

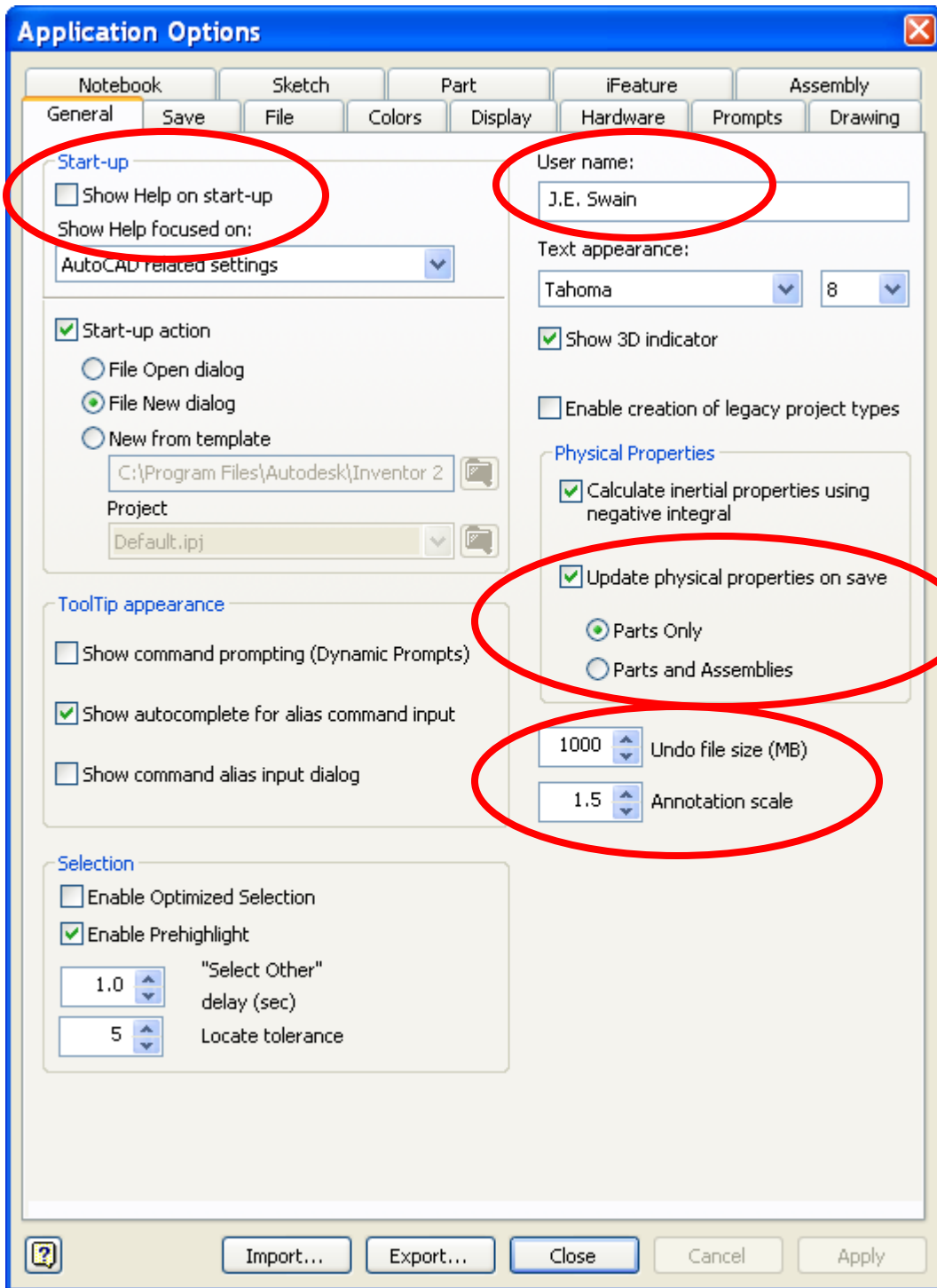
# Suggested Application Options Settings for Autodesk Inventor 2008

The following are suggested application option settings for Autodesk Inventor. This document was created based on the 2008 release of the software. They are also typical for the 2004 thru 2007 releases, though there are some variations. These settings are accessed through the Tools pull-down menu > Application Options.

Inventor 2008 has a new pair of commands in the Application Options dialog box. The **Import** and **Export** commands allow you to save changes to the application option settings and share them with other users.

They can also be used to make backup copies of your settings, allowing you to explore different choices, then return to earlier settings if desired.

## General



Option for **Show Help on start-up** is turned off. This option can be useful when first working with a new version of Inventor, but once you are comfortable with the information shown in the “What’s New” portion of the Help menu this can be turned off.

