

Running CADfix in Batch Mode

CADfix 11

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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 11\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 11\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 11  
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 <cw c_file>	Identifies the CADfix Wizard control (.cwc) file that should be used to control the automatic model processing options and run parameters. The <cw c_file> argument is the name of the cwc file, for example w izard.cw c. 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<cw i_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.
-m odel s <files .txt>	<p>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 m odel s.dat</p> <p>The contents of files .txt should be a list of CAD file names, for example:</p> <pre> test.igs lid.igs casting.sat bracket.stp housing.igs </pre> <p>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.</p>
-m erge <filenam e>	This option prepares CADfix for the merging of batch files into a single database on import. The <filenam e> 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 <filenam e>	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.
-w orkingdir <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.
-dam em ory <num ber>	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.

<code>-noresource db</code>	Place at the end of the command line to ensure that the .CADfixNNN user configuration file found in the HOME directory is not used
<code>-xml report</code>	<p>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:</p> <ol style="list-style-type: none"> a. <code>'-xmlreport 1'</code> (report will be written to the 'launch' folder, name "cadfixResults.xml") b. <code>'-xmlreport <path>'</code> (report will be written to the named file [if this file already exists then it will be backed up])
<code>-in portsystem <name></code>	Where <name> is the name of the import system e.g. Step. The command line switch will be followed by a folder path.
<code>-regressiontest1</code>	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 CWC¹ 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, codePage	Code page number for non-unicode characters
34	import, assemblies, mode	choose the mode used to process an assembly: the complete assembly, each component in turn or selected components of the assembly
35	import, assemblies, subpartid	where the option is chosen select the identity of the assembly subpart to be processed

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

36	<code>import, assemblies, explode</code>	explode the assembly on import
37	<code>import, assemblies, emptyComponents</code>	import empty components of the assembly or ignore them
38	<code>import, assemblies, freeLines</code>	to control the import of free lines in an assembly
39	<code>import, assemblies, freeSurfs</code>	to control the import of free surfaces in an assembly
40	<code>import, assemblies, freeFaces</code>	to control the import of free faces in an assembly
41	<code>import, assemblies, freePartsName</code>	Sets the naming of free parts
42	<code>import, autoGroup, colour, apply</code>	instruct CADfix to create sets of geometry entities based on colour attributes during import
43	<code>import, autoGroup, colour, type</code>	specify the geometry entities to be collected into sets
44	<code>import, autoGroup, colour</code>	Single option to incorporate 32 and 33
45	<code>import, autoGroup, layer, apply</code>	instruct CADfix to create sets of geometry entities based on layer attributes during import
46	<code>import, autoGroup, layer, type</code>	specify the geometry entities to be collected into sets
47	<code>import, autoGroup, layer, values</code>	specify the layer numbers to be used for collecting geometry into sets
48	<code>import, autoGroup, layer</code>	Single option to incorporate 35, 36 and 37
49	<code>import, autoGroup, label</code>	Specify a part type, a label and the set name for processing during the transformation stage
50	<code>import, temporaryFiles</code>	leave any temporary files used during the wizard process
51	<code>import, resolveDegenerateLines</code>	Process degenerate lines during import
52	<code>import, recentreModel</code>	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.
53	<code>import, convertUnits</code>	convert the units on import
54	<code>import, scaleModel</code>	scale the model on import
55	<code>Import, calcMassProps</code>	Calculate mass properties after import operation
56	<code>import, iges, useDefaults</code>	If you have changed the import and repair options and wish to reset to the defaults
57	<code>import, iges, definitionEntities</code>	all entities flagged as "definition" entities are to be imported
58	<code>import, iges, blankedEntities</code>	all entities that are flagged as being blanked (e.g. hidden construction data) are to be imported
59	<code>import, iges, freeLines</code>	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.
60	<code>import, iges, freeLinesFont</code>	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.
61	<code>import, iges, constructionGeom</code>	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.
62	<code>import, iges, linesAsNURBS</code>	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.

63	<code>import,iges,splitCreasedNURBS</code>	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
64	<code>import,iges,turningAngle</code>	minimum turn angle for the splitting of discontinuous edges is normally set to 20 degrees
65	<code>import,iges,splitClosedNURBS</code>	Check Split closed NURBS surfaces to split any closed NURBS surfaces that exist in the model.
66	<code>import,iges,pcurves</code>	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.
67	<code>import,iges,calcGoodTol</code>	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.
68	<code>import,iges,mergeDuplicates</code>	to automatically merge duplicate points and edges at the model processing tolerance.
69	<code>import,iges,boundUntrimmed</code>	to create new edges around the boundary of any surfaces that are not currently trimmed
70	<code>import,iges,completeBoundaries</code>	to complete any face boundaries that contain gaps
71	<code>import,iges,stretchFaces</code>	to stretch the face boundaries together
72	<code>import,iges,buildBodies</code>	to automatically make solids from the model topology
73	<code>import,iges,extraAttributes</code>	Will assign to each CADfix entity created other attributes from the IGES file such as DE number, Form flag and IGES Entity type.
74	<code>import,step,freeLines</code>	Import free lines/points if any entities that are not connected to a face, such as centre lines, are to be imported.
75	<code>import,step,freeSurfs</code>	free surfaces if any unused or free surfaces are to be imported
76	<code>import,step,freeFaces</code>	Import free faces if any unused faces are to be imported.
77	<code>import,step,splitDiscontinuous</code>	The STEP file may contain NURBS edges that need to be split on import. Use to split any G1 discontinuous edges.
78	<code>import,step,splitAngle</code>	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the splitAngle . This angle defaults to 20 degrees.
79	<code>import,step,importPFID</code>	Import the STEP PFID attribute
80	<code>import,step,importPMI</code>	Switch on the imposing of Product Manufacturing Information (PMI) from the STEP file. For viewing only
81	<code>import,vdafs,freeLines</code>	Import free lines if any entities that are not connected to a face, such as centrelines, are to be imported.
82	<code>import,vdafs,freeSurfs</code>	Import free surfaces if any unused or free surfaces are to be imported
83	<code>import,vdafs,freeFaces</code>	Import free faces if any unused faces are to be imported
84	<code>import,vdafs,splitDiscontinuous</code>	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

