol value}}`	Some CAD systems create overly complex NURBS definitions with a high number of knots. Use to find any NURBS entities that can be redefined using a lower number of knots (and control points) such that the replacement NURBS is within the specified tolerance of the original NURBS.
392	`'Non-tangent patches',R-SU-NT:{{apply value}{tol value}{angle value}{factor value}}`	Use to fix any NURBS surfaces that are G1 discontinuous (non-tangent). The prepare process will attempt to fix the discontinuity by removing a row of control points or by splitting the surface along the line of discontinuity. A row of control points will only be removed if the subsequent movement of the surface is less than the specified tolerance.
393	`'Non-tangent segments',R-CU-NT:{{apply value}{tol value}{angle value}}`	Use to fix any NURBS edges that are G1 discontinuous (non-tangent). The prepare process will attempt to fix the discontinuity by removing knot points or by splitting the edge at the point of discontinuity. A knot point will only be removed if the subsequent movement of the edge is less than the specified tolerance.
394	`'Non-smooth patches',R-SU-NS:{{apply value}{ratio value}}`	Use to fix any NURBS edges that are G2 discontinuous (non-smooth). The prepare process will attempt to remove the discontinuity by moving knot points so that the curvature change between adjacent knots is less than the specified **Curvature ratio**
395	`'Non-smooth segments',R-CU-NS:{{apply value}{ratio value}}`	Use to fix any NURBS surfaces that are G2 discontinuous (non-smooth). The prepare process will attempt to remove the discontinuity by moving knot points so that the curvature change between adjacent knots is less than the specified **Curvature ratio**.
396	`'Surface parameterisation',R-SU-PCN:{{apply value}{condition value}{rate value}{tol value}}`	This option looks for knots with negative values and shifts all the knots so that the first one is zero. Also knots ranges from [0,1] are found and then CADfix shifts the knots start to zero and then scales them to end at 1. In a parametric surface such as a NURBS a unit step (dP) in the U or V direction space corresponds to a given distance in model space (dX). The ratio dP/dX is the derivative or rate of the parameter space. Some geometric processes can become upset if this derivative is very different from unity. **Parametric conditioning** will attempt to address this issue for surfaces scaling them to be approximately the same

		size as the model space it occupies. This scaling does not affect the shape of the parametric surface. The **Speed rate** for surfaces ensures the change in parametric speed over a given range does not exceed a threshold. CADfix will try and refit the surface to provide an improved but not necessarily fully fixed result.
397	`'Curve parameterisation',R-CU-PCN:{{apply value}{condition value}{rate value}{tol value}}`	The differences in parametric speed along an edge are smoothed by the various parameters given
398	`'Curve knot hyper-multiplicities',R-CU-HMK:{{apply value}{condition value}{rate value}{tol value}}`	NURBS curve knot hyper-multiplicity adjustment
399	`'Surface knot hyper-multiplicities',R-SU-HMK:{{apply value}{condition value}{rate value}{tol value}}`	NURBS surface knot hyper-multiplicity adjustment
400	`'Folded curves',R-CU-FO:{{apply value}{tol value}}`	to find and fix any NURBS edges where the angle between the normal at each successive edge segment changes by more than 90 degrees.
401	`'Oversize surfaces',R-FA-TRM:{{apply value}{percent value}}`	to trim the size of any NURBS surfaces to the size of the dependant face. Any NURBS surface whose area exceeds the area of the dependant face by more than the specified **Oversize** factor will be trimmed.
402	`'Closed surfaces',R-SU-CL:{{apply value}{tol value}}`	Some target systems, such as Pro/ENGINEER and ANSYS, do not adequately support closed NURBS surfaces. Use to fix any closed NURBS that are found to be closed within the specified tolerance.
403	`'Degenerate surfaces',R-SU-DC:{{apply value} {tol value} {simplify value} {split value} {flare value} {refit value} {trimback value:mm}}`	to fix any NURBS surfaces that are found to be degenerate. A degenerate NURBS surface is typically one in which the iso-parameter lines, in either the U or V directions, converge to a single point. The NURBS surface will be; simplified, flared, split, trimmed and refitted depending on the users choice.
404	`'Degenerate surface corners',R-SU-DP :{{apply 1} {angmin 5.0} {angtype FLAT}}`	A corner of a NURBS surface forms a tangential (~180) or very sharp (~0) angle.
405	`'Large edge-surface gap',R-FA-EG:{{apply value}{tol value}{lock value}}`	The gap between the edge of a face and the underlying surface known also as sloppiness is adjusted to be within the value given
406	`'Smooth surfaces',R-SU-SMO:{{apply value}{maxChange value}}`	A new smooth (C1/C2) cubic NURBS is fitted to replace the original surface within the tolerance. The fitted NURBS has the same shape as the original.
407	`'Undersize surfaces',R-FA-XTN:{{apply value}{undersize value}}`	to ensure that any surface whose area is smaller than the dependant face is extended to match the size of that face.
408	`'Seamless faces',R-FA-SEA:{{apply value}}`	Some target systems, such as Pro/ENGINEER, do not support seamless faces. A seamless face is a face that is embedded in a cylindrical or conical surface, for example, that does not have a seam edge running along its length. Use to split the face and create a new seam edge.
409	`'Missing surfaces',R-FA-EMD:{{apply value}{tol value}}`	New surfaces are created for any missing ones by using the edges that form the closed loop defining the face. The new internal "blend" surfaces is converted to s NURBS.
410	`'Sharp edge angle',R-LO-SA:{{apply value}{angle value}}`	removes sliver faces where the join between two adjacent edges in a loop forms a very sharp angle
411	`'Sharp corner angle',R-LO-SA  : {{apply 0} {angle 5.0} {tol DEFAULT} {filletsonly 0} {fixaction MERGE}}`	The joint between two adjacent edges in a loop forms a sharp angle.
412	`'Zero-area faces,R-FA-ZAR:{{apply value}{tol value}}`	Check for zero area faces
413	`'Refit faces,R-FA-RPL:{{apply value}{tol value}}`	Apply the face refitting algorithm to create new embedded surfaces
414	`'Degenerate tangent,R-CU-G1D:{{apply value}{tol value}}`	Check for degenerate tangents

415	`'Join short edges,R-ED-JOI:{{apply value}{tol value}}`	Join short edges in model to neighbours. Look for edges less than the minimum length supplied and join to an appropriate neighbouring edge if the default tangent angle criteria of 1 degree is not exceeded.
416	`'Sloppy surface singularity,R-FA-VFS:{{apply value}{tol value}}`	Heal surfaces with singularities (i.e. to translate the shape until a vertex of the face coincides with the singularity) using a solid-body transformation. For each singular surface if there is a dependent face with vertices within a given threshold of the singularity, the shape will be translated and rotated to heal the singularities to the vertices
417	`'Shared topology',R-MO-POP: {{apply 0}}`	Where two or more bodies are connected by shared faces or edges or vertex they are separated
418	`export`	Switch the Wizard **Export** stage ON or OFF
419	`export,system`	specify the export CAD system to be used
420	`export,fileSuffix`	The Wizard automatically sets the export file name. The file name is generated by appending the characters a suffix given here to the original model name. Otherwise _cf will be used.
421	`export,fileExtension`	The file name extension will be set here
422	`export,always`	CADfix will always continue to the end, even if errors are encountered during Repair or Prepare stage
423	`export,restoreModel`	If the model was re-centred on import, then it is generally necessary to restore the model to its original position when the output file is generated. Uncheck this option if the model should not be restored to its original position.
424	`export,assemblies,explode`	Explode any assemblies on export
425	`export,sendLabels`	This parameter controls the data written to the export file as a label. Choose **NAME** to write the name assigned by CADfix to each entity. Choose **XASG** to write the actual label assigned to an entity, where this exists. This is the default option. Choose **TYPART** to write the label data as a combination of the CADfix entity type and database instance number. Choose **COMBO** to write the label data as a combination of all of the above options. Choose **OFF** to prevent any label data being written to the export file.
426	`export,sendColours`	By default, any colour attributes assigned to the model entities will also be written to the export file. Set OFF if the export of these attributes is not required
427	`export,sendLayers`	By default, any layer attributes assigned to the model entities will also be written to the export file. Set OFF if the export of these attributes is not required
428	`export,sendUserAttrs`	if user attribute information is available such as material properties these can be exported as assignments on the geometry
429	`export,customCommand`	TCL script to be executed instead of the standard Wizard Export
430	`export,sendGeomForm`	**NURBS surface conversion** gives the ability to get NURBS surfaces converted back to surfaces-of-revolution (SoR) during the export stage. Apart from **OFF**, the option can be set to convert NURBS back to SoR's if that's what the NURBS was originally imported from, or to **auto-detect** all NURBS to see if they match a SoR, regardless of their **original form**.
431	`export,exportSet`	Allows the user to specify the export set as different from the working set
432	`export,textNoteEnabled`	CATIA specific option
433	`export,textNoteChars`	Text Note characters
434	`export,textNotePosX`	Text Note X position
435	`export,textNotePosY`	Text Note Y position
436	`export,textNotePosZ`	Text Note Z position
437	`export,textNoteSize`	Text Note size
438	`export,textNoteColour`	Text Note colour
439	`export,textNoteLayer`	Text Note layer
440	`export,textNotePlane`	Text Note plane (e.g. yz)