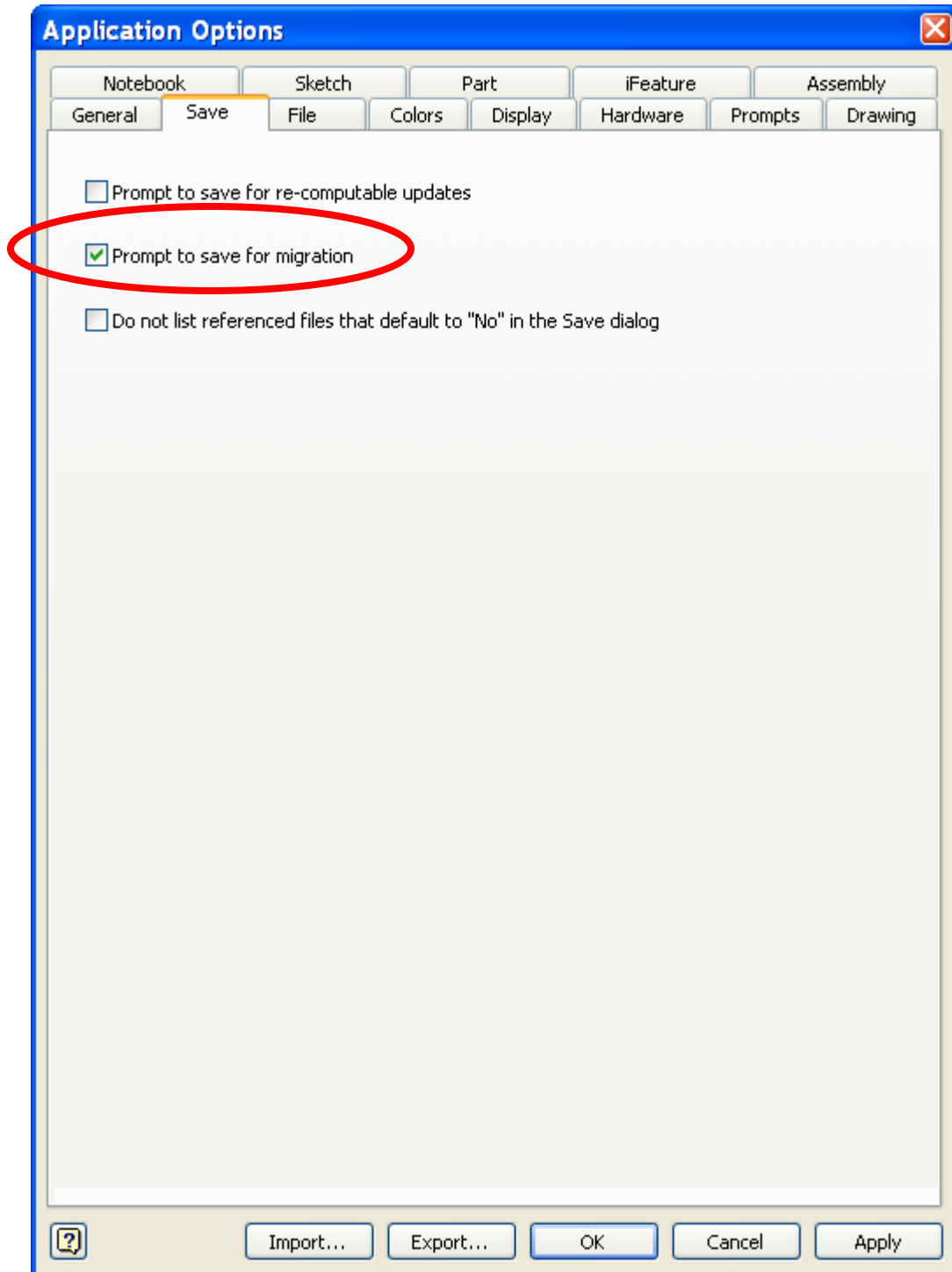
The **User name** has been changed. This text is used in the default Autodesk title blocks for the “Drawn By” name, so it has been changed to be more appropriate for the title block text.

**Update physical properties on save** is set to **Parts Only**. This command forces the mass properties to be recalculated when a part file is closed. The Parts and Assemblies option isn’t turned on, as this can slow down closing large assembly files.

**Undo file size (MB)** set to 1000. The size of this file directly affects how many steps can be undone during a session. The larger the file size, the more steps that can be undone. This is the maximum value allowed.