85	<code>import, vdaFs, splitAngle</code>	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
86	<code>import, stl, mergeVertices</code>	By default, all duplicate vertices will be merged as the model is imported. Switch OFF if duplicate vertices should not be merged.
87	<code>import, stl, facetedSurfaces</code>	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 faceted representation subsequently held in CADfix can be roughly converted to geometry using an experimental interactive tool. The "Faceted 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.
88	<code>import, stl, traceShells</code>	To trace shells in the model
89	<code>import, stl, reorient</code>	Option to re-orient all imported STL facets to be consistent, where possible
90	<code>import, 3ds, facetedSurfaces</code>	For quick viewing of the 3DS model this converts facets to a CADfix entity type that appears and handles like a CAD surface.
91	<code>import, acis, freeLines</code>	Import free lines if any entities that are not connected to a face, such as centrelines, are to be imported
92	<code>import, acis, freeSurfs</code>	Import free surfaces if any unused or free surfaces are to be imported.
93	<code>import, acis, freeFaces</code>	free faces if any unused faces are to be imported
94	<code>import, acis, splitDiscontinuous</code>	The ACIS file may contain NURBS edges that need to be split on import. Use to split any G1 discontinuous edges.
95	<code>import, acis, splitAngle</code>	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the splitAngle . This angle defaults to 20 degrees.
96	<code>import, acis, labelAttributeName</code>	Name of CADfix XASG to be used to define the ACIS entity label value, default is LABEL
97	<code>import, acis, layerAttributeName</code>	Name of CADfix XASG to be used to define the ACIS entity layer value, default is LAYE R
98	<code>import, acis, splitPeriodics</code>	Option to split periodic surfaces
99	<code>import, dxf, freeLines</code>	Import free lines if any entities that are not connected to a face, such as centrelines, are to be imported.
100	<code>import, dxf, freeSurfs</code>	Import free surfaces if any unused or free surfaces are to be imported.
101	<code>import, dxf, freeFaces</code>	free faces if any unused faces are to be imported
102	<code>import, dxf, importBlanked</code>	Import or ignore blanked entities
103	<code>import, dxf, importProxy</code>	Option to control import of DXF proxy entities
104	<code>import, dxf, facetsToFaces</code>	Option to import DXF facets as BRep faces
105	<code>import, dxf, splitDiscontinuous</code>	The DXF/DWG file may contain NURBS edges that need to be split on import. Use to split any G1 discontinuous edges.
106	<code>import, dxf, splitAngle</code>	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the splitAngle . This angle defaults to 20 degrees.
107	<code>import, dxf, splitPeriodics</code>	Option to split periodic surfaces
108	<code>import, parasolid, freeLines</code>	Import free lines if any entities that are not connected to a face, such as centrelines, are to be imported
109	<code>import, parasolid, freeSurfs</code>	Import free surfaces if any unused or free surfaces are to be imported.
110	<code>import, parasolid, freeFaces</code>	Import free faces if any unused faces are to be imported.
111	<code>import, parasolid, units</code>	The Parasolid unit of length is meters. 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.
112	<code>import, parasolid, splitClosedNURBS</code>	to control the splitting of closed NURBS during the import of a Parasolid file

113	<code>import,parasolid,Seamless</code>	to control the splitting of seamless faces during the import of a Parasolid file
114	<code>import,parasolid,splitDiscontinuous</code>	The Parasolid file may contain NURBS edges that need to be split on import. Use to split any G1 discontinuous edges.
115	<code>import,parasolid,splitAngle</code>	G1 discontinuous NURBS edges will only be split where the subtended angle at the discontinuity exceeds the splitAngle . This angle defaults to 20 degrees.
116	<code>import,parasolid,explodeAssembly</code>	Parasolid assemblies can be exploded on import
117	<code>import,cadds,freeLines</code>	Import free lines if any entities that are not connected to a face, such as centrelines, are to be imported.
118	<code>import,cadds,freeSurfs</code>	Import free surfaces if any unused or free surfaces are to be imported.
119	<code>import,cadds,importBlanked</code>	Import or ignore blanked entities
120	<code>import,cadds,pointTol</code>	The CADDs file may contain coincident points and to distinguish these from points to be merged a Point coincidence tolerance may be applied.
121	<code>import,cadds,nameByPart</code>	assembly components labelled by part name
122	<code>import,catia,importNoShows</code>	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.
123	<code>import,catia,importNoPicks</code>	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.
124	<code>import,catia,importOnlyRoot</code>	use this option if problems are encountered when importing models from a particular supplier.
125	<code>import,catia,importOnlyMasterWorkspace</code>	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.
126	<code>import,catia,dittosAsAssembly</code>	Import CATIA DITTO assemblies. This requires a special version of the translator based on Spatial' s InterOp for which there is a royalty payment
127	<code>import,catia,solidmAsFacets</code>	Process CATIA mock-up solids or SOLIDM parts as facets
128	<code>import,catia,importPipes</code>	Import CATIA V4 piping entities (*PIP)
129	<code>import,catia,layerFilter</code>	option controls whether to use the user-specified layer filter while translating CATIA V4 file
130	<code>import,catia,layerLayerFilter</code>	Import the CATIA V4 layer filter attribute
131	<code>import,catia,useAnalyticSurfs</code>	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.
132	<code>import,catia,3DCurveRegen</code>	option controls the regeneration of 3D curves
133	<code>import,catia,3DCurvePref</code>	option sets the preference to be used while generating trimmed curves for a surface
134	<code>import,catia,freeLines</code>	Import free lines if any entities that are not connected to a face, such as centrelines, are to be imported.
135	<code>import,catia,freeSurfs</code>	Import free surfaces if any unused or free surfaces are to be imported.
136	<code>import,catia,freeFaces</code>	Import free faces if any unused faces are to be imported.
137	<code>import,catia,splitDiscontinuous</code>	The CATIA file may contain NURBS edges that need to