441	export,textNoteAngle	Text Note angle
442	export,checkEmpty	will not write the output file if you have an empty model
443	export,calcMassProps	Calculate the mass properties at the end of the batch process
444	export,iges,minres	This parameter indicates the smallest distance, in model space units, that is considered as discernible by the downstream system. Any coordinate locations in the output file that are separated by a distance smaller than the minimum resolution will be considered to be coincident. When the IGES file is flavoured for I-DEAS the value of this parameter is pre-set. Otherwise, the default minimum resolution is set according to the size of the model, unless the IGES file is being flavoured for CATIA, in which case the value defaults to 0.02.
445	export,iges,mode	In an IGES file a solid model can be defined in one of three ways. Choose **TRIM** surfaces to define the faces of each solid using trimmed surface entities. Choose **BOUND** surfaces to define the faces of each solid as bounded surfaces. Choose **BREP** solids if the downstream system supports this IGES entity type.
446	export,iges,pcurves	A trimmed surface can be defined with reference to the model space curves and / or the parameter space curves. Choose **MDUMMY** if trimmed surface entities are to be defined with reference to the model space curves only. Choose **PDUMMY** if trimmed surface entities are to be defined with reference to the parameter space curves only. Choose **ONLY** if trimmed surface entities are to be defined with reference to the model space curves and the parameter space curves. Choosing this option will increase the size of the file generated.
447	export,iges,pcurvefit	This parameter controls the degree of the P-curves written to the IGES file. Choose **LINEAR** to write degree '1' P-curves. Choose **CUBIC** to write degree '3' P-curves. Choose **BEST** to write degree '5' P-curves wherever possible.
448	export,iges,form	A form flag will be assigned to every NURBS surfaces defined in the IGES file. The form flag is used by some systems to determine the use of these entities. For instance, if a cylindrical NURBS surface is flagged as a surface of revolution, I-DEAS will interpret that surface to display the centreline. Choose **OFF** to flag the NURBS surface form as "unspecified". Choose **ANA** to flag any surface that could be replaced by an analytic surface as "analytic". Choose **SOR** to flag any cylindrical or conical surfaces as "surfaces of revolution".
449	export,iges,shapeTrim	Use if NURBS surface entities are to be trimmed to fit the size of the dependant face.
450	export,iges,defColour	Specify the default entity colour from a range given
451	export,iges,dirLabel	Use the export label in the 8-character directory section "label" field
452	export,iges,perBodyLayers	Put each body on own layer
453	export,step,lengthTol	This parameter indicates the smallest distance in model space units that is considered as discernible by the downstream system. Any coordinate locations in the output file that are separated by a distance smaller than the length uncertainty tolerance will be considered to be coincident.
454	export,step,protocol	Select the appropriate STEP protocol for the downstream system.
455	export,step,merge	A component that has been selectively imported from a STEP assembly can be merged back into that assembly. Use to update the content of an existing STEP file so that it uses the new component definition. If this option is chosen, then the STEP file must already exist. None of the other components in the assembly will be affected by this operation.

456	`export,step,singlepart`	for exporting models with multiple bodies. When this is not checked then each body has its own product definition section in the STEP file. For some CAD systems, e.g. UG, this will produce multiple part files when the STEP file is read. If you want all the bodies in a single part file then check this option and the STEP file will only have one product definition, and CAD systems will create one-part file with multiple solids in it.
457	`export,step,nonmanshells`	A non-manifold shell is a collection of connected faces containing an edge that is shared by more than 2 faces, e.g. a T-junction of faces. This can be used to automatically split up the shells along their non-manifold edges. The default setting would normally be OFF as most systems do not accept non-manifold shells. For STEP the default is ON.
458	`export,step,polyshells`	An option for the stitching of polyshells during import
459	`export,step,description`	To populate the user definable fields in the STEP spec
460	`export,step,name`	To populate the user definable fields in the STEP spec
461	`export,step,author`	To populate the user definable fields in the STEP spec
462	`export,step,organisation`	To populate the user definable fields in the STEP spec
463	`export,step,authorisation`	To populate the user definable fields in the STEP spec
464	`export,step,codePage`	Code page number for non-Unicode characters
465	`export,stl,method`	Two methods can be used to control the quality of the facetted model. The method used to generate the STL file can produce markedly different results. Choose **CURVATURE** to generate an STL file containing the minimum number of facets. The facets created may have large aspect ratios and may be unsuitable for some downstream applications. Choose **QUALITY** to create facets that are as regular as possible within the constraints set by the faceting parameters.
466	`export,stl,facetSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.
467	`export,stl,facetTurn`	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
468	`export,stl,facetLength`	This parameter specifies the maximum facet size in model space units based on edge length. If no restriction is to be placed on the maximum facet size then this parameter can be left unspecified. CADfix will calculate the facet length as approximately 1/100th times the overall model diameter.
469	`export,stl,expandRate`	An expansion rate is used for the generation of quality triangular facets. The rate will control how facets sizes will increase as they move away from the edge of a face. A number between 1.0 and 3.0 and represents the expansion factor for facets. A bigger factor means that you will get larger elements in the middle of faces while a factor of 1.0 means the elements will not grow much away from the boundary.
470	`export,stl,minFacets`	This parameter specifies the minimum number of faces to be created on each edge.
471	`export,stl,minThrough`	This parameter is designed to refine the facets in narrow regions giving the user specified number of facets between opposite edges in a narrow face or part of a face.
472	`export,stl,smooth`	The triangular faceting for quality STL will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
473	`export,stl,useSources`	Make use of on any FLITE3D sources when generating the STL faceted model
474	`export,stl,openShells`	By default, a model can only be exported to STL if the faces define one or more solid bodies. Use to allow any unclosed shells to be exported to STL.
475	`export,stl,multiSolids`	multiple solid (or body) models can be written into an STL file as separately grouped entities

476	`export,stl,perBodyMultiSolids`	When selecting the STL facets for each CADfix body are grouped into separate STL "solid" entities in the STL ASCII export file. Each solid group would also carry the assigned label as a tag.
477	`export,stl,multiFiles`	Multiple STL files are generated on export based on a number of criteria from separate bodies to colours to labels.
478	`export,stl,sharedFaces`	Used when exporting multiple bodies. Rather than creating duplicate sets of facets on shared faces just a single shared set of facets are used.
479	`export,stl,zones`	When creating the STL facets for exporting respect any zones that exist within the model. This means that internal geometry edges in the zones will be ignored during the STL faceting.
480	`export,stl,explode`	Explode STL of an assembly model into separate files
481	`export,stl,wraps`	Export STL result of a shrink wrap
482	`export,stl,format`	Select either ASCII or Binary STL export file type
483	`export,stl,R-FG-FLD`	Checking for STL facet fold integrity on export
484	`export,stl,R-FG-ZAR`	Checking for STL facet zero area integrity on export
485	`export,stl,R-FG-INT`	Checking for STL facet integrity on export
486	`export,stl,R-FG-FLD,angle`	Angle value for fold integrity
487	`export,stl,R-FG-ZAR,areamin`	Area value for zero area integrity
488	`export,stl,R-FG-INT,full`	Full facet integrity check
489	`export,3ds,method`	Two methods can be used to control the quality of the facetted model. The method used to generate the 3DS file can produce markedly different results. Choose **CURVATURE** to generate an 3DS file containing the minimum number of facets. The facets created may have large aspect ratios and may be unsuitable for some downstream applications. Choose **QUALITY** to create facets that are as regular as possible within the constraints set by the faceting parameters.
490	`export,3ds,facetSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.
491	`export,3ds,facetTurn`	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
492	`export,3ds,facetLength`	This parameter specifies the maximum facet size in model space units based on edge length. If no restriction is to be placed on the maximum facet size then this parameter can be left unspecified. CADfix will calculate the facet length as approximately 1/100th times the overall model diameter.
493	`export,3ds,expandRate`	An expansion rate is used for the generation of quality triangular facets. The rate will control how facets sizes will increase as they move away from the edge of a face. A number between 1.0 and 3.0 and represents the expansion factor for facets. A bigger factor means that you will get larger elements in the middle of faces while a factor of 1.0 means the elements will not grow much away from the boundary.
494	`export,3ds,minFacets`	This parameter specifies the minimum number of faces to be created on each edge.
495	`export,3ds,minThrough`	This parameter is designed to refine the facets in narrow regions giving the user specified number of facets between opposite edges in a narrow face or part of a face.
496	`export,3ds,smooth`	The triangular faceting for quality 3DS will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
497	`export,3ds,openShells`	By default, a model can only be exported to 3DS if the faces define one or more solid bodies. Use to allow any unclosed shells to be exported to 3DS.
498	`export,3ds,zones`	When creating the 3DS facets for exporting respect any zones that exist within the model. This means that