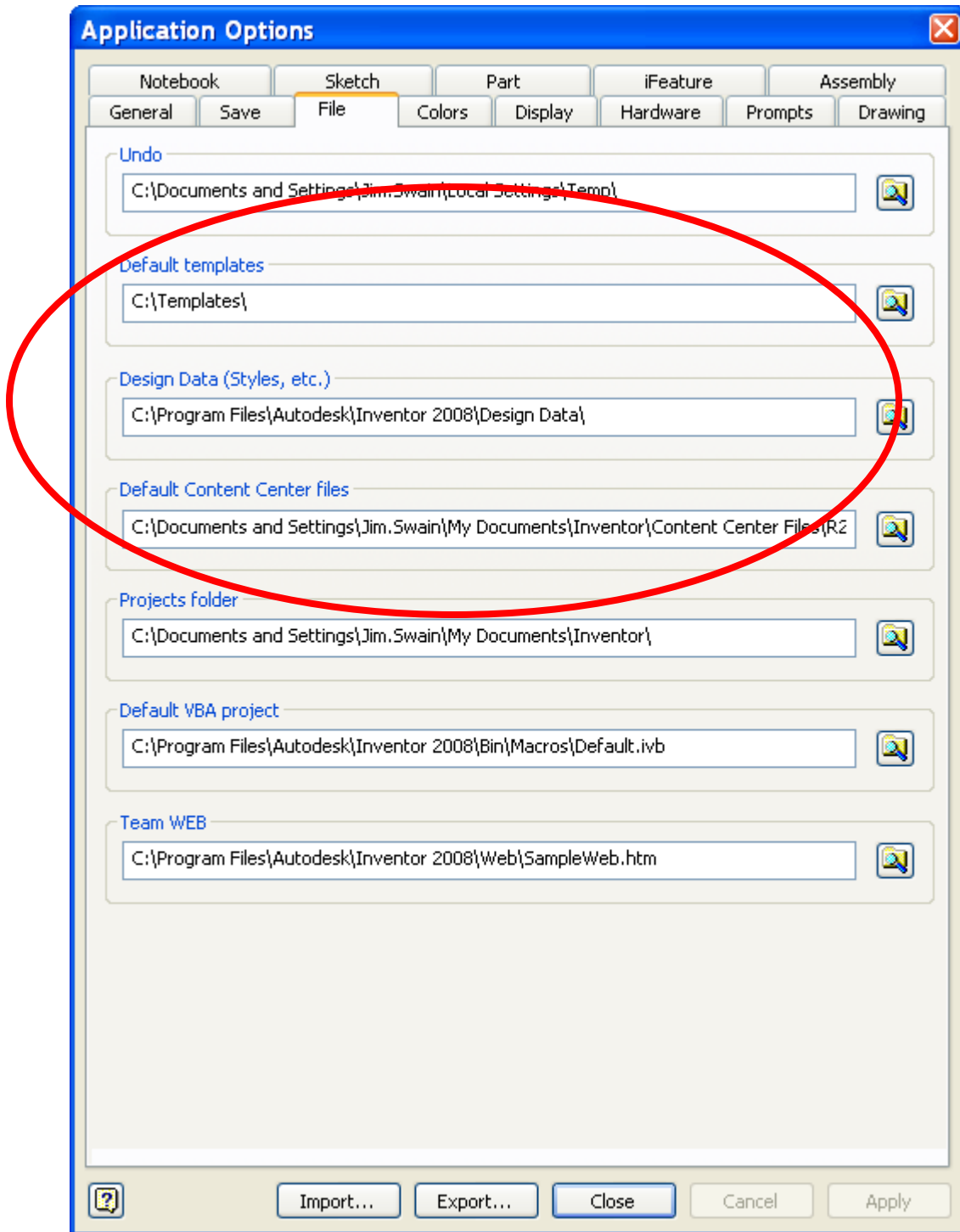
**Annotation scale** set to 1.5. This controls the size of dimensions during sketching, making them easier to read. Drawing dimensions aren’t affected by this value.

## Save



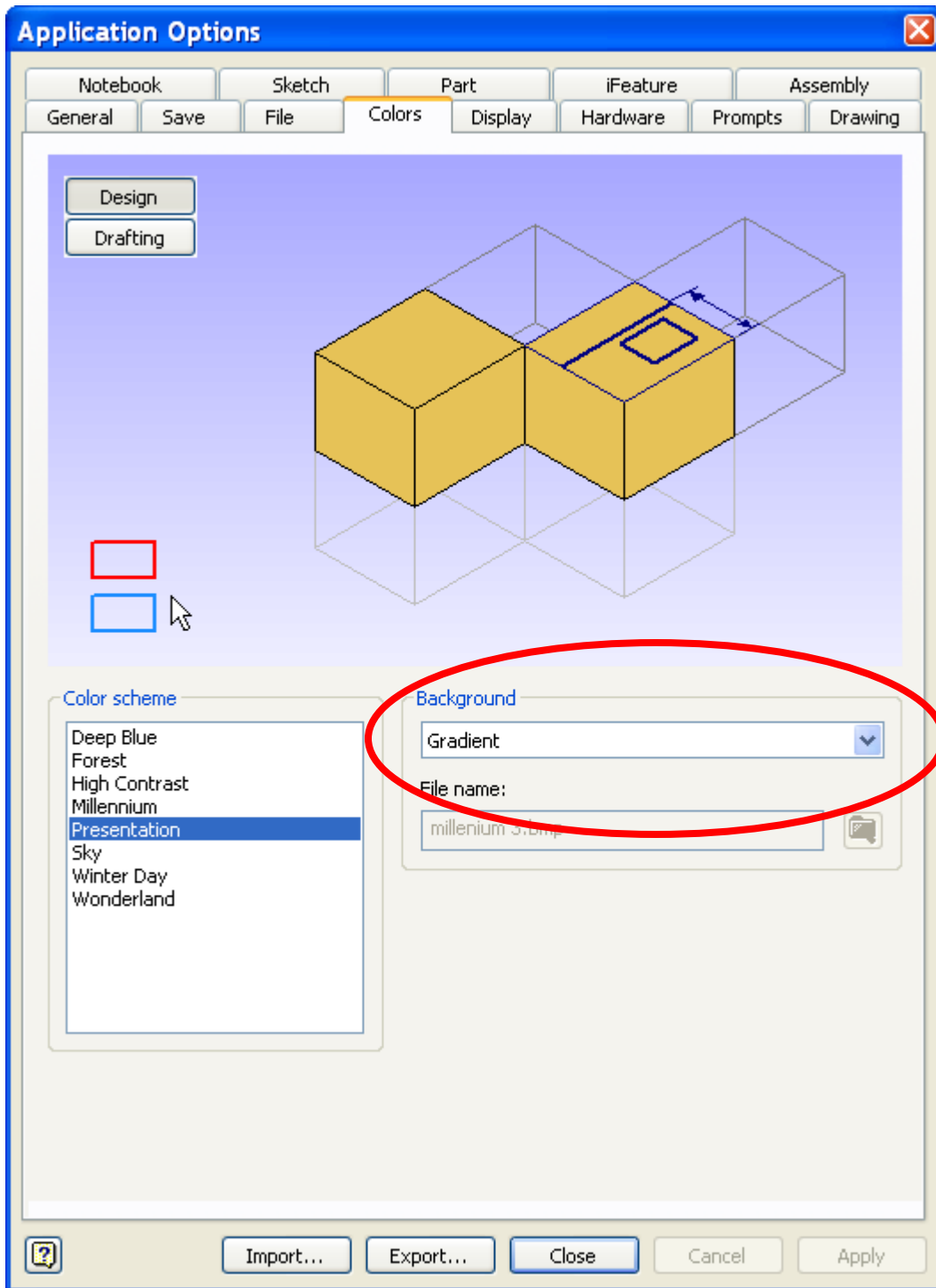
The **Prompt to save for migration** will ask if a migrated file (one coming from an older release) should be saved when the file is closed. This prompt will come up, even if no other changes have been made to the file.

