

SUBJECT: REPAIRS AND DIAGNOSTICS

KEYWORDS: ERROR MESSAGE, XPERT






DIAGNOSING AND SOLVING ERRORS IN SOLIDWORKS

The intention of the document is to provide you a fact sheet of the available diagnostics and repair tools in SolidWorks and how they can be used to solve error messages within the software. Although covered in our training courses, this document provides all the available tools in one single reference guide

Error Symbols

Error symbols will be placed on features, sketches and mates that have problems. You can reveal more detail reasons by right clicking and selecting **What's Wrong?**

Some of the common symbols and explanations are explained below:

- > Follows a feature with an **external reference**
- >? Follows a feature with an external reference **out of context**
- >* Follows a feature with **Locked** external references
- >x Follows a feature with **Broken** external references
- (f) Precedes a component that is **fixed** in an assembly
- (-) Precedes sketch/ component that is **under defined**
- (+) Precedes a sketch/ component that is **over defined**
- (?) Precedes a sketch/ component that is **not solved**
-  Is placed on an item that requires a **Rebuild**
-  A warning in the part/ feature below
-  An error in the part/ feature below
-  A warning in the part/ feature or the mate is over-defined. The feature will solve but could cause problems further down the line
-  An error in the part/ feature or the mate is not solved. The feature fails due to this error

System Options

There are a number of system options (**Tools> Options> System Options**) that are relevant for error diagnostics. The following are the most applicable:

Tools> Options> System Options.....

General

Show Errors Every Rebuild- *Shows the error message summary screen after every change*

Warn before Saving Documents with Update Errors- *warns about existing errors prior to saving*

Colours

Colour Scheme Settings- *define your own colours for sketches not solved, dangling, over-defined etc.*

Performance

Verification on Rebuild- *More thorough level of error checking ideal for surface modelling*

Rebuild Assembly on Load- *Will highlight errors immediately after opening*

Backup/ Recover

Set backups and Autorecover frequencies

Advanced

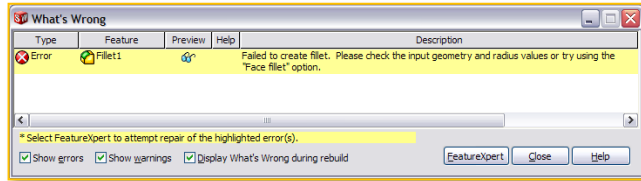
Allows you to bring back dismissed messages from previous SolidWorks sessions

SWIFT/ Xpert Tools

SWIFT (SolidWorks Intelligent Feature Technology) was introduced into SolidWorks during 2007. The suite of Xpert tools have been expanded ever since and help the user of the software speed up their designs and resolve errors in their models.

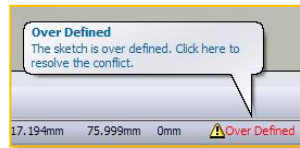
SWIFT tools will help diagnose Fillet and Draft features, as well as Sketch and Mate problems. The idea is to take the emphasis away from the user knowing SolidWorks inside out so they can concentrate on designing, not troubleshooting. The suite of SWIFT products include:

Feature Xpert



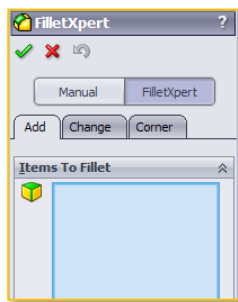
Available when errors are found with fillet and draft features. Feature Xpert can be used to attempt to solve the problem, typically splitting the feature into multiple commands to allow the feature to work.

Sketch Xpert



Available when a sketch becomes unsolvable. The Sketch Xpert will advise a number of solutions which can be accepted to get back to a solvable status. Also accessed from **Tools > Sketch Xpert**

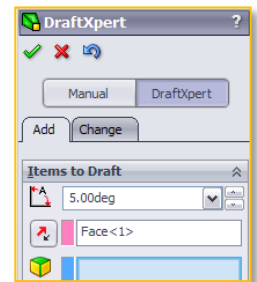
Fillet Xpert



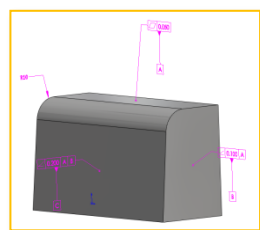
Used to speed up creating and editing fillets. Can be accessed at the top of the fillet command and is available for constant radius fillets only. Use the **Add** tab to apply multiple radii quickly, the **Change** tab to delete and edit the value of fillets, and the **Corner** tab to change the result of a fillet where 3 or more edges converge.

Draft Xpert

Used to speed up adding draft to faces. Combined with the draft analysis tool, the Draft Xpert allows multiple draft angles to be applied and sorts them into the solvable order. Will show the draft status live on screen.



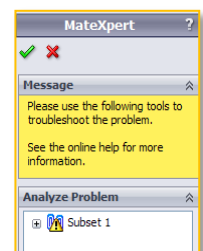
Dim Xpert



Used to fully constrain a model with dimensions, datums and tolerances. This makes drawing creation much quicker as the dimensioning scheme is already done. Is also a prerequisite for the **TolAnalyst** stack up add-in.

Mate Xpert

Used to highlight subsets of mates that are failing, whether over-defined or unsolved. This will highlight likely areas where the problems lie.