		internal geometry edges in the zones will be ignored during the 3DS faceting.
499	`export,3ds,wraps`	Export 3DS result of a shrink wrap
500	`export,acis,resabs`	This parameter indicates the modelling tolerance used by the target system and defaults to 1.0E-06.
501	`export,acis,tolerant`	To define every geometric entity in the ACIS file with reference to a tolerance.
502	`export,acis,bodyChecker`	To check the validity of the ACIS file.
503	`export,acis,checkerLevel`	The **Checker level** export options are available to control the complexity level of the body checker. The level can be: 10, 20, 30, 40, 50, 60 or 70. The default is **20**. The higher the level the more checking that is done and the slower the export.
504	`export,acis,version`	specify the export ACIS version number
505	`export,acis,nonmanshells`	Non manifold shells can be exported to ACIS
506	`export,TecnomatixMLP,resabs`	This parameter indicates the modelling tolerance used by the target system and defaults to 1.0E-06.
507	`export,TecnomatixMLP,tolerant`	To define every geometric entity in the TechnomatixMPL file with reference to a tolerance.
508	`export,TecnomatixMLP,bodyChecker`	To check the validity of the TechnomatixMPL file.
509	`export,TecnomatixMLP,checkerLevel`	The **Checker level** export options are available to control the complexity level of the body checker. The level can be: 10, 20, 30, 40, 50, 60 or 70. The default is **20**. The higher the level the more checking that is done and the slower the export.
510	`export,TecnomatixMLP,centrelines`	Create hole centrelines
511	`export,parasolid,version`	specify the export Parasolid version number
512	`export,parasolid,tolerant`	to define every edge in the model with reference to a tolerance
513	`export,parasolid,bodyChecker`	to check the validity of the Parasolid file
514	`export,parasolid,edgetolType`	Every edge defined in a Parasolid model has a tolerance associated with it. By default, **AUTO** would be selected. Choose **LOC** to set the edge tolerances locally for each edge, or **GLOB** to assign the same tolerance value to every edge in the model.
515	`export,parasolid,edgetol`	If the edge tolerances are to be set locally for each edge, then the **tolerance.** parameter indicates the minimum tolerance that will be assigned to a particular edge, otherwise tolerance should be set with reference to the modelling tolerance used by the downstream system.
516	`export,parasolid,scalef`	The Parasolid unit of length is meters. If the model was originally defined using a different unit of length this parameter can be used to scale the model on export, as appropriate.
517	`export,parasolid,maxerr`	Choose an appropriate value for **Maximum errors** to report
518	`export,parasolid,sheetBodies`	Open shells as individual faces permit the export of mixed models.
519	`export,parasolid,multibodyasasm`	Create and export an assembly for multiple geometry models
520	`export,parasolid,format`	There are three output format options; **ASCII**, a machine dependent **BINARY** format and a machine independent binary or **NEUTRAL** format.
521	`export,parasolid,colhier`	Default is ANY CHILD PARENT
522	`export,cadds,pointTol`	This parameter indicates the smallest distance, in model space units, that is considered as discernible by the downstream system. Any coordinate locations in the output file that are separated by a distance smaller than the point coincidence tolerance will be considered to be coincident.
523	`export,cadds,saveAs`	The user can choose to save in the CADDS older 4X or newer 5X format
524	`export,catia,modelDimension`	This is the work space used by CATIA affecting the tolerance of internal operations such as; projections, identical curves, closing of volumes and infinity within the model. All being calculated as a function of the

		model dimension. The model dimension settings is in millimetres and if not set will default within CATIA to 10000.
525	`export,catia,exportSOLIDE`	CADfix bodies maybe exported to CATIA v4 as **SOL** or **VOL** entities. The default is to export bodies as *SOL and open shells as *VOL. If the set being exported is a mixture of bodies and open shells then all the bodies and open shells will be exported according to the body preference, i.e. we can't export bodies as *SOL and open shells as *VOL in the same export.
526	`export,catia,solidmPipes`	Option to convert original CATIA V4 pipe bodies in to SOLIDM bodies on export to V4
527	`export,catia,shellsAsVols`	Open shells maybe exported as either **VOL** or **FAC** entities. The open shell *VOL are strictly invalid in CATIA V4, but as we can't export skins (*SKN) it is the next best thing as it actually preserves the topology between the faces. CATIA V4 users can "extract" *SKN entities from these. Exporting skins as *FAC produces unconnected faces in V4. If the set being exported is a mixture of bodies and open shells then all the bodies and open shells will be exported according to the body preference, i.e. we can't export bodies as *SOL and open shells as *VOL in the same export.
528	`export,catia,mirrorKeepOriginal`	Option to keep original parts affected by a mirror transform when the transform is removed on export
529	`export,catia,mirrorPlane`	Option to enable the auto-generation of a mirrored copy of the model during export
530	`export,catia,assemblyAsDittos`	Option to convert assembly models to CATIA V4 dittos entities
531	`export,catia,exportLayerFilters`	Option to export layer filter definitions to CATIA V4
532	`export,catia,exporter`	CATIA export interface
533	`export,catia,solidmSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface when generating SOLIDM bodies in CATIA V4 export.
534	`export,catia,solidmTurn`	This parameter specifies the maximum angle that a facet can turn through before it is too large and must be refined, when generating SOLIDM bodies in CATIA V4 export
535	`export,catia,InterOpAPI`	choose Interop option either **legacy** or **connect**
536	`export,catia5,saveVersion`	Option to identify the version of CATIA V5 to write, e.g. CATIA5R24
537	`export,catia5,hybrid`	Option to create hybrid PartBody entities during V5 export. If disabled, then Geometric Set will be generated.
538	`export,catia5,promoteGeomNodes`	Promote any (geometry) nodes in the top-level assembly node to be full blown components
539	`export,catia5,singleCATProduct`	++discontinued++
540	`export,catia5,allCGR`	Convert all parts to CGR on exporting to V5
541	`export,catia5,exporter`	Option to set the CATIA V5 export method <deprecated>
542	`export,catia5,cgrSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface when generating CGR facets
543	`export,catia5,cgrTurn`	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined, when generating CGR facets
544	`export,catia5,asmFileNames`	Controls the file names used when exporting an assembly to V5. If set to FILENAME, then the original (as imported) filename will be re-used on export. If set to PARTNAME then the assigned label on the assembly node will be used instead. If you want to preserve the same file structure when import/exporting a V5 assembly, then select FILENAME.
545	`export,catia5,InterOpAPI`	choose Interop option either **legacy** or **connect**
546	`export,catia5,mkshells`	map the exported CATIA V5 entities to shells rather than solids