## File



These three settings can be changed to point to a common location on a company network. This way templates, threaded hole sizes, fasteners, etc. don't have to be reloaded onto each machine when changes are made.

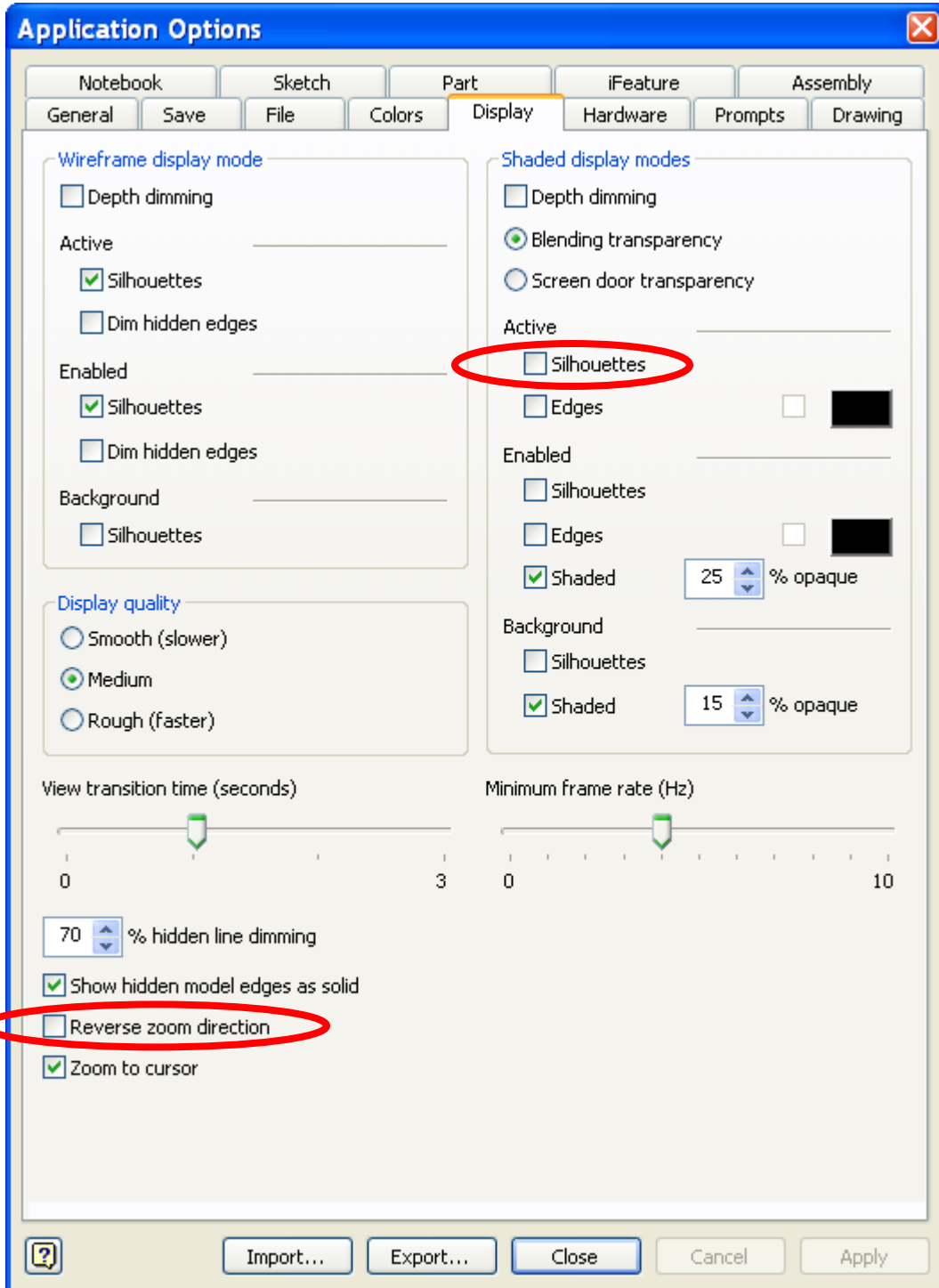
## Colors



You should experiment with the various color schemes to find the one that works best for them.

The **Background** can be changed to **1 color** to increase the performance of the graphics card.