		be split on import. Check Use to split any G1 discontinuous edges.
138	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.
139	import, catia, importer	Refers to different CATIA translator options: GSSL, INTEROP_GENERIC and INTEROP_ACIS. Some of these require extra royalty payments.
140	import, catia5, importHidden	Import hidden entities if entities tagged as hidden are to be imported.
141	import, catia5, importCGR	Import of CATIA V5 .cgr facet data
142	import, catia5, freeLines	Import free lines if any entities that are not connected to a face, such as centrelines, are to be imported.
143	import, catia5, freeSurfs	Import free surfaces if any unused or free surfaces are to be imported.
144	import, catia5, freeFaces	Import free faces if any unused faces are to be imported.
145	import, catia5, importer	Refers to different CATIA translator options: INTEROP_GENERIC and INTEROP_ACIS. These require extra royalty payments.
146	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.
147	import, catia5, units	Specify the CATIA V5 units
148	import, ug, importHidden	Import hidden entities if entities tagged as hidden are to be imported.
149	import, ug, importNonVisibleLayers	Import non-visible layers if layer entities tagged as non-visible are to be imported.
150	import, ug, freeLines	Import free lines if any entities that are not connected to a face, such as centrelines, are to be imported.
151	import, ug, freeSurfs	Import free surfaces if any unused or free surfaces are to be imported.
152	import, ug, freeFaces	Import free faces if any unused faces are to be imported.
153	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.
154	import, ug, importer	Refers to different CATIA translator options: INTEROP_GENERIC and INTEROP_ACIS. These require extra royalty payments.
155	import, ug, splitClosedNURBS	to control whether closed NURBS from a NX file are split during import
156	import, ug, splitSeamless	To control whether seamless faces from a NX file are split during import
157	import, ug, assemblyPathPref	Assembly path preference
158	import, inventor, freeLines	Import free lines if any entities that are not connected to a face, such as centrelines, are to be imported.
159	import, inventor, freeSurfs	Import free surfaces if any unused or free surfaces are to be imported.
160	import, inventor, freeFaces	Import free faces if any unused faces are to be imported.
161	import, inventor, assemblyPathPref	Select assembly path preference
162	import, inventor, importUnfolded	Import unfolded bodies
163	import, inventor, importer	Fix/stitch polysHELLS
164	import, sw, freeLines	Import free lines if any entities that are not connected to a face, such as centrelines, are to be imported.
165	import, sw, freeSurfs	Import free surfaces if any unused or free surfaces are to be imported.
166	import, sw, freeFaces	Import free faces if any unused faces are to be imported.
167	import, sw, importHidden	Import hidden instances if any hidden components are to be imported
168	import, sw, importSuppressed	Import suppressed instances if any suppressed components are to be imported
169	import, sw, splitClosedNURBS	Splitting of closed NURBS during import of SolidWorks data

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

206	<code>import,prc,freeLines</code>	Import of Hoops PRC free lines
207	<code>import,prc,freeSurfs</code>	Import of Hoops PRC free surfaces
208	<code>import,prc,freeFaces</code>	Import of Hoops PRC free faces
209	<code>import,prc,onlyFacets</code>	Import of only Hoops PRC facet data
210	<code>import,pdf,freeLines</code>	Import of 3D PDF free lines
211	<code>import,pdf,freeSurfs</code>	Import of 3D PDF free surfaces
212	<code>import,pdf,freeFaces</code>	Import of 3D PDF free faces
213	<code>import,pdf,onlyFacets</code>	Import of only 3D PDF facet data
214	<code>import,flite,kntred</code>	Option to enable the removal of knots from Flite NURBS
215	<code>import,flite,kntredtol</code>	Tolerance used to reduce the complexity of Flite NURBS surfaces by removing knots.
216	<code>Import,solar,kntred</code>	Option to enable the removal of knots from solar NURBS
217	<code>import,solar,kntredtol</code>	Tolerance used to reduce the complexity of solar NURBS surfaces by removing knots.
218	<code>repair</code>	Switch the Wizard Repair stage ON or OFF
219	<code>repair,maxTol</code>	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.
220	<code>repair,buildOption</code>	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.
221	<code>repair,boundUntrimmed</code>	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.
222	<code>repair,wireframe</code>	Make allowance for wireframe within the imported geometry and repair
223	<code>repair,approximateSurfaces</code>	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.
224	<code>repair,allowVoids</code>	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.
225	repair, voids	The user has the option to Delete , Remove or Keep any voids found in the solid model during the repair process
226	repair, deleteWireframe	Delete redundant wireframe geometry
227	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
228	repair, preserveKernels	Preserve kernels in cellular bodies with shared topology
229	repair, deleteUnusedFaces	Under some circumstances you may wish to Delete unused faces during the repair process that may only require bodies.
230	repair, calcMassProps	Calculate the mass properties after the Repair process is complete
231	repair, customCommand	TCL script to be executed instead of the standard Wizard Repair
232	transform	Switch the Wizard Transform stage ON or OFF
233	transform, joinBodies, apply	to automatically search and replace groups of bodies with a single body
234	transform, joinBodies, tol	Tolerance used for touching bodies
235	transform, joinBodies, mode	Create a manifold or non-manifold bodies
236	transform, joinBodies, allowVoids	Allow voids in the final joined body
237	transform, joinFragmented, apply	Join faces with fragmented surfaces
238	transform, joinFragmented, tol	Tolerance used for joining faces with fragmented surfaces
239	transform, joinFaces, apply	to automatically search and replace groups of faces with a single face.
240	transform, joinFaces, type	Choose to join all faces or just slivers to their neighbours
241	transform, joinFaces, width	Set the maximum width of the sliver
242	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.
243	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.
244	transform, joinFaces, splits	Allow narrow face splits
245	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 Geometry
246	transform, joinFaces, faceangle	Set maximum face angle deviation
247	transform, joinFaces, edgeangle	Set maximum edge angle deviation
248	transform, joinFaces, angle	The tangency angle may be used to control how faces are collected into groups
249	transform, joinFaces, filtermultiloop	Exclude multi-loop faces from the face joining
250	transform, joinFaces, filterlarge	Exclude relatively large faces from the face joining
251	transform, joinFaces, filtermultisided	Exclude multi-sided faces from the face joining
252	transform, joinFaces, rejectthreesided	Reject from the face joining three-sided faces
253	transform, joinFaces, rejectmultisided	Reject from the face joining multi-sided faces or those greater than four-sided
254	transform, joinFaces, matchlayers	Make use of matching layer in forming groups
255	transform, joinFaces, matchcolours	Make use of matching colours when joining groups
256	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.
257	transform, joinFaces, change	the Maximum change in displacement between the original surfaces and the final surface may be used as the acceptance criteria.
258	transform, joinEdges, apply	to automatically search for chains of edges and replace them with a single NURB edge
259	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
260	transform, collapse, apply	to automatically collapse or remove any parts that are less than the Minimum feature size – see below