547	export,catia5,promoteGeomNodes	An option that only affects hybrid assemblies where the top-level node has a "(geometry)" node in it. When this option is enabled then each part in the (geometry) set will be exported as a separate component referenced by the top-level node.
548	export,proe,tolerance	tolerance used for the export of Pro/E data
549	export,proe,tolType	Choose **Rel** to specify the tolerance as a factor of the model size. Choose **Abs** to specify the tolerance as an absolute value.
550	export,proe,version	specify the export CREO version number
551	export,proe,names	Use original component names
552	export,proe,eachInTurn	++Not used for batch processing++
553	export,proe,asFacets	Export CREO model as facets
554	export,proe,facetSag	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.
555	export,dxf,version	specify DXF version number
556	export,dxf,bodyChecker	The ACIS body checker maybe run on the exported data.
557	export,dxf,tolerant	As the ACIS format forms the basis of a DXF export this is used to define every geometric entity in the file with reference to a tolerance.
558	export,dxf,explodeFacetted	Explode facetted surfaces
559	export,ansys,btol	This parameter sets the tolerance to be used by ANSYS when a Boolean operation is performed on the model. Set the value to OFF if Boolean operations are not to be performed in ANSYS.
560	export,femgv,modelTol	This parameter sets the geometric modelling tolerance to be used in the downstream system
561	export,femgv,workspace	This parameter indicates the default size of the workspace buffer when opening the model in FEMGV
562	export,femgv,lineDIV	Use this parameter to indicate the required number of divisions on each edge in the model
563	export,femgv,history	A log file recording the commands generated during the FEMGV import process can be automatically created by checking this option
564	export,sc03,geomis2d	Option to flag the SC03 export as being 2D
565	export,sc03,tessellate	controls whether the tessellation data is written to an export file
566	export,sc03,TT	Thin and thick subdivided geometry is export
567	export,sc03,ftol	facet tolerance for the tessellation
568	export,sc03,PROX2D	Included in the export is proximity 2D data
569	export,sc03,content	Option to control whether geometry, mesh or both are exported to SC03
570	export,sc03,bodiesOnly	Option to export only solid bodies to SC03
571	export,sc03,edgeTags	edge labels maybe passed to the SC03 model export file
572	export,sc03,imprintEdges	Edge Imprinting may be carried out on the fly during export.
573	export,sc03,imprintEdgesTol	Used to select the imprinting Maximum gap
574	export,sc03,imprintBodies	Body Imprinting may be carried out on the fly during export.
575	export,sc03,imprintBodiesTol	Used to select the imprinting Maximum gap
576	export,sc03,imprintBodiesAng	Used to select the imprinting Maximum draft angle
577	export,sc03,matchPairs	Match existing face pairs during imprinting
578	export,sc03,uniqueNames	Option to ensure all SC03 parts are given unique names
579	export,gridpro,method	Three methods can be used to control the quality of the facetted model. The method used to generate the GridPro file can produce markedly different results. **Quality** is the default and used to create facets that are as regular as possible within the constraints set by the faceting parameters. Choose **Curvature** to generate a GridPro file containing the minimum number of facets. The facets created may have large aspect ratios and may be unsuitable for some downstream applications.

		Choose **Mesh** to generate a GridPro file using an existing mesh. An existing mesh could only exist if some external TCL script run during the batch process had generated it.
580	`export,gridpro,facetSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.
581	`export,gridpro,facetTurn`	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
582	`export,gridpro,facetLength`	This parameter specifies the maximum facet size in model space units based on edge length. If no restriction is to be placed on the maximum facet size, then this parameter can be left unspecified. CADfix will calculate the facet length as approximately 1/100th times the overall model diameter.
583	`export,gridpro,expandRate`	An expansion rate is used for the generation of quality triangular facets. The rate will control how facets sizes will increase as they move away from the edge of a face. A number between 1.0 and 3.0 and represents the expansion factor for facets. A bigger factor means that you will get larger elements in the middle of faces while a factor of 1.0 means the elements will not grow much away from the boundary.
584	`export,gridpro,minFacets`	This parameter specifies the minimum number of faces to be created on each edge.
585	`export,gridpro,minThrough`	This parameter is designed to refine the facets in narrow regions giving the user specified number of facets between opposite edges in a narrow face or part of a face.
586	`export,gridpro,smooth`	The triangular faceting for quality will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
587	`export,gridpro,useSources`	Make use of on any mesh division control sources when generating the faceted model
588	`export,gridpro,zones`	When creating the GridPro facets for exporting respect any zones that exist within the model. This means that internal geometry edges in the zones will be ignored during the GridPro faceting.
589	`export,dem,method`	Two methods can be used to control the quality of the facetted model. The method used to generate the DEM file can produce markedly different results. Choose **Curvature** to generate a DEM file containing the minimum number of facets. The facets created may have large aspect ratios and may be unsuitable for some downstream applications. Choose **Quality** to create facets that are as regular as possible within the constraints set by the faceting parameters.
590	`export,dem,facetSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.
591	`export,dem,facetTurn`	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
592	`export,dem,facetLength`	This parameter specifies the maximum facet size in model space units based on edge length. If no restriction is to be placed on the maximum facet size, then this parameter can be left unspecified. CADfix will calculate the facet length as approximately 1/100th times the overall model diameter.
593	`export,dem,expandRate`	An expansion rate is used for the generation of quality triangular facets. The rate will control how facets sizes will increase as they move away from the edge of a face. A number between 1.0 and 3.0 and represents the expansion factor for facets. A bigger factor means that you will get larger elements in the middle of faces while

		a factor of 1.0 means the elements will not grow much away from the boundary.
594	`export,dem,minFacets`	This parameter specifies the minimum number of faces to be created on each edge.
595	`export,dem,minThrough`	This parameter is designed to refine the facets in narrow regions giving the user specified number of facets between opposite edges in a narrow face or part of a face.
596	`export,dem,smooth`	The triangular faceting for quality will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
597	`export,dem,useSources`	Make use of on any mesh division control sources when generating the faceted model
598	`export,dem,zones`	When creating the DEM facets for exporting respect any zones that exist within the model. This means that internal geometry edges in the zones will be ignored during the DEM faceting.
599	`export,xpatch,method`	Two methods can be used to control the quality of the facetted model. The method used to generate the **Xpatch** file can produce markedly different results. Choose **Curvature** to generate a **Xpatch** file containing the minimum number of facets. The facets created may have large aspect ratios and may be unsuitable for some downstream applications. Choose **Quality** to create facets that are as regular as possible within the constraints set by the faceting parameters.
600	`export,xpatch,facetSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.
601	`export,xpatch,facetTurn`	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
602	`export,xpatch,facetLength`	This parameter specifies the maximum facet size in model space units based on edge length. If no restriction is to be placed on the maximum facet size, then this parameter can be left unspecified. CADfix will calculate the facet length as approximately 1/100th times the overall model diameter.
603	`export,xpatch,expandRate`	An expansion rate is used for the generation of quality triangular facets. The rate will control how facets sizes will increase as they move away from the edge of a face. A number between 1.0 and 3.0 and represents the expansion factor for facets. A bigger factor means that you will get larger elements in the middle of faces while a factor of 1.0 means the elements will not grow much away from the boundary.
604	`export,xpatch,minFacets`	This parameter specifies the minimum number of faces to be created on each edge.
605	`export,xpatch,minThrough`	This parameter is designed to refine the facets in narrow regions giving the user specified number of facets between opposite edges in a narrow face or part of a face.
606	`export,xpatch,smooth`	The triangular faceting for quality will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
607	`export,xpatch,useSources`	Make use of on any mesh division control sources when generating the faceted model
608	`export,xpatch,zones`	When creating the Xpatch facets for exporting respect any zones that exist within the model. This means that internal geometry edges in the zones will be ignored during the Xpatch faceting.
609	`export,centaur,tol`	The Spline fit tolerance
610	`export,centaur,tangType`	end tangency type
611	`export,centaur,surfEndCond`	Controls surface end conditions
612	`export,centaur,tangTol`	End tangency fitting tolerance in degrees
613	`export,centaur,equalSpacedEdges`	set the knot spacing to be equal