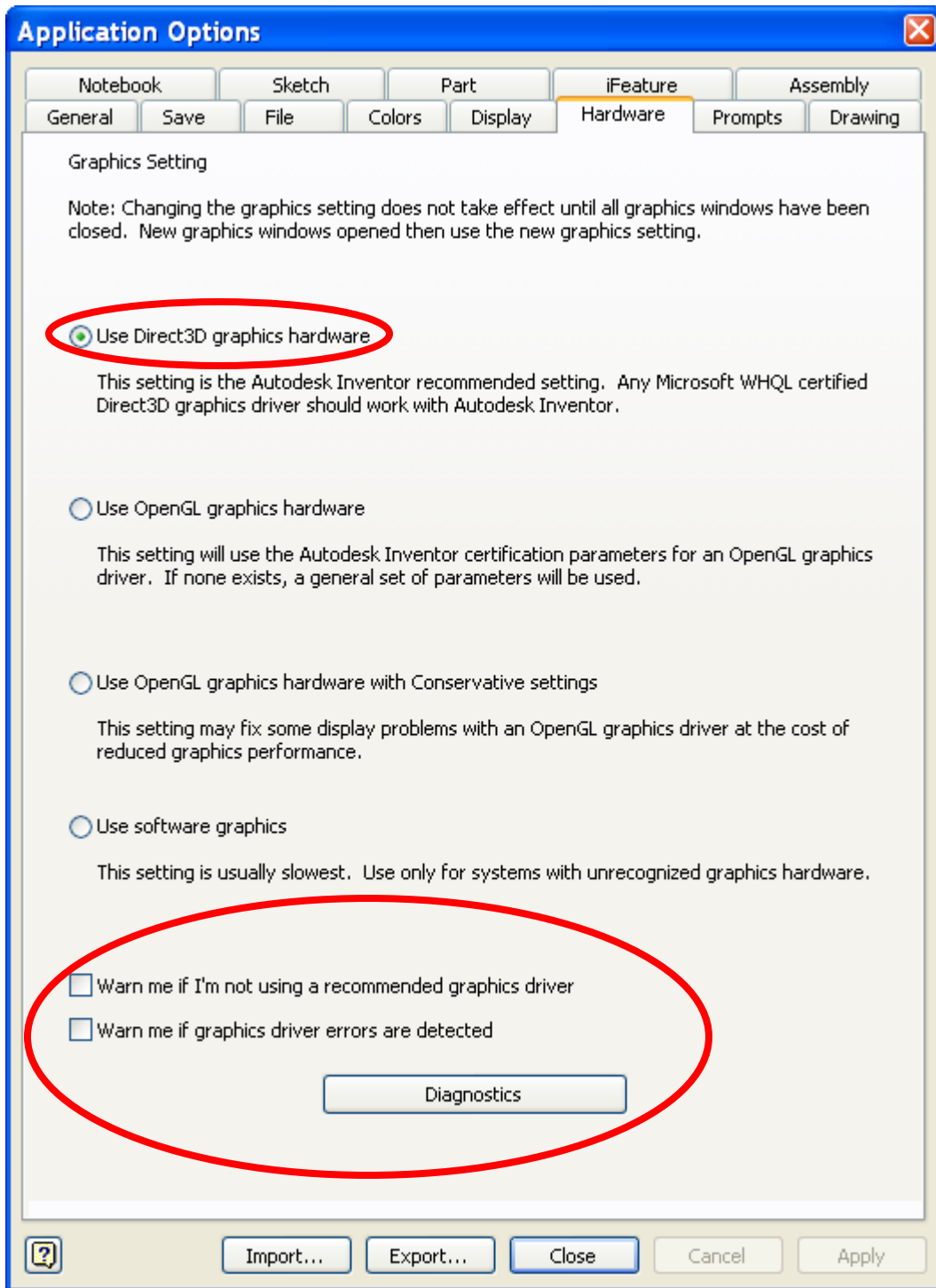
## Display



If designs frequently use cylindrical, conical or spherical shaped parts it can be very useful to turn on the display of the active part's silhouette edges. These edges are then available for projection onto the sketch plane.

The Reverse zoom direction option can be turned on to change Inventor's wheel mouse behavior to match the default behavior of AutoCAD, if desired.