261	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.
262	transform,collapseOption	Collapse FACES or EDGES
263	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.
264	transform,pinch,apply	Check for and pinch sharp edge or corner angles in the model
265	transform,pinch,angle	Set the angle for the sharp edge angle test
266	transform,pinch,tol	Set the bridging length at which to pinch
267	transform,narrow,apply	Check for narrow regions
268	transform,narrow,width	Minimum width defining a narrow region can be given as an embedded tolerance or a user supplied value
269	transform,narrow,type	The type of narrow region can be Necks, Tapers or Both
270	transform,narrow,fixaction	Split or Pinch are the type of fix choices
271	transform,holes,removeHoles	to automatically remove any holes whose radius is smaller than the Max. diameter – see below
272	transform,holes,plugHoles	Create bodies from holes
273	transform,holes,drillTips	Look for conical drill tip features in the model and remove them
274	transform,holes,centrelines	Generate centre lines for holes
275	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.
276	transform,holes,blind	Blind holes remove only those holes that pass part way through the model.
277	transform,holes,through	Through holes remove only those holes that pass through the model
278	transform,holes,complex	Complex holes (three or more openings) remove holes that have three or more openings.
279	transform,holes,searchMode	The Cross-section mode
280	transform,holes,check2dSize	Check exit size parameter is used to check the 2D bounding box (or diameter) of the edges defining the hole exit.
281	transform,holes,check3dSize	Check feature size defines the 3D bounding box of the whole hole.
282	transform,holes,checkSize	this is a No size limit option to look for all the selected hole types in the model regardless of size.
283	transform,holes,size2d	The actual check exit size value
284	transform,holes,size3d	The actual check feature size value
285	transform,holes,size	setting the Maximum Diameter specifies the diameter above which holes will not be found.
286	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.
287	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.
288	transform,holes,delete	when creating bodies from holes delete the original body
289	transform,protrusions,remove	to automatically remove any protrusions whose radius is smaller than the Max. diameter – see below
290	transform,protrusions,check2dSize	Check exit size parameter is used to check the 2D

		bounding box (or diameter) of the edges defining the protrusion.
291	transform,protrusions,check3dSize	Check feature size defines the 3D bounding box of the whole protrusion.
292	transform,protrusions,size2d	The actual check exit size value
293	transform,protrusions,size3d	The actual check feature size value
294	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.
295	transform,protrusions,checkSize	this is a No size limit option to look for all the selected protrusions types in the model regardless of size.
296	transform,protrusions,size	setting the Maximum Diameter specifies the diameter above which protrusions will not be found.
297	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.
298	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.
299	transform,removeSeams,tol	tolerance used for evaluating the seamless face
300	transform,removeSeams,pln	applies to planes
301	transform,removeSeams,cyl	applies to cylinders
302	transform,removeSeams,con	applies to cones
303	transform,removeSeams,sph	applies to spheres
304	transform,removeSeams,tor	applies to toroids
305	transform,deleteFilletts,apply	to automatically remove fillets
306	transform,deleteFilletts,radmax	A maximum radius may be set to define the radius
307	transform,deleteFilletts,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.
308	transform,deleteFilletts,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.
309	transform,deleteFilletts,tol	This value is used to control accuracy of the operation.
310	transform,deleteFilletts,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.
311	transform,deleteFilletts,angtol	Is used to determine when adjacent surfaces are considered parallel (default = 3 degrees).
312	transform,deleteFilletts,joinedges	After removing fillets join edges to remove old vertices
313	transform,deleteFilletts,facemod	<i>Redundant as scars are always left</i>
314	transform,deleteChamfers,apply	To automatically remove chamfers from the model
315	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.
316	transform,deleteChamfers,ang1	The upper corner angle of the chamfer.
317	transform,deleteChamfers,d2	The distance from the dihedral angle corner to down the side of the chamfer.
318	transform,deleteChamfers,ang2	The smallest dihedral angle between chamfer faces.
319	transform,deleteChamfers,tol	This value is used to control accuracy of the operation.
320	transform,deleteChamfers,angtol	Is used to determine when adjacent surfaces are considered parallel (default = 3 degrees).
321	transform,deleteChamfers,joinedges	Chains of edges left after the process will also be removed.
322	transform,deleteChamfers,facemod	++Not used for batch processing++
323	transform,deleteFeatures,apply	to automatically remove features
324	transform,deleteFeatures,tol	tolerance used for evaluating the features
325	transform,deleteFeatures,joinedges	Scan for multiple continuous edges between vertices

		and join into a single edge
326	transform,deleteFeatures, facemod	<i>Redundant as scars are always left</i>
327	transform,deleteFeatures, set	to specify a set name
328	transform,imprintEdges, apply	Set the program to imprint edges that coincide or overlap
329	transform,imprintEdges, tol	Set tolerance for edge imprinting
330	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
331	transform,imprintBodies, tol	Tolerance used for body imprinting
332	transform,deleteBodies, apply	Automatically delete bodies from the model using either a volume or a bounding box dimension
333	transform,deleteBodies, method	Specify whether volume or bounding box is to be used
334	transform,deleteBodies, size	Specify a value for the volume or bounding box diagonal dimension
335	transform,externalFilter, apply	Selection of only external model data activated
336	transform,externalFilter, tol	Set the tolerance for the removal of internal detail
337	transform,externalFilter, maxgap	Maximum gap to be ignored during the internal detail removal
338	transform,externalFilter, mode	Option to switch on aggressive filtering
339	transform,externalFilter, perf	Option controlling performance cost of aggressive filtering, 1 is fast/low accuracy, 10 is fast/accuracy
340	transform,externalFilter, level	Set the level of data removal to faces, bodies or instances
341	transform,wrap, apply	Automatically create a shrink wrap of the model
342	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.
343	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.
344	transform,wrap, facetSize	Specify the typical facet size to be used for shrink wrapping
345	transform,decimate, apply	Decimate the shrink wrap model by making planar areas coarse
346	transform,decimate, tol	Specify the tolerance to which the decimate function will work
347	transform,unfacet, apply	Option to enable/disable the unfacet action
348	transform,unfacet, edge_vertextol	Max expected difference between a vertex of a faceted edge and the original edge the facets were made from
349	transform,unfacet, edge_sagtol	Max expected difference between a centre of a faceted edge and the original curve the facets were made from
350	transform,unfacet, face_vertextol	Max expected difference between a vertex of a faceted face and the original face the facets were made from
351	transform,unfacet, face_sagtol	Max expected difference between centroid of a faceted face and the original face the facets were made from
352	transform,unfacet, face_normtol	Max expected difference between faceted face normal and the original face the facets were made from
353	transform,unfacet, no_unclosed	Option to reject any faceted bodies that are not closed shells after unfaceting
354	transform,deform, apply	Switch on deforming of CAD geometry to match deformed mesh
355	transform,deform, fitMethod	Select a fit method from either position only or position plus derivatives
356	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.
357	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.
358	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.
359	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.
360	transform, deform, delete	Enable this option to automatically delete the original surfaces following an acceptable refit.
361	Transform, calcMassProps	Calculate the mass properties after the transform operations finish
362	transform, customCommand	TCL script to be executed instead of the standard Wizard Transform
363	prepare	Switch the Wizard Prepare stage ON or OFF
364	prepare, system	specifies the prepare target system if different from the export system
365	prepare, debug	Use to print values without performing prepare for export
366	prepare, customCommand	TCL script to be executed instead of the standard Wizard Prepare
367	'Overlapping curves', R-CU-EM:{{apply value}}{tol value}}	Check for whether a curve lies wholly within or partially overlaps another curve
368	'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.
369	'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
370	'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
371	'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.
372	'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.
373	'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
374	'Non-NURBS edges', R-ED-AN:{{apply value}}	Check edges to convert any non-NURBS edges to NURBS entities
375	'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.
376	'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.
377	'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
378	'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