Sketch Repair Tools

When in sketcher mode it is sometimes difficult to diagnose problems that prevent from being created. The common feedback that prompts an error is the lack of an instant preview on the Extrude/ revolve feature. Typical sketching errors include;

Overlapping Geometry

An untrimmed section at a vertex point

Gaps between vertices

Two entities not converging at a vertex

Two lines lay on top of each other

Difficult to see, typical for converted edges

Over definition

Too many un-necessary dimensions or relations

Dangling Entities

Dimensions/ Relations that have lost their link due to subsequent deletion

Over defined sketches and dangling entities should be highlighted with error/ warning markers and diagnosed through the **What's Wrong** dialogue. Other errors are not so easy to spot, below are a few things that can help:

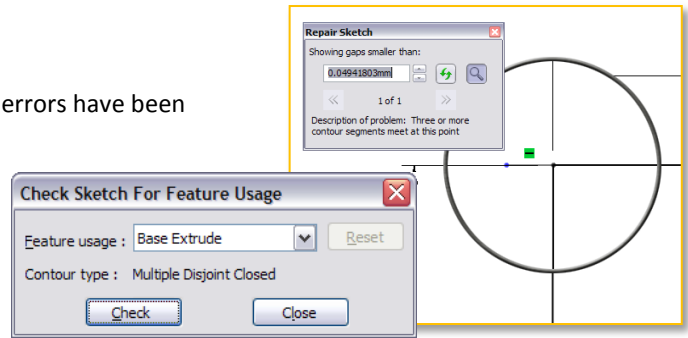
Tools that are available to you include:

Tools> Sketch Tools> Repair Sketch


This will highlight (using the magnifying glass tool) where sketch errors have been found

Tools> Sketch Tools> Check Sketch for Feature

This allows you to check the current sketch against its intended feature usage so you know beforehand the likelihood of it working. The list of features to check to include all of the standard SolidWorks features. If it fails this check it will indicate why and immediately allow you to use the **Repair Sketch** facility.



Rebuild

The rebuild tool will refresh the model after a change. Sometimes this is done automatically; sometimes this must be done manually. The rebuild traffic light symbol  will be seen in the bottom left hand corner of the SW window as well as attached to a particular part or feature. When this appears, either select the corresponding traffic light icon from the Standard toolbar or use **CTRL + B** from your keyboard.

The CTRL + B rebuild will only rebuild the last few changes that have been made to the model. A more thorough rebuild, which rebuilds every item in the tree in sequence is the **CTRL + Q** rebuild. This has no corresponding toolbar icon and is keyboard shortcut only. CTRL Q has been known to highlight errors not previously seen, but also solve errors that just require a rebuild.

Imported Data

Due to tolerance discrepancies between two CAD systems, imported models don't always translate correctly, similar to the spoken language. Therefore if using imported data (such as STEP, Parasolid, IGES, SAT) there are a few tools that can check and heal errors during imports.

Firstly it is useful to know that the Parasolid file type is the one most closely associated with SolidWorks and if available should be the preferred choice. Also you can see the supported file types by simply looking at **File> Open** and checking the list of file types at the bottom of the window.

When you import data you will be given the option to run **Import Diagnostics**. This tool will highlight errors that SolidWorks has found due to the translation. It also gives you the chance to **Attempt to Heal All** errors that are found (typically faulty faces or gaps). If you choose not to run the diagnostics tool immediately you can find it via **Tools> Import Diagnostics**.

A second check is found under **Tools> Check** this will check for problems with edges and face contacts and will highlight the errors, this is a good alternative when Import Diagnostics is not available.

Graphics Problems

Graphical issues can be diagnosed to a variety of reasons. Firstly check for interferences in your model, this is a common graphics problem that will only come to light on drawing sheets. Run **Tools> Interference Detection** in an assembly to check.

Within a drawing you may find edges seem to have disappeared. Try RMB on the view and selecting **Show Edge** maybe you accidentally hid them earlier.

Try CTRL Q rebuilding, this may refresh the graphics up to date.

Edge quality sometimes needs setting higher, try **Tools> Options> Document Properties> Image Quality** and increase the two slider bars (note this may lead to reduced performance).

Graphics card configurations are vital to SolidWorks' day to day performance, not only should you use a certified card, you should also use the tried and tested driver (IT IS NOT ALWAYS THE NEWEST DRIVER).

Check <http://www.solidworks.com/pages/services/VideoCardTesting.html> to ensure your driver is correct

Error Reporting

For general troubleshooting problems use some or all of the following to help:

SolidWorks Help Topics

Help> SolidWorks Help

SolidWorks Customer Portal

www.solidworks.com/customerportal

Register as an active subscription customer to enjoy a wealth of online benefits including **The Knowledge Base** and **User Groups and Forums...**great for problem solving

SolidWorks RX

Found in **START> All Programs> SolidWorks 2009> SolidWorks Tools**, the RX problem capture tool can run a diagnostics check to find any hardware related issues (check the web link to ensure you have the certified graphics driver installed), and also provides a facility to reproduce problems using a problem capture tools. This is ideal for your support representatives as it shows your workflow in video format and collects vital system information that can diagnose problems with the software.

Search Engines

Google, Yahoo and the like are great for generic problems, run a search and see what you find

Solid Solutions Support Desk:

Call **01926 333777** or

E-mail support@solidsolutions.co.uk