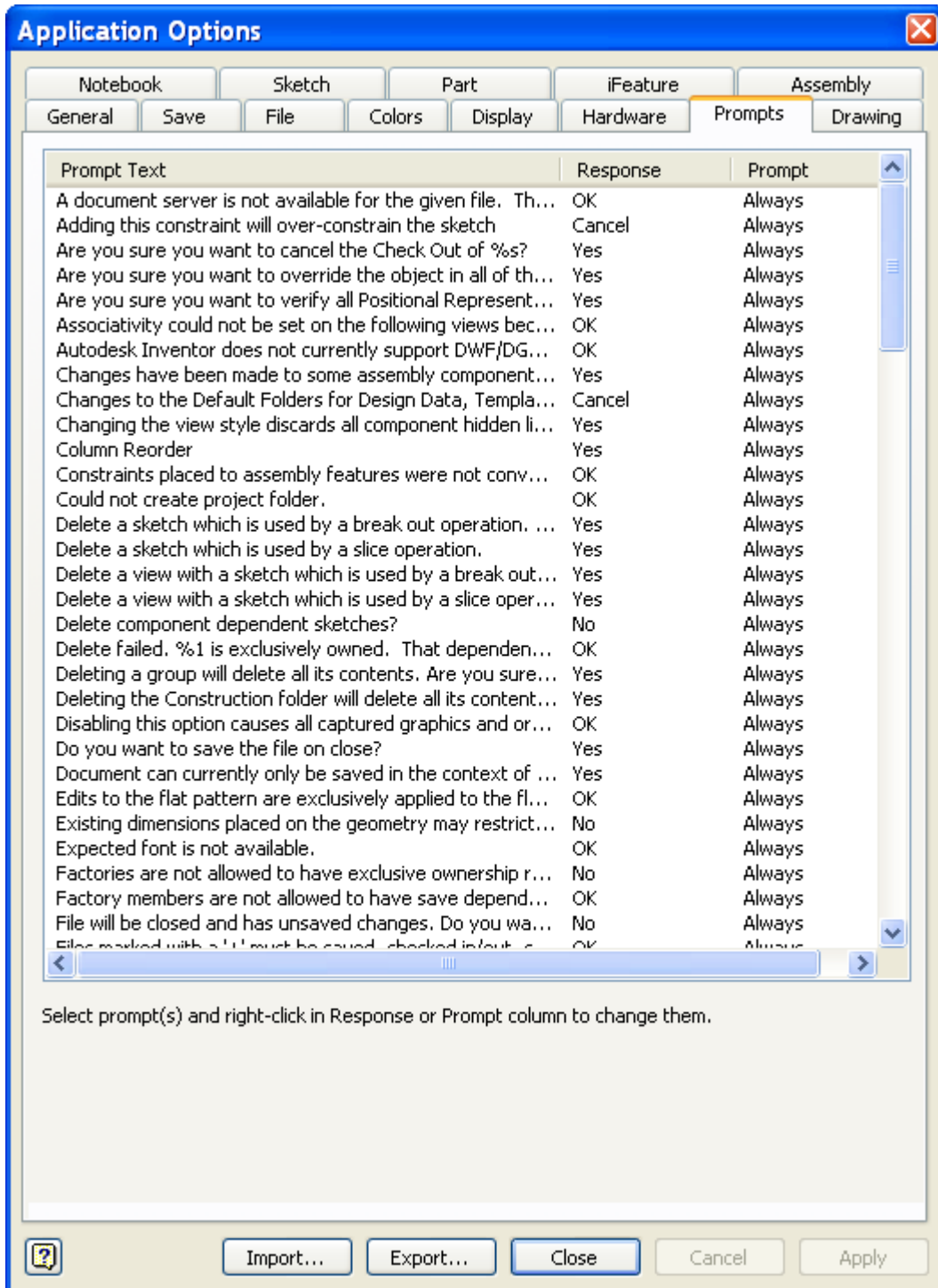
## Hardware



Direct3D graphics drivers have been available in Inventor beginning with R11. Make sure you are running a certified graphics card and driver.

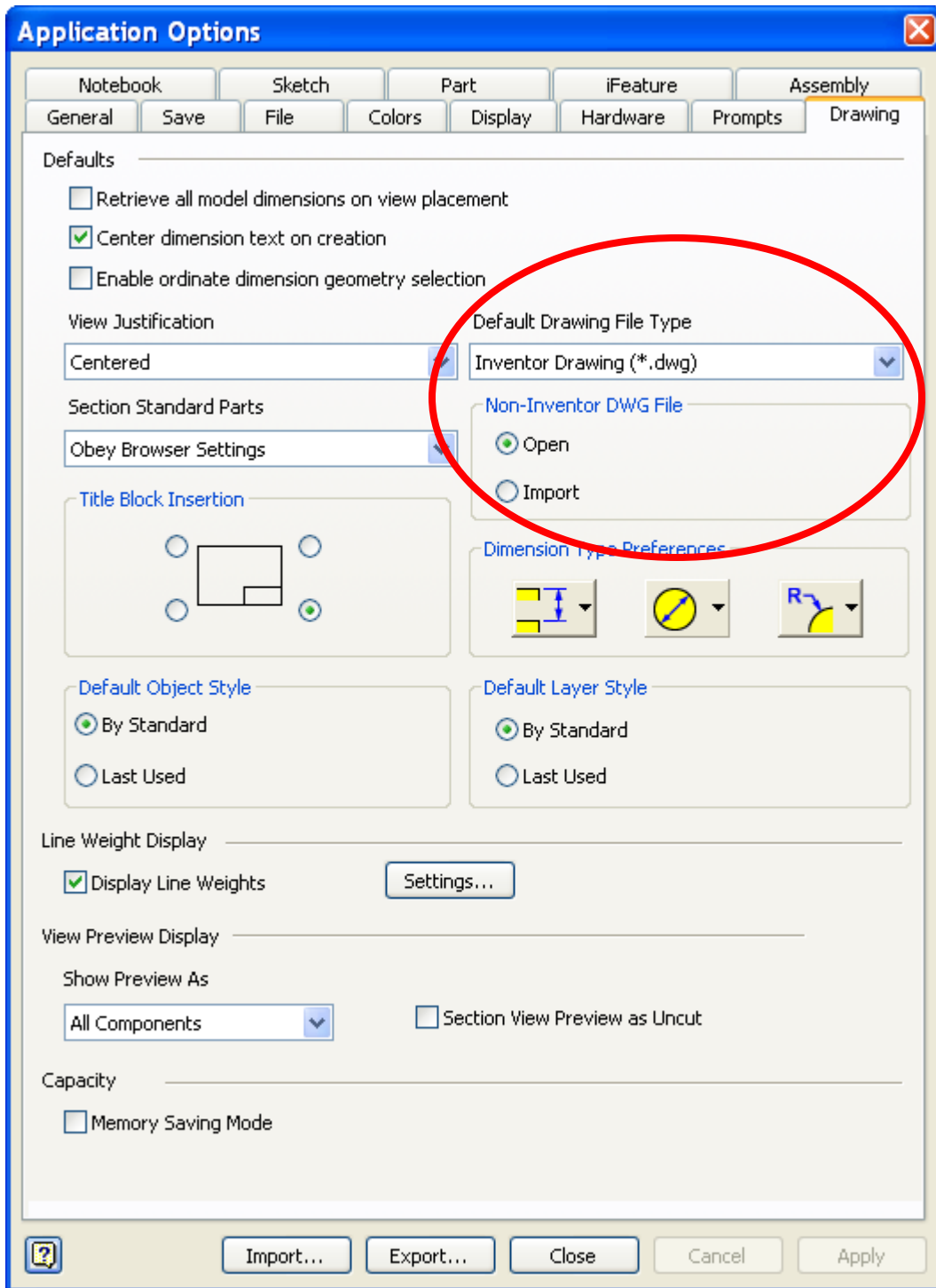
The warning options can be turned on if desired, however the Diagnostics command should be run when Inventor has a 3D model open and displayed in shaded mode.