379	'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.
380	'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.
381	'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.
382	'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.
383	'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.
384	'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.
385	'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
386	'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 .
387	'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.
388	'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
389	'Curve knot hyper-multiplicities',R-CU-HMK:{{apply value}}{condition	NURBS curve knot hyper-multiplicity adjustment

	<code>value}{rate value}{tol value}}</code>	
390	<code>'Surface knot hyper-multiplicities',R-SU-HMK:{{apply value}{condition value}{rate value}{tol value}}</code>	NURBS surface knot hyper-multiplicity adjustment
391	<code>'Folded curves',R-CU-FO:{{apply value}{tol value}}</code>	to find and fix any NURBS edges where the angle between the normal at each successive edge segment changes by more than 90 degrees.
392	<code>'Oversize surfaces',R-FA-TRM:{{apply value}{percent value}}</code>	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.
393	<code>'Closed surfaces',R-SU-CL:{{apply value}{tol value}}</code>	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.
394	<code>'Degenerate surfaces',R-SU-DC:{{apply value} {tol value} {simplify value} {split value} {flare value} {refit value} {trimback value:mm}}</code>	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.
395	<code>'Degenerate surface corners',R-SU-DP :{{apply 1} {angmin 5.0} {angtype FLAT}}</code>	A corner of a NURBS surface forms a tangential (~180) or very sharp (~0) angle.
396	<code>'Large edge-surface gap',R-FA-EG:{{apply value}{tol value}{lock value}}</code>	The gap between the edge of a face and the underlying surface known also as sloppiness is adjusted to be within the value given
397	<code>'Smooth surfaces',R-SU-SMO:{{apply value}{maxChange value}}</code>	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.
398	<code>'Undersize surfaces',R-FA-XTN:{{apply value}{undersize value}}</code>	to ensure that any surface whose area is smaller than the dependant face is extended to match the size of that face.
399	<code>'Seamless faces',R-FA-SEA:{{apply value}}</code>	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.
400	<code>'Missing surfaces',R-FA-EMD:{{apply value}{tol value}}</code>	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.
401	<code>'Sharp edge angle',R-LO-SA:{{apply value}{angle value}}</code>	removes sliver faces where the join between two adjacent edges in a loop forms a very sharp angle
402	<code>'Sharp corner angle',R-LO-SA : {{apply 0} {angle 5.0} {tol DEFAULT} {filletsonly 0} {fixaction MERGE}}</code>	The joint between two adjacent edges in a loop forms a sharp angle.
403	<code>'Zero-area faces,R-FA-ZAR:{{apply value}{tol value}}</code>	Check for zero area faces
404	<code>'Refit faces,R-FA-RPL:{{apply value}{tol value}}</code>	Apply the face refitting algorithm to create new embedded surfaces
405	<code>'Degenerate tangent,R-CU-G1D:{{apply value}{tol value}}</code>	Check for degenerate tangents
406	<code>'Join short edges,R-ED-JOI:{{apply value}{tol value}}</code>	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.
407	<code>'Sloppy surface singularity,R-FA-VFS:{{apply value}{tol value}}</code>	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 dependant face with vertices within a given threshold of the singularity, the shape will be translated and rotated to heal the singularities to the vertices

408	'Shared topology',R-MO-POP: {{apply 0}}	Where two or more bodies are connected by shared faces or edges or vertex they are separated
409	export	Switch the Wizard Export stage ON or OFF
410	export,system	specify the export CAD system to be used
411	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 <code>_cf</code> will be used.
412	export,fileExtension	The file name extension will be set here
413	export,always	CADfix will always continue to the end, even if errors are encountered during Repair or Prepare stage
414	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.
415	export,assemblies,explode	Explode any assemblies on export
416	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.
417	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
418	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
419	export,sendUserAttrs	if user attribute information is available such as material properties these can be exported as assignments on the geometry
420	export,customCommand	TCL script to be executed instead of the standard Wizard Export
421	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 .
422	export,exportSet	Allows the user to specify the export set as different from the working set
423	export,textNoteEnabled	CATIA specific option
424	export,textNoteChars	Text Note characters
425	export,textNotePosX	Text Note X position
426	export,textNotePosY	Text Note Y position
427	export,textNotePosZ	Text Note Z position
428	export,textNoteSize	Text Note size
429	export,textNoteColour	Text Note colour
430	export,textNoteLayer	Text Note layer
431	export,textNotePlane	Text Note plane (e.g. yz)
432	export,textNoteAngle	Text Note angle
433	export,checkEmpty	will not write the output file if you have an empty model
434	export,calcMassProps	Calculate the mass properties at the end of the batch process
435	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.
436	<code>export,iges,mode</code>	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.
437	<code>export,iges,pcurves</code>	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.
438	<code>export,iges,pcurvefit</code>	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.
439	<code>export,iges,form</code>	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".
440	<code>export,iges,shapeTrim</code>	Use if NURBS surface entities are to be trimmed to fit the size of the dependant face.
441	<code>export,iges,defColour</code>	Specify the default entity colour from a range given
442	<code>export,iges,dirLabel</code>	Use the export label in the 8-character directory section "label" field
443	<code>export,iges,perBodyLayers</code>	Put each body on own layer
444	<code>export,step,lengthTol</code>	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.
445	<code>export,step,protocol</code>	Select the appropriate STEP protocol for the downstream system.
446	<code>export,step,merge</code>	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.
447	<code>export,step,singlepart</code>	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.
448	<code>export,step,nonmanshells</code>	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.
449	<code>export, step, polyshells</code>	An option for the stitching of polyshells during import
450	<code>export, step, description</code>	To populate the user definable fields in the STEP spec
451	<code>export, step, name</code>	To populate the user definable fields in the STEP spec
452	<code>export, step, author</code>	To populate the user definable fields in the STEP spec
453	<code>export, step, organisation</code>	To populate the user definable fields in the STEP spec
454	<code>export, step, authorisation</code>	To populate the user definable fields in the STEP spec
455	<code>export, step, codePage</code>	Code page number for non-Unicode characters
456	<code>export, stl, method</code>	Two methods can be used to control the quality of the faceted 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.
457	<code>export, stl, facetSag</code>	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.
458	<code>export, stl, facetTurn</code>	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
459	<code>export, stl, facetLength</code>	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 1/100 th times the overall model diameter.
460	<code>export, stl, expandRate</code>	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.
461	<code>export, stl, minFacets</code>	This parameter specifies the minimum number of faces to be created on each edge.
462	<code>export, stl, minThrough</code>	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.
463	<code>export, stl, smooth</code>	The triangular faceting for quality STL will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
464	<code>export, stl, useSources</code>	Make use of on any FLITE3D sources when generating the STL faceted model
465	<code>export, stl, openShells</code>	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.
466	<code>export, stl, multiSolids</code>	multiple solid (or body) models can be written into an STL file as separately grouped entities
467	<code>export, stl, perBodyMultiSolids</code>	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.
468	<code>export, stl, multiFiles</code>	Multiple STL files are generated on export based on a number of criteria from separate bodies to colours to labels.
469	<code>export, stl, sharedFaces</code>	Used when exporting multiple bodies. Rather than creating duplicate sets of facets on shared faces just a single shared set of facets are used.