614	`export,centaur,aspectRatio`	Maximum patch aspect ratio in degrees
615	`export,centaur,growthRatio`	When using the Adaptive option for surface splines the surfaces splines are controlled with a **Max.** growth rate option
616	`export,centaur,equalSpacedSurfs`	Surface splines can be controlled by setting **Equal spaced**
617	`export,centaur,scalef`	The local spline export scale factor tolerance
618	`export,centaur,gridU`	Number of patches for surfaces in the U direction
619	`export,centaur,gridV`	Number of patches for surfaces in the V direction
620	`export,centaur,gridT`	Number of segments for curves T
621	`export,centaur,mindiv`	Set the minimum division
622	`export,centaur,maxdiv`	Set the maximum division
623	`export,centaur,initgrid`	Set the initial grid parameters
624	`export,centaur,maxgrid`	St the maximum grid size (UxV)
625	`export,flite3d,tol`	The Spline fit tolerance
626	`export,flite3d,tangType`	end tangency type
627	`export,flite3d,surfEndCond`	Controls surface end conditions
628	`export,flite3d,tangTol`	End tangency fitting tolerance in degrees
629	`export,flite3d,equalSpacedEdges`	set the knot spacing to be equal
630	`export,flite3d,aspectRatio`	Maximum patch aspect ratio in degrees
631	`export,flite3d,growthRatio`	When using the Adaptive option for surface splines the surfaces splines are controlled with a **Max.** growth rate option
632	`export,flite3d,equalSpacedSurfs`	Surface splines can be controlled by setting **Equal spaced**
633	`export,flite3d,scalef`	The local spline fit parameter tolerance
634	`export,flite3d,gridU`	Number of patches for surfaces in the U direction
635	`export,flite3d,gridV`	Number of patches for surfaces in the V direction
636	`export,flite3d,gridT`	Number of segments for curves T
637	`export,flite3d,mindiv`	Set the minimum division
638	`export,flite3d,maxdiv`	Set the maximum division
639	`export,flite3d,initgrid`	Set the initial value
640	`export,flite3d,maxgrid`	St the maximum growth value
641	`export,solar,tol`	The Spline fit tolerance
642	`export,solar,tangType`	end tangency type
643	`export,solar,surfEndCond`	Controls surface end conditions
644	`export,solar,tangTol`	End tangency fitting tolerance in degrees
645	`export,solar,equalSpacedEdges`	set the knot spacing to be equal
646	`export,solar,aspectRatio`	Maximum patch aspect ratio in degrees
647	`export,solar,growthRatio`	When using the Adaptive option for surface splines the surfaces splines are controlled with a **Max.** growth rate option
648	`export,solar,equalSpacedSurfs`	Surface splines can be controlled by setting **Equal spaced**
649	`export,solar,scalef`	The local spline fit parameter tolerance
650	`export,solar,gridU`	Number of patches for surfaces in the U direction
651	`export,solar,gridV`	Number of patches for surfaces in the V direction
652	`export,solar,gridT`	Number of segments for curves T
653	`export,solar,mindiv`	Set the minimum division
654	`export,solar,maxdiv`	Set the maximum division
655	`export,solar,initgrid`	Set the initial value
656	`export,solar,maxgrid`	St the maximum growth value
657	`export,cadoe,modelDimension`	This is the work space used by CADOE affecting the tolerance of internal operations
658	`export,jt,fileFormat`	The versions supported
659	`export,jt,fileStructure`	**Per part** - All assembly nodes in a product structure hierarchy are stored in a single JT file, and each part node in the hierarchy is stored in an individual JT file in a subdirectory that is of the same name as the assembly JT file. **Fully shattered** - Each product structure node in the hierarchy is stored in an individual JT file. **Monolithic** - All product structure is stored in a single JT file.
660	`export,jt,includeBrep`	Whether to include the BREP in the file
661	`export,jt,includeMassProps`	Mass properties are included with the export file

662	`export,jt,numLODs`	**L**evel **O**f **D**etail values are used for creating multiple tessellations data sets. There are three options: 1, 2 or 3 defining different levels of model faceting.
663	`export,jt,multiSolidsAsAsm`	Whether to define multi-solids models as an assembly
664	`export,jt,useConfigFile`	Specifying whether an external configuration file is to be used.
665	`export,jt,configFile`	Path defining the configuration file for externally control supporting functions in a deployed application
666	`export,pdf,facetSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.
667	`export,pdf,facetTurn`	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
668	`export,pdf,useTemplateFile`	User defined PDF template definition to be user
669	`export,pdf,templateFile`	PDF template file path
670	`export,pdf,viewBox`	PDF template definition
671	`export,pdf,pagesize`	PDF template page size
672	`export,pdf,orientation`	PDF template document orientation
673	`export,pdf,title`	PDF template document title
674	`export,pdf,author`	PDF template document author
675	`export,pdf,subject`	PDF template document subject
676	`export,pdf,creator`	PDF template document creator
677	`export,pdf,LOD`	PDF template level of detail
678	`export,pdf,chordHeightRatio`	PDF template definition
679	`export,pdf,minimalTriAngle`	PDF template definition
680	`export,pdf,lineWidth`	PDF template document thickness of line
681	`export,agps,pcurveTol`	tolerance for control pcurve projection
682	`export,agps,createCSC`	create surface connections
683	`export,matlab,facetSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.
684	`export,matlab,facetTurn`	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
685	`export,matlab,facetLength`	This parameter specifies the maximum facet size in model space units based on edge length. If no restriction is to be placed on the maximum facet size, then this parameter can be left unspecified. CADfix will calculate the facet length as approximately 1/100th times the overall model diameter.
686	`export,matlab,minFacets`	This parameter specifies the minimum number of faces to be created on each edge.
687	`export,mesh,type`	Export of mesh maybe from three sources: free **tri**angular Delaunay meshing created using the **Parameter** settings, **quad** meshing using a combination of PAVE mesh and mapped mesh using the **Parameter** settings, or from any existing mesh that may have been created with the Mesh Manager.
688	`export,mesh,method`	Two methods can be used to control the quality of the triangular elements (not relevant to **Quad** meshing). Choose **Minimum number** to generate a mesh file containing the minimum number of triangular elements. The facets created may have large aspect ratios and may be unsuitable for some downstream applications. Choose **Quality facets** to create triangular mesh elements that are as regular as possible within the constraints set by the faceting parameters.
689	`export,mesh,format`	A **Format** of either **FME** (CADfix's own mesh file format) or **NASTRAN** (bulk data deck) or **ABAQUS (.ABA)** surfaces elements or **CGNS** may be chosen
690	`export,mesh,facetSag`	This parameter specifies the maximum allowable distance between a facet and its underlying surface. The default maximum facet sag is set according to the size of the model.

691	`export,mesh,facetTurn`	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
692	`export,mesh,facetLength`	This parameter specifies the maximum facet size in model space units based on edge length. If no restriction is to be placed on the maximum facet size, then this parameter can be left unspecified. CADfix will calculate the facet length as approximately 1/100th times the overall model diameter.
693	`export,mesh,expandRate`	An expansion rate is used for the generation of quality triangular facets. The rate will control how facets sizes will increase as they move away from the edge of a face. A number between 1.0 and 3.0 and represents the expansion factor for facets. A bigger factor means that you will get larger elements in the middle of faces while a factor of 1.0 means the elements will not grow much away from the boundary.
694	`export,mesh,minFacets`	This parameter specifies the minimum number of faces to be created on each edge.
695	`export,mesh,minThrough`	This parameter is designed to refine the facets in narrow regions giving the user specified number of facets between opposite edges in a narrow face or part of a face.
696	`export,mesh,smooth`	The triangular faceting for quality will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
697	`export,mesh,useSources`	Make use of on any mesh division control sources when generating the faceted model
698	`export,mesh,zones`	When creating the mesh facets for exporting respect any zones that exist within the model. This means that internal geometry edges in the zones will be ignored during the mesh faceting.