## Prompts



Default prompt behavior can be changed as desired.

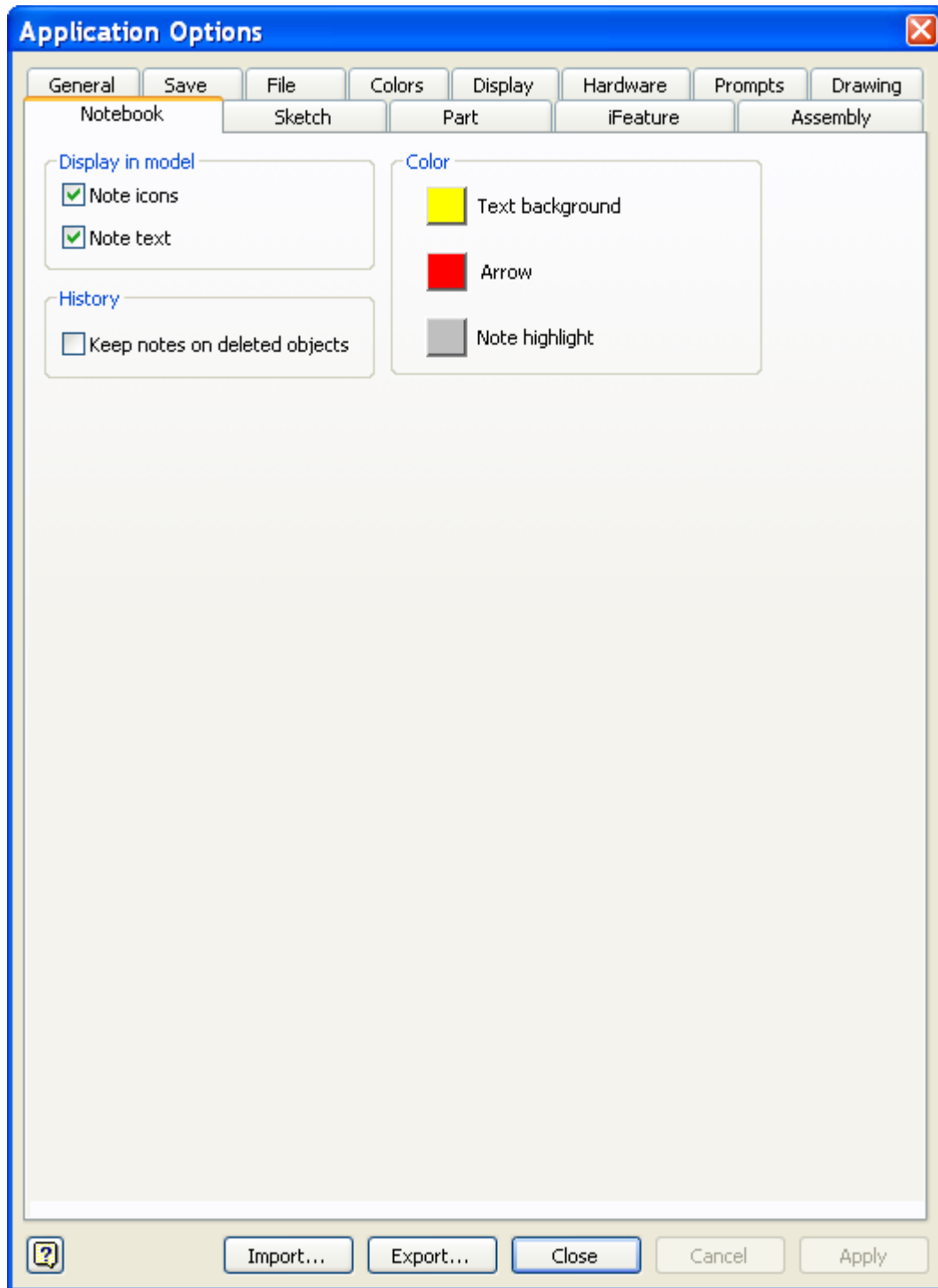
## Drawing



The **Default Drawing File Type** can be set to **Inventor Drawing (\*.dwg)**. This setting allows easier drawing sharing with AutoCAD.