470	<code>export, stl, zones</code>	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.
471	<code>export, stl, explode</code>	Explode STL of an assembly model into separate files
472	<code>export, stl, wraps</code>	Export STL result of a shrink wrap
473	<code>export, stl, format</code>	Select either ASCII or Binary STL export file type
474	<code>export, 3ds, method</code>	Two methods can be used to control the quality of the faceted model. The method used to generate the 3DS file can produce markedly different results. Choose CURVATURE to generate a 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.
475	<code>export, 3ds, facetSag</code>	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.
476	<code>export, 3ds, facetTurn</code>	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
477	<code>export, 3ds, facetLength</code>	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 1/100 th times the overall model diameter.
478	<code>export, 3ds, expandRate</code>	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.
479	<code>export, 3ds, minFacets</code>	This parameter specifies the minimum number of faces to be created on each edge.
480	<code>export, 3ds, minThrough</code>	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.
481	<code>export, 3ds, smooth</code>	The triangular faceting for quality 3DS will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
482	<code>export, 3ds, openShells</code>	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.
483	<code>export, 3ds, zones</code>	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.
484	<code>export, 3ds, wraps</code>	Export 3DS result of a shrink wrap
485	<code>export, acis, resabs</code>	This parameter indicates the modelling tolerance used by the target system and defaults to 1.0E-06.
486	<code>export, acis, tolerant</code>	To define every geometric entity in the ACIS file with reference to a tolerance.
487	<code>export, acis, bodyChecker</code>	To check the validity of the ACIS file.
488	<code>export, acis, checkerLevel</code>	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.
489	<code>export, acis, version</code>	specify the export ACIS version number
490	<code>export, acis, nonmanshells</code>	Non manifold shells can be exported to ACIS
491	<code>export, TecnomatixMLP, resabs</code>	This parameter indicates the modelling tolerance used

		by the target system and defaults to 1.0E-06.
492	export,TecnomatixMLP,tolerant	To define every geometric entity in the TechnomatixMPL file with reference to a tolerance.
493	export,TecnomatixMLP,bodyChecker	To check the validity of the TechnomatixMPL file.
494	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.
495	export,TecnomatixMLP,centrelines	Create hole centrelines
496	export,parasolid,version	specify the export Parasolid version number
497	export,parasolid,tolerant	to define every edge in the model with reference to a tolerance
498	export,parasolid,bodyChecker	to check the validity of the Parasolid file
499	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.
500	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.
501	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.
502	export,parasolid,maxerr	Choose an appropriate value for Maximum errors to report
503	export,parasolid,sheetBodies	Open shells as individual faces permit the export of mixed models.
504	export,parasolid,multibodyasasm	Create and export an assembly for multiple geometry models
505	export,parasolid,format	There are three output format options; ASCII , a machine dependent BINARY format and a machine independent binary or NEUTRAL format.
506	export,parasolid,colhier	Default is ANY CHILD PARENT
507	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.
508	export,cadds,saveAs	The user can choose to save in the CADDs older 4X or newer 5X format
509	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.
510	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.
511	export,catia,solidmPipes	Option to convert original CATIA V4 pipe bodies in to SOLIDM bodies on export to V4
512	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.
513	export,catia,mirrorKeepOriginal	Option to keep original parts affected by a mirror transform when the transform is removed on export
514	export,catia,mirrorPlane	Option to enable the auto-generation of a mirrored copy of the model during export
515	export,catia,assemblyAsDittos	Option to convert assembly models to CATIA V4 dittos entities
516	export,catia,exportLayerFilters	Option to export layer filter definitions to CATIA V4
517	export,catia,exporter	CATIA export interface
518	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.
519	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
520	export,catia,InterOpAPI	choose Interop option either legacy or connect
521	export,catia5,saveVersion	Option to identify the version of CATIA V5 to write, e.g. CATIA5R24
522	export,catia5,hybrid	Option to create hybrid PartBody entities during V5 export. If disabled, then Geometric Set will be generated.
523	export,catia5,promoteGeomNodes	Promote any (geometry) nodes in the top-level assembly node to be full blown components
524	export,catia5,singleCATProduct	++discontinued++
525	export,catia5,allCGR	Convert all parts to CGR on exporting to V5
526	export,catia5,exporter	Option to set the CATIA V5 export method <deprecated>
527	export,catia5,cgrSag	This parameter specifies the maximum allowable distance between a facet and its underlying surface when generating CGR facets
528	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
529	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.
530	export,catia5,InterOpAPI	choose Interop option either legacy or connect
531	export,catia5,mkshells	map the exported CATIA V5 entities to shells rather than solids
532	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.
533	export,proe,tolerance	tolerance used for the export of Pro/E data
534	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.
535	export,proe,version	specify the export CREO version number
536	export,proe,names	Use original component names
537	export,proe,eachInTurn	++Not used for batch processing++
538	export,proe,asFacets	Export CREO model as facets
539	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.
540	<code>export,dxf,version</code>	specify DXF version number
541	<code>export,dxf,bodyChecker</code>	The ACIS body checker maybe run on the exported data.
542	<code>export,dxf,tolerant</code>	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.
543	<code>export,dxf,explodeFacetted</code>	Explode facetted surfaces
544	<code>export,ansys,btol</code>	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.
545	<code>export,femgv,modelTol</code>	This parameter sets the geometric modelling tolerance to be used in the downstream system
546	<code>export,femgv,workspace</code>	This parameter indicates the default size of the workspace buffer when opening the model in FEMGV
547	<code>export,femgv,lineDIV</code>	Use this parameter to indicate the required number of divisions on each edge in the model
548	<code>export,femgv,history</code>	A log file recording the commands generated during the FEMGV import process can be automatically created by checking this option
549	<code>export,sc03,geomis2d</code>	Option to flag the SC03 export as being 2D
550	<code>export,sc03,tessellate</code>	controls whether the tessellation data is written to an export file
551	<code>export,sc03,TT</code>	Thin and thick subdivided geometry is export
552	<code>export,sc03,ftol</code>	facet tolerance for the tessellation
553	<code>export,sc03,PROX2D</code>	Included in the export is proximity 2D data
554	<code>export,sc03,content</code>	Option to control whether geometry, mesh or both are exported to SC03
555	<code>export,sc03,bodiesOnly</code>	Option to export only solid bodies to SC03
556	<code>export,sc03,edgeTags</code>	edge labels maybe passed to the SC03 model export file
557	<code>export,sc03,imprintEdges</code>	Edge Imprinting may be carried out on the fly during export.
558	<code>export,sc03,imprintEdgesTol</code>	Used to select the imprinting Maximum gap
559	<code>export,sc03,imprintBodies</code>	Body Imprinting may be carried out on the fly during export.
560	<code>export,sc03,imprintBodiesTol</code>	Used to select the imprinting Maximum gap
561	<code>export,sc03,imprintBodiesAng</code>	Used to select the imprinting Maximum draft angle
562	<code>export,sc03,matchPairs</code>	Match existing face pairs during imprinting
563	<code>export,sc03,uniqueNames</code>	Option to ensure all SC03 parts are given unique names
564	<code>export,gridpro,method</code>	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.
565	<code>export,gridpro,facetSag</code>	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.
566	<code>export,gridpro,facetTurn</code>	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
567	<code>export,gridpro,facetLength</code>	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 1/100 th