Appendix 2: Example CADfix CWC File

```
cwc,version                                                  : 1200

preferredUnits                     [MM, CM, M, INCH, FOOT] : MM
maxWizardTol                                  [OFF, value] : OFF
workSet                                        [ALL, value] : ALL
continueOnError                                     [0, 1] : 1
saveBetweenOps                                      [0, 1] : 0
saveOnExport                                        [0, 1] : 0
diagnosticCheck                     [ALWAYS, FAIL, NEVER] : FAIL
qualityStandard                [nocheck, jama45, jamaALL] : nocheck
longNames                              [DEFAULT, 0, 1] : DEFAULT

import                                                  [1] : 1

import,system                     [IGES, STEP, STL, ACIS] : IGES
import,colours                                      [0, 1] : 1
import,layers                                       [0, 1] : 1
import,labels                                       [0, 1] : 1
import,userAttrs                                    [0, 1] : 0
import,assemblies,mode          [COMPLETE, EACH, SELECTED] : COMPLETE
import,assemblies,subpartid              [NULL, value] : NULL
import,assemblies,explode                           [0, 1] : 0
import,assemblies,emptyComponents                   [0, 1] : 1
import,assemblies,freeLines                         [0, 1] : 1
import,assemblies,freeSurfs                         [0, 1] : 1
import,assemblies,freeFaces                         [0, 1] : 1
import,assemblies,freePartsName            [NULL, value] : NULL
import,autoGroup,colour                                  : {{apply 0} {type {}}}
import,autoGroup,layer                                   : {{apply 0} {type {}} {value ANY}}
import,temporaryFiles                               [0, 1] : 1
import,resolveDegenerateLines            [ALWAYS, NEVER] : ALWAYS
import,convertUnits        [OFF, MM, CM, M, INCH, FOOT] : OFF
import,scaleModel                             [OFF, value] : OFF

################ IGES Import parameters ################
import,iges,useDefaults                         [OFF, ON] : ON
import,iges,definitionEntities                  [OFF, ON] : DEFAULT
import,iges,blankedEntities                     [OFF, ON] : DEFAULT
import,iges,freeLines                           [OFF, ON] : DEFAULT
import,iges,freeLinesFont          [0, 1, 2, 3, 4, 5] : DEFAULT
import,iges,constructionGeom                    [OFF, ON] : DEFAULT
import,iges,linesAsNURBS                        [OFF, ON] : DEFAULT
import,iges,splitCreasedNURBS                   [OFF, ON] : DEFAULT
import,iges,turningAngle                         [value] : DEFAULT
import,iges,splitClosedNURBS                    [OFF, ON] : DEFAULT
import,iges,pcurves             [PREF, GAPS, ON, OFF] : DEFAULT
import,iges,calcGoodTol                         [OFF, ON] : DEFAULT
import,iges,mergeDuplicates                     [OFF, ON] : DEFAULT
import,iges,boundUntrimmed                      [OFF, ON] : DEFAULT
import,iges,completeBoundaries                  [OFF, ON] : DEFAULT
import,iges,stitchFaces                         [OFF, ON] : DEFAULT
import,iges,buildBodies                         [OFF, ON] : DEFAULT
import,iges,extraAttributes                     [OFF, ON] : DEFAULT
######################## END ########################

############### STEP Import parameters ################
import,step,freeLines                              [0, 1] : DEFAULT
import,step,freeSurfs                               [0, 1] : DEFAULT
import,step,freeFaces                               [0, 1] : DEFAULT
import,step,splitDiscontinuous                     [0, 1] : DEFAULT
import,step,splitAngle                           [value] : DEFAULT
######################## END ########################

############### VDAFS Import parameters ################
import,vdafs,freeLines                             [0, 1] : DEFAULT
import,vdafs,freeSurfs                             [0, 1] : DEFAULT
import,vdafs,freeFaces                             [0, 1] : DEFAULT
import,vdafs,splitDiscontinuous                   [0, 1] : DEFAULT
import,vdafs,splitAngle                          [value] : DEFAULT
######################## END ########################

################ STL Import parameters ################
import,stl,mergeVertices                      [NO, YES] : DEFAULT
import,stl,facettedSurfaces                   [NO, YES] : YES
import,stl,traceShells                         [OFF, ON] : DEFAULT
######################## END ########################
```

```
################# ACIS Import parameters #################
import,acis,freeLines                        [0, 1] : DEFAULT
import,acis,freeSurfs                        [0, 1] : DEFAULT
import,acis,freeFaces                        [0, 1] : DEFAULT
import,acis,splitDiscontinuous               [0, 1] : DEFAULT
import,acis,splitAngle                      [value] : DEFAULT
######################### END #########################


################# DXF/DWG Import parameters #################
import,dxf,freeLines                         [0, 1] : DEFAULT
import,dxf,freeSurfs                         [0, 1] : DEFAULT
import,dxf,freeFaces                         [0, 1] : DEFAULT
import,dxf,splitDiscontinuous                [0, 1] : DEFAULT
import,dxf,splitAngle                       [value] : DEFAULT
######################### END #########################


############## Parasolid Import parameters ##############
import,parasolid,freeLines                   [0, 1] : DEFAULT
import,parasolid,freeSurfs                   [0, 1] : DEFAULT
import,parasolid,freeFaces                   [0, 1] : DEFAULT
import,parasolid,units           [MM, CM, M, INCH, FOOT] : MM
import,parasolid,splitDiscontinuous          [0, 1] : DEFAULT
import,parasolid,splitAngle                 [value] : DEFAULT
import,parasolid,explodeAssembly             [0, 1] : 0
import,parasolid,splitClosedNURBS            [0, 1] : 1
import,parasolid,splitSeamless               [0, 1] : 1
######################### END #########################


################# CADDS Import parameters #################
import,cadds,freeLines                       [0, 1] : DEFAULT
import,cadds,freeSurfs                       [0, 1] : DEFAULT
import,cadds,pointTol                       [value] : DEFAULT
import,cadds,nameByPart                      [0, 1] : 1
######################### END #########################


################# CATIA4 Import parameters ##############
import,catia,importNoShows                   [0, 1] : DEFAULT
import,catia,importNoPicks                   [0, 1] : DEFAULT
import,catia,importOnlyRoot                  [0, 1] : DEFAULT
import,catia,importOnlyMasterWorkspace       [0, 1] : DEFAULT
import,catia,dittosAsAssembly                [0, 1] : 1
import,catia,solidmAsFacets                  [0, 1] : 0
import,catia,importPipes                     [0, 1] : DEFAULT
import,catia,layerFilter            [-, LAYCUR, ALL] : ALL
import,catia,useAnalyticSurfs                [0, 1] : DEFAULT
import,catia,3DCurveRegen                    [0, 1] : DEFAULT
import,catia,3DCurvePref                     [0, 1] : DEFAULT
import,catia,freeLines                       [0, 1] : 0
import,catia,freeSurfs                       [0, 1] : 0
import,catia,freeFaces                       [0, 1] : 1
import,catia,splitDiscontinuous              [0, 1] : DEFAULT
import,catia,splitAngle                     [value] : DEFAULT
import,catia,importer              [DEFAULT, value] : DEFAULT
######################### END #########################


################# CATIA5 Import parameters ##############
import,catia5,importHidden                   [0, 1] : DEFAULT
import,catia5,importCGR                      [0, 1] : DEFAULT
import,catia5,freeLines                      [0, 1] : DEFAULT
import,catia5,freeSurfs                      [0, 1] : DEFAULT
import,catia5,freeFaces                      [0, 1] : DEFAULT
import,catia5,importer             [DEFAULT, value] : DEFAULT
import,catia5,labelType                     [value] : ELEMENT
######################### END #########################


################# UG Import parameters #################
import,ug,importHidden                       [0, 1] : DEFAULT
import,ug,freeLines                          [0, 1] : DEFAULT
import,ug,freeSurfs                          [0, 1] : DEFAULT
import,ug,freeFaces                          [0, 1] : DEFAULT
import,ug,units                  [MM, CM, M, INCH, FOOT] : MM
import,ug,importer                 [DEFAULT, value] : DEFAULT
import,ug,splitClosedNURBS                   [0, 1] : 1
import,ug,splitSeamless                      [0, 1] : 1
######################### END #########################


############## Inventor Import parameters ##############
```

```
import,inventor,freeLines                            [0, 1] : DEFAULT
import,inventor,freeSurfs                             [0, 1] : DEFAULT
import,inventor,freeFaces                             [0, 1] : DEFAULT
import,inventor,importer                 [DEFAULT, value] : DEFAULT
######################## END ########################

################# SW Import parameters #################
import,sw,freeLines                                  [0, 1] : DEFAULT
import,sw,freeSurfs                                   [0, 1] : DEFAULT
import,sw,freeFaces                                   [0, 1] : DEFAULT
import,sw,importHidden                                [0, 1] : DEFAULT
import,sw,importSuppressed                            [0, 1] : DEFAULT
import,sw,splitClosedNURBS                            [0, 1] : 1
import,sw,splitSeamless                               [0, 1] : 1
import,sw,importer                       [DEFAULT, value] : DEFAULT
######################### END #########################

################# ProE Import parameters #################
import,proe,freeLines                                [0, 1] : DEFAULT
import,proe,freeSurfs                                 [0, 1] : DEFAULT
import,proe,freeFaces                                 [0, 1] : DEFAULT
import,proe,vertexTol                          [value] : DEFAULT
import,proe,facets                                   [0, 1] : DEFAULT
import,proe,importBlanked                             [0, 1] : DEFAULT
import,proe,importer                      [DEFAULT, value] : DEFAULT
######################### END #########################

############### ANSYS Import parameters ###############
import,ansys,freeLines                               [0, 1] : DEFAULT
import,ansys,freeSurfs                                [0, 1] : DEFAULT
import,ansys,splitDiscontinuous                       [0, 1] : DEFAULT
import,ansys,splitAngle                         [value] : DEFAULT
######################### END #########################

################# AGPS Import parameters #################
import,agps,freeLines                                [0, 1] : DEFAULT
import,agps,freeSurfs                                 [0, 1] : DEFAULT
import,agps,splitDiscontinuous                        [0, 1] : DEFAULT
import,agps,splitAngle                          [value] : DEFAULT
import,agps,keepTempEntities                          [0, 1] : DEFAULT
import,agps,pcurveToXYZTol                      [value] : DEFAULT
######################### END #########################

################# SC03 Import parameters #################
import,sc03,userAttrs                                [0, 1] : DEFAULT
import,sc03,strictPMPLUS                              [0, 1] : DEFAULT
######################### END #########################

################# JT Import parameters #################
import,jt,freeLines                                  [0, 1] : DEFAULT
import,jt,freeSurfs                                   [0, 1] : DEFAULT
import,jt,freeFaces                                   [0, 1] : DEFAULT
import,jt,splitClosedNURBS                            [0, 1] : DEFAULT
import,jt,splitSeamless                               [0, 1] : DEFAULT
import,jt,onlyFacets                                  [0, 1] : DEFAULT
######################### END #########################

repair                                               [0, 1] : 1

repair,maxTol                            [DEFAULT, value] : DEFAULT
repair,buildOption [SOLIDS, FACES, SOLIDS+FACES, UNKNOWN] : SOLIDS
repair,boundUntrimmed                 [AUTO, ALWAYS, NEVER] : AUTO
repair,approximateSurfaces                           [0, 1] : 0
repair,voids                        [Delete, Remove, Keep] : Delete
repair,keepUnused                                    [0, 1] : 0
repair,deleteUnusedFaces                             [0, 1] : 0

transform                                            [0, 1] : 0

transform,joinBodies,apply                           [0, 1] : 0
transform,joinBodies,tol                 [DEFAULT, value] : DEFAULT
transform,joinBodies,mode                  [UNITE, SHARE] : UNITE
transform,joinBodies,allowVoids                      [0, 1] : 0
transform,joinFragmented,apply                       [0, 1] : 0
transform,joinFragmented,tol                   [value] : DEFAULT
transform,joinFaces,apply                            [0, 1] : 0
transform,joinFaces,type                 [SLIVERS, ALL] : SLIVERS
transform,joinFaces,width                [DEFAULT, value] : DEFAULT
```

```
transform,joinFaces,faceangle                      [value] : 1.0
transform,joinFaces,edgeangle                      [value] : 5.0
transform,joinFaces,filtermultiloop                [0, 1] : 1
transform,joinFaces,filterlarge                    [0, 1] : 1
transform,joinFaces,filtermultisided               [0, 1] : 1
transform,joinFaces,rejectthreesided               [0, 1] : 0
transform,joinFaces,rejectmultisided               [0, 1] : 0
transform,joinFaces,matchlayers                    [0, 1] : 1
transform,joinFaces,matchcolours                   [0, 1] : 1
transform,joinFaces,resolution                     [value] : 50
transform,joinFaces,change             [DEFAULT, value] : DEFAULT
transform,joinEdges,apply                          [0, 1] : 0
transform,joinEdges,angle                          [value] : 1.0
transform,collapse,apply                           [0, 1] : 0
transform,collapse,type                [FACES, EDGES] : EDGES
transform,collapse,length              [DEFAULT, value] : DEFAULT
transform,pinch,apply                              [0, 1] : 0
transform,pinch,angle                  [DEFAULT, value] : DEFAULT
transform,pinch,tol                    [DEFAULT, value] : DEFAULT
transform,pinch,filletsonly                        [0, 1] : 0
transform,pinch,fixaction [MERGE, IMPRINT, PINCH, PINCH2] : MERGE
transform,holes,removeHoles                        [0, 1] : 0
transform,holes,plugHoles                          [0, 1] : 0
transform,holes,drillTips                          [0, 1] : 0
transform,holes,type       [ALL, THROUGH, BLIND, COMPLEX] : THROUGH
transform,holes,checkSize                          [0, 1] : 1
transform,holes,size                   [DEFAULT, value] : DEFAULT
transform,holes,tol                    [DEFAULT, value] : DEFAULT
transform,holes,xsect                     [CIRC, ANY] : CIRC
transform,holes,delete                             [0, 1] : 1
transform,protrusions,remove                       [0, 1] : 0
transform,protrusions,checkSize                    [0, 1] : 1
transform,protrusions,size             [DEFAULT, value] : DEFAULT
transform,protrusions,tol              [DEFAULT, value] : DEFAULT
transform,protrusions,xsect               [CIRC, ANY] : CIRC
transform,removeSeams,apply                        [0, 1] : 0
transform,removeSeams,pln                          [0, 1] : 0
transform,removeSeams,cyl                          [0, 1] : 1
transform,removeSeams,con                          [0, 1] : 1
transform,removeSeams,sph                          [0, 1] : 0
transform,removeSeams,tor                          [0, 1] : 0
transform,removeSeams,tol              [DEFAULT, value] : DEFAULT
transform,deleteFillets,apply                      [0, 1] : 0
transform,deleteFillets,radmax         [DEFAULT, value] : DEFAULT
transform,deleteFillets,radtol         [DEFAULT, value] : DEFAULT
transform,deleteFillets,type             [CONST, VARY] : VARY
transform,deleteFillets,joinedges                  [0, 1] : 1
transform,imprintEdges,apply                       [0, 1] : 0
transform,imprintEdges,tol             [DEFAULT, value] : DEFAULT
transform,imprintBodies,apply                      [0, 1] : 0
transform,imprintBodies,tol            [DEFAULT, value] : DEFAULT
transform,externalFilter,apply                     [0, 1] : 0
transform,externalFilter,tol           [DEFAULT, value] : DEFAULT
transform,externalFilter,maxgap        [DEFAULT, value] : DEFAULT
transform,externalFilter,level [INSTANCES, BODIES, FACES] : BODIES

prepare                                            [0, 1] : 1

############# GenericIGES Prepare parameters ############
'Overlapping curves'             , R-CU-EM  : {{apply 0} {tol DEFAULT}}
'Short edges'                    , R-ED-TI  : {{apply 0} {tol DEFAULT}}
'Narrow faces'                   , R-FA-NA  : {{apply 0} {widthmin DEFAULT}}
'Closed surfaces'                , R-SU-CL  : {{apply 1} {tol DEFAULT}}
'Degenerate surfaces'            , R-SU-DC  : {{apply 0} {tol DEFAULT} {simplify 1} {flare 0}
{refit 0}}
'Degenerate surface corners'     , R-SU-DP  : {{apply 0} {angmin 5.0} {angtype BOTH}}
'Smooth surfaces'                , R-SU-SMO : {{apply 0} {tol DEFAULT}}
'Surface parameterisation'       , R-SU-PCN : {{apply 0} {condition SPEED} {rate:local 3}
{rate:global 5} {tol DEFAULT}}
'Curve parameterisation'         , R-CU-PCN : {{apply 0} {condition SPEED} {rate 3} {tol
DEFAULT}}
'Folded curves'                  , R-CU-FO  : {{apply 1} {tol DEFAULT}}
'Missing surfaces'               , R-FA-EMD : {{apply 0} {tol DEFAULT}}
'Undersize surfaces'             , R-FA-XTN : {{apply 1} {undersize 0.9999}}
'Zero-area faces'                , R-FA-ZAR : {{apply 1} {tol DEFAULT}}
'Refit faces'                    , R-FA-RPL : {{apply 0} {tol DEFAULT}}
'Surface simplification'         , R-SU-SIM : {{apply 1} {tol DEFAULT} {type all}}
'Edge simplification'            , R-CU-SIM : {{apply 0} {tol DEFAULT}}
```

```
'Non-NURBS surfaces'             , R-FA-AN  : {{apply 1} {keepplanes 0}}
'Non-NURBS edges'                , R-ED-AN  : {{apply 1}}
'Rational NURBS surfaces'        , R-SU-RAT : {{apply 0} {tol DEFAULT}}
'Rational NURBS curves'          , R-CU-RAT : {{apply 0} {tol DEFAULT}}
'Low-degree surfaces'            , R-SU-LDG : {{apply 0} {degree 3} {ignoredegr1 1}}
'Low-degree curves'              , R-CU-LDG : {{apply 0} {degree 3} {ignoredegr1 1}}
'High-degree surfaces'           , R-SU-HD  : {{apply 1} {tol DEFAULT} {degree 5}}
'High-degree curves'             , R-CU-HD  : {{apply 1} {tol DEFAULT} {degree 5}}
'Dense NURBS surface definition' , R-SU-KNT : {{apply 0} {tol DEFAULT}}
'Dense NURBS curve definition'   , R-CU-KNT : {{apply 0} {tol DEFAULT}}
'Curve knot hyper-multiplicities' , R-CU-HMK : {{apply 0} {tol DEFAULT} {maxmult 1}}
'Surface knot hyper-multiplicities', R-SU-HMK : {{apply 0} {tol DEFAULT} {maxmult 1}}
'Seamless faces'                 , R-FA-SEA : {{apply 1}}
'Oversize surfaces'              , R-FA-TRM : {{apply 1} {percent 5}}
'Non-tangent patches'            , R-SU-NT  : {{apply 1} {test G1} {angle 0.1} {c1mode REL}
{c1tol 1.0E-03} {tol DEFAULT} {factor 3}}
'Non-tangent segments'           , R-CU-NT  : {{apply 1} {test G1} {angle 0.1} {c1mode REL}
{c1tol 1.0E-03} {tol DEFAULT}}
'Non-smooth patches'             , R-SU-NS  : {{apply 0} {ratio 1.0E-03}}
'Non-smooth segments'            , R-CU-NS  : {{apply 0} {ratio 1.0E-03}}
'Degenerate tangent'             , R-CU-G1D : {{apply 0} {tol DEFAULT}}
'Join short edges'               , R-ED-JOI : {{apply 0} {tol DEFAULT}}
'Unshare'                        , R-MO-POP : {{apply 0}}
'Large edge-surface gap'         , R-FA-EG  : {{apply 1} {tol DEFAULT} {lock NOlock}}
'Sloppy surface singularity'     , R-FA-VFS : {{apply 1} {tol DEFAULT}}
######################### END #########################

export                                        [0, 1] : 1

  export,system          [GenericIGES, IGESforIDEAS12...] : GenericIGES
  export,fileSuffix                           [value] : _cf
  export,fileExtension               [DEFAULT, value] : igs
  export,always                               [0, 1] : 0
  export,restoreModel                         [0, 1] : 1
  export,assemblies,explode                   [0, 1] : 0
  export,sendLabels       [OFF, NAME, XASG, TYPART, COMBO] : XASG
  export,sendColours                        [OFF, ON] : ON
  export,sendLayers                         [OFF, ON] : ON
  export,sendUserAttrs                      [OFF, ON] : ON
  export,sendGeomForm              [OFF, XASG, AUTO] : OFF
  export,textNoteEnabled                      [0, 1] : 0

  ############# GenericIGES Export parameters #############
  export,iges,minres                 [DEFAULT, value] : DEFAULT
  export,iges,mode               [TRIM, BOUND, BREP] : BOUND
  export,iges,pcurves                 [ON, OFF, ONLY] : OFF
  export,iges,pcurvefit           [LINEAR, CUBIC, BEST] : CUBIC
  export,iges,form                  [OFF, ANA, SOR] : OFF
  export,iges,shapeTrim                     [OFF, ON] : OFF
  export,iges,defColour    [OFF, K, B, G, C, R, M, Y, W] : OFF
  export,iges,dirLabel                      [OFF, ON] : OFF
  ######################### END #########################
```