		times the overall model diameter.
568	<code>export, gridpro, expandRate</code>	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.
569	<code>export, gridpro, minFacets</code>	This parameter specifies the minimum number of faces to be created on each edge.
570	<code>export, gridpro, minThrough</code>	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.
571	<code>export, gridpro, smooth</code>	The triangular faceting for quality will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
572	<code>export, gridpro, useSources</code>	Make use of on any mesh division control sources when generating the faceted model
573	<code>export, gridpro, zones</code>	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.
574	<code>export, dem, method</code>	Two methods can be used to control the quality of the faceted 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.
575	<code>export, dem, facetSag</code>	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.
576	<code>export, dem, facetTurn</code>	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
577	<code>export, dem, facetLength</code>	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 1/100 th times the overall model diameter.
578	<code>export, dem, expandRate</code>	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.
579	<code>export, dem, minFacets</code>	This parameter specifies the minimum number of faces to be created on each edge.
580	<code>export, dem, minThrough</code>	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.
581	<code>export, dem, smooth</code>	The triangular faceting for quality will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
582	<code>export, dem, useSources</code>	Make use of on any mesh division control sources when generating the faceted model
583	<code>export, dem, zones</code>	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.
584	<code>export, xpatch, method</code>	Two methods can be used to control the quality of the faceted 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.
585	<code>export, xpatch, facetSag</code>	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.
586	<code>export, xpatch, facetTurn</code>	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
587	<code>export, xpatch, facetLength</code>	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 1/100 th times the overall model diameter.
588	<code>export, xpatch, expandRate</code>	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.
589	<code>export, xpatch, minFacets</code>	This parameter specifies the minimum number of faces to be created on each edge.
590	<code>export, xpatch, minThrough</code>	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.
591	<code>export, xpatch, smooth</code>	The triangular faceting for quality will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
592	<code>export, xpatch, useSources</code>	Make use of on any mesh division control sources when generating the faceted model
593	<code>export, xpatch, zones</code>	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.
594	<code>export, centaur, tol</code>	The Spline fit tolerance
595	<code>export, centaur, tangType</code>	end tangency type
596	<code>export, centaur, surfEndCond</code>	Controls surface end conditions
597	<code>export, centaur, tangTol</code>	End tangency fitting tolerance in degrees
598	<code>export, centaur, equalSpacedEdges</code>	set the knot spacing to be equal
599	<code>export, centaur, aspectRatio</code>	Maximum patch aspect ratio in degrees
600	<code>export, centaur, growthRatio</code>	When using the Adaptive option for surface splines the surfaces splines are controlled with a Max. growth rate option
601	<code>export, centaur, equalSpacedSurfs</code>	Surface splines can be controlled by setting Equal spaced
602	<code>export, centaur, scalef</code>	The local spline export scale factor tolerance
603	<code>export, centaur, gridU</code>	Number of patches for surfaces in the U direction
604	<code>export, centaur, gridV</code>	Number of patches for surfaces in the V direction
605	<code>export, centaur, gridT</code>	Number of segments for curves T
606	<code>export, centaur, mindiv</code>	Set the minimum division
607	<code>export, centaur, maxdiv</code>	Set the maximum division
608	<code>export, centaur, initgrid</code>	Set the initial grid parameters
609	<code>export, centaur, maxgrid</code>	St the maximum grid size (UxV)
610	<code>export, flite3d, tol</code>	The Spline fit tolerance
611	<code>export, flite3d, tangType</code>	end tangency type