Appendix 3: Example of a CADfix Batch Log File

```
#########################################################################
##
## Record of batch job started at 12:27 23 Feb 2018
##
## Batch files:
##    c:/data/cube.igs
##
#########################################################################


Error reading configuration file:
  c:/data/test.cwc

Importing model:
  c:/data/cube.igs

Processing model:
  c:/Data/cube.fbm

Repairing...

Preparing...

Exporting geometry (IGESforIDEAS12)...

Geometry exported to file:
  c:/Data/cube_cf.igs

Log file written to:
  c:/Data/cube_wizard.log

Model processing complete

Total elapsed time: 00:00:05

-------------


###################################################################
##
## Batch job completed at 12:27
##
###################################################################
```

Appendix 4: Example of CADfix IGES Import Log File

```
THE FILE CONTAINS  IGES  DATA
                                    ---------------------------

 * Message *  SET  - Y1 created
--------------------------------------------- THE START SECTION : ---

 ----------------------------------------------------------------------

NUMBER OF DIRECTORY ENTRIES READ :    34
NUMBER OF PARAMETER DATA RECORDS PROCESSED :   210
Resolving     27 forward pointer references
DELETING TEMPORARY GEOMETRY
IGES FILE TRANSLATION COMPLETE

REPAIRING IGES MODEL
Completing loops, gaps: 8.7E-03, on shape: 1.7E-01
 * Message *  Merged gap (0.0000000) in V1 between: Q1 and Q4
 * Message *  Merged gap (0.0000000) in V1 between: Q3 and Q6
 * Message *  Merged gap (0.0000000) in V1 between: Q5 and Q8
 * Message *  Merged gap (0.0000000) in V1 between: Q7 and Q2
 * Message *  Merged gap (0.0000000) in V2 between: Q5 and Q9
 * Message *  Merged gap (0.0000000) in V2 between: Q10 and Q12
 * Message *  Merged gap (0.0000000) in V2 between: Q11 and Q14
 * Message *  Merged gap (0.0000000) in V2 between: Q13 and Q7
 * Message *  Merged gap (0.0000000) in V3 between: Q3 and Q15
 * Message *  Merged gap (0.0000000) in V3 between: Q16 and Q18
 * Message *  Merged gap (0.0000000) in V3 between: Q17 and Q10
 * Message *  Merged gap (0.0000000) in V4 between: Q1 and Q19
 * Message *  Merged gap (0.0000000) in V4 between: Q20 and Q22
 * Message *  Merged gap (0.0000000) in V4 between: Q21 and Q16
 * Message *  Merged gap (0.0000000) in V5 between: Q11 and Q24
 * Message *  Merged gap (0.0000000) in V5 between: Q23 and Q20
Healing NURBS lines
Orienting surfaces
...oriented surface V1
...oriented surface V2
...oriented surface V3
...oriented surface V4
...oriented surface V5
...oriented surface V6
REPAIRING COMPLETE

  Filename>
15 points packed
5 general property assignments packed

 entities in this model:
 P  L  S  B SE SH  T CS  Z TX SQ SB CL OB XA
 8 12  7  1  1  6  0  0  0  1  0  0  0  0 27
```

Appendix 5: Example of Individual Model Wizard Log File

```
#########################################################################
##
## Wizard summary log file for model:
##      c:/Data/cube.fbm
##
## Requested operations:
##      Repair :     YES
##      Transform :  NO
##      Prepare :    YES                 (Target system - IGESforIDEAS12)
##      Export :     YES                 (Target system - IGESforIDEAS12)
##      PDQ check :  NO
##                   Version 12 (nnnnn/mmmmm) UBID: cadfix1200fcs.xyzw
#########################################################################


Importing model from IGES (0.023Mb)...

Deleting construction geometry

Validating face definitions

Orienting faces

Writing model database...

Import log written to file:
  c:/Data/cube_IGESimport.log

Opening model (0.055Mb)...

Assessing model status... DONE

Unknown model units. Units assumed to be Millimeters.

Elapsed time: 00:00:03

TOPOLOGICAL REPAIR

Re-assessing model size

Splitting closed NURBS surfaces
new surfaces created: 0

Checking edge connectivity
poorly connected edges found: 0
degenerate edges found: 0

Checking for unembedded faces
unembedded faces found: 0

Checking face integrity

    Healing sloppy edges

    Normalising NURBS surfaces

    Splitting closed NURBS surfaces
    new surfaces created: 0
    surfaces that failed: 0

    Fixing partially collapsed faces
    partially collapsed faces fixed: 0   (unfixed: 0)

    Fixing unclosed loops

    Orienting faces

    Checking for non-finite-area faces
    non-finite-area faces found: 0

Completed face integrity checks

Checking integrity of existing solid bodies

    Checking for voids
```

```
        voids removed: 0

        Orienting existing solid bodies

    Building new solid bodies
        Failed to build any new solid bodies

    Removing all unused parts

    Model successfully repaired

    TOPOLOGICAL REPAIR COMPLETED

    Elapsed time: 00:00:01

    PREPARE FOR EXPORT (IGESforIDEAS12)

    Fixing closed NURBS surfaces

    Trimming back NURBS surfaces

    Attempting to simplify poorly defined NURBS surfaces

    Fixing discontinuous NURBS surfaces

    Fixing degenerate NURBS surfaces

    Healing sloppy faces

    Checking model integrity

    Orienting faces

    Checking for failures...  DONE

    Model successfully prepared

    PREPARE FOR EXPORT COMPLETED

    Elapsed time: 00:00:01

    EXPORT (IGESforIDEAS12)

    Exporting geometry...

    Geometry exported to file:
      c:/Data/cube_cf.igs

    EXPORT COMPLETED

    Elapsed time: 00:00:00

    ########################### REPORT ###########################

    MODEL      c:/Data/cube.fbm
    STATUS     SUCCESS
    IMPORT     SUCCESS - Model imported from c:/data/cube.igs
    REPAIR     SUCCESS - Model successfully repaired
    TRANSFORM   NONE - Not requested
    PREPARE    SUCCESS - Model successfully prepared
    EXPORT     SUCCESS - Geometry exported to file: c:/Data/cube_cf.igs
    PDQ         NONE - Not requested

    ############################ END ############################
```