612	<code>export, flite3d, surfEndCond</code>	Controls surface end conditions
613	<code>export, flite3d, tangTol</code>	End tangency fitting tolerance in degrees
614	<code>export, flite3d, equalSpacedEdges</code>	set the knot spacing to be equal
615	<code>export, flite3d, aspectRatio</code>	Maximum patch aspect ratio in degrees
616	<code>export, flite3d, growthRatio</code>	When using the Adaptive option for surface splines the surfaces splines are controlled with a Max. growth rate option
617	<code>export, flite3d, equalSpacedSurfs</code>	Surface splines can be controlled by setting Equal spaced
618	<code>export, flite3d, scalef</code>	The local spline fit parameter tolerance
619	<code>export, flite3d, gridU</code>	Number of patches for surfaces in the U direction
620	<code>export, flite3d, gridV</code>	Number of patches for surfaces in the V direction
621	<code>export, flite3d, gridT</code>	Number of segments for curves T
622	<code>export, flite3d, mindiv</code>	Set the minimum division
623	<code>export, flite3d, maxdiv</code>	Set the maximum division
624	<code>export, flite3d, initgrid</code>	Set the initial value
625	<code>export, flite3d, maxgrid</code>	St the maximum growth value
626	<code>export, solar, tol</code>	The Spline fit tolerance
627	<code>export, solar, tangType</code>	end tangency type
628	<code>export, solar, surfEndCond</code>	Controls surface end conditions
629	<code>export, solar, tangTol</code>	End tangency fitting tolerance in degrees
630	<code>export, solar, equalSpacedEdges</code>	set the knot spacing to be equal
631	<code>export, solar, aspectRatio</code>	Maximum patch aspect ratio in degrees
632	<code>export, solar, growthRatio</code>	When using the Adaptive option for surface splines the surfaces splines are controlled with a Max. growth rate option
633	<code>export, solar, equalSpacedSurfs</code>	Surface splines can be controlled by setting Equal spaced
634	<code>export, solar, scalef</code>	The local spline fit parameter tolerance
635	<code>export, solar, gridU</code>	Number of patches for surfaces in the U direction
636	<code>export, solar, gridV</code>	Number of patches for surfaces in the V direction
637	<code>export, solar, gridT</code>	Number of segments for curves T
638	<code>export, solar, mindiv</code>	Set the minimum division
639	<code>export, solar, maxdiv</code>	Set the maximum division
640	<code>export, solar, initgrid</code>	Set the initial value
641	<code>export, solar, maxgrid</code>	St the maximum growth value
642	<code>export, cadoe, modelDimension</code>	This is the work space used by CADOE affecting the tolerance of internal operations
643	<code>export, jt, fileFormat</code>	The versions supported
644	<code>export, jt, fileStructure</code>	<p>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.</p> <p>Fully shattered - Each product structure node in the hierarchy is stored in an individual JT file.</p> <p>Monolithic - All product structure is stored in a single JT file.</p>
645	<code>export, jt, includeBrep</code>	Whether to include the BREP in the file
646	<code>export, jt, includeMassProps</code>	Mass properties are included with the export file
647	<code>export, jt, numLODs</code>	Level Of Detail values are used for creating multiple tessellations data sets. There are three options: 1, 2 or 3 defining different levels of model faceting.
648	<code>export, jt, multiSolidsAsAsm</code>	Whether to define multi-solids models as an assembly
649	<code>export, jt, useConfigFile</code>	Specifying whether an external configuration file is to be used.
650	<code>export, jt, configFile</code>	Path defining the configuration file for externally control supporting functions in a deployed application
651	<code>export, pdf, facetSag</code>	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.
652	<code>export, pdf, facetTurn</code>	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
653	<code>export, pdf, useTemplateFile</code>	User defined PDF template definition to be user

654	<code>export, pdf, templateFile</code>	PDF template file path
655	<code>export, pdf, viewBox</code>	PDF template definition
656	<code>export, pdf, pagesize</code>	PDF template page size
657	<code>export, pdf, orientation</code>	PDF template document orientation
658	<code>export, pdf, title</code>	PDF template document title
659	<code>export, pdf, author</code>	PDF template document author
660	<code>export, pdf, subject</code>	PDF template document subject
661	<code>export, pdf, creator</code>	PDF template document creator
662	<code>export, pdf, LOD</code>	PDF template level of detail
663	<code>export, pdf, chordHeightRatio</code>	PDF template definition
664	<code>export, pdf, minimalTriAngle</code>	PDF template definition
665	<code>export, pdf, lineWidth</code>	PDF template document thickness of line
666	<code>export, agps, pcurveTol</code>	tolerance for control pcurve projection
667	<code>export, agps, createCSC</code>	create surface connections
668	<code>export, matlab, facetSag</code>	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.
669	<code>export, matlab, facetTurn</code>	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
670	<code>export, matlab, facetLength</code>	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 1/100 th times the overall model diameter.
671	<code>export, matlab, minFacets</code>	This parameter specifies the minimum number of faces to be created on each edge.
672	<code>export, mesh, type</code>	Export of mesh maybe from three sources: free triangular 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.
673	<code>export, mesh, method</code>	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.
674	<code>export, mesh, format</code>	A Format of either FME (CADfix's own mesh file format) or NASTRAN (bulk data deck) may be chosen
675	<code>export, mesh, facetSag</code>	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.
676	<code>export, mesh, facetTurn</code>	This parameter specifies the maximum angle that an edge facet can turn through before it is too large and must be refined.
677	<code>export, mesh, facetLength</code>	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 1/100 th times the overall model diameter.
678	<code>export, mesh, expandRate</code>	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.

679	<code>export,mesh,minFacets</code>	This parameter specifies the minimum number of faces to be created on each edge.
680	<code>export,mesh,minThrough</code>	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.
681	<code>export,mesh,smooth</code>	The triangular faceting for quality will be further processed to smooth the facet shapes towards the ideal of equal edge lengths
682	<code>export,mesh,useSources</code>	Make use of on any mesh division control sources when generating the faceted model
683	<code>export,mesh,zones</code>	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 : 1100

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,mergeDuplicatas [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,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,deleteFilletlets,apply [0, 1] : 0
transform,deleteFilletlets,radmax [DEFAULT, value] : DEFAULT
transform,deleteFilletlets,radtol [DEFAULT, value] : DEFAULT
transform,deleteFilletlets,type [CONST, VARY] : VARY
transform,deleteFilletlets,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} {clmode REL}}
{cltol 1.0E-03} {tol DEFAULT} {factor 3}}
'Non-tangent segments'          , R-CU-NT  : {{apply 1} {test G1} {angle 0.1} {clmode REL}}
{cltol 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 2016  
##  
## 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 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 11 (26993/17870) UBID: cadfix1100fcs.6e9ak9g5  
#####
```

```
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