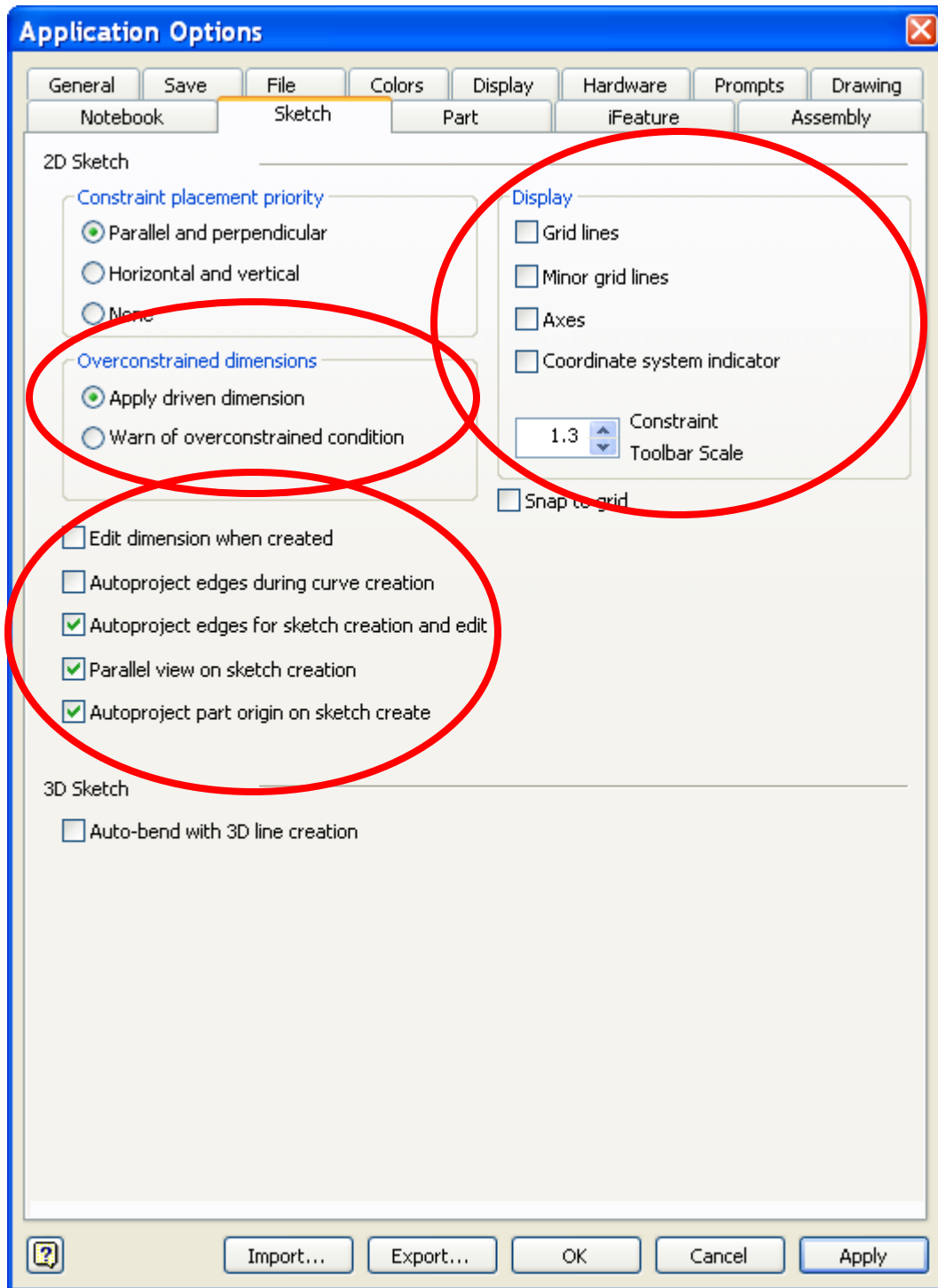
Make sure the **Open** option is set for **Non-Inventor DWG File** behavior. AutoCAD data can then be copy and pasted without stepping thru the Import wizard.

## Notebook



The settings on this tab have been left unchanged.

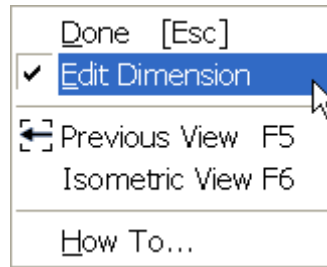
## Sketch



Change the **Overconstrained dimensions** behavior to **Apply driven dimension**. This will avoid having a warning dialog box appear whenever an overconstraining dimension is created.

The **Edit dimension when created** setting can be changed here or with the right-click menu of the General Dimension command while sketching. This should be on when

capturing and existing design in Inventor. It can be turned off when exact dimensions values aren't known and dimensions are being created more as place-holders.



**Autoproject edges for sketch creation and edit** can be on for relatively simple part cross sections, but should be turned off before placing sketches on more complex faces, such as gears.

Parallel view on sketch creation is a matter of personal preference. If it is turned on it the viewpoint shifts to be looking straight at the sketch plane whenever a sketch is created or edited. The previous viewpoint is restored when the sketch editing is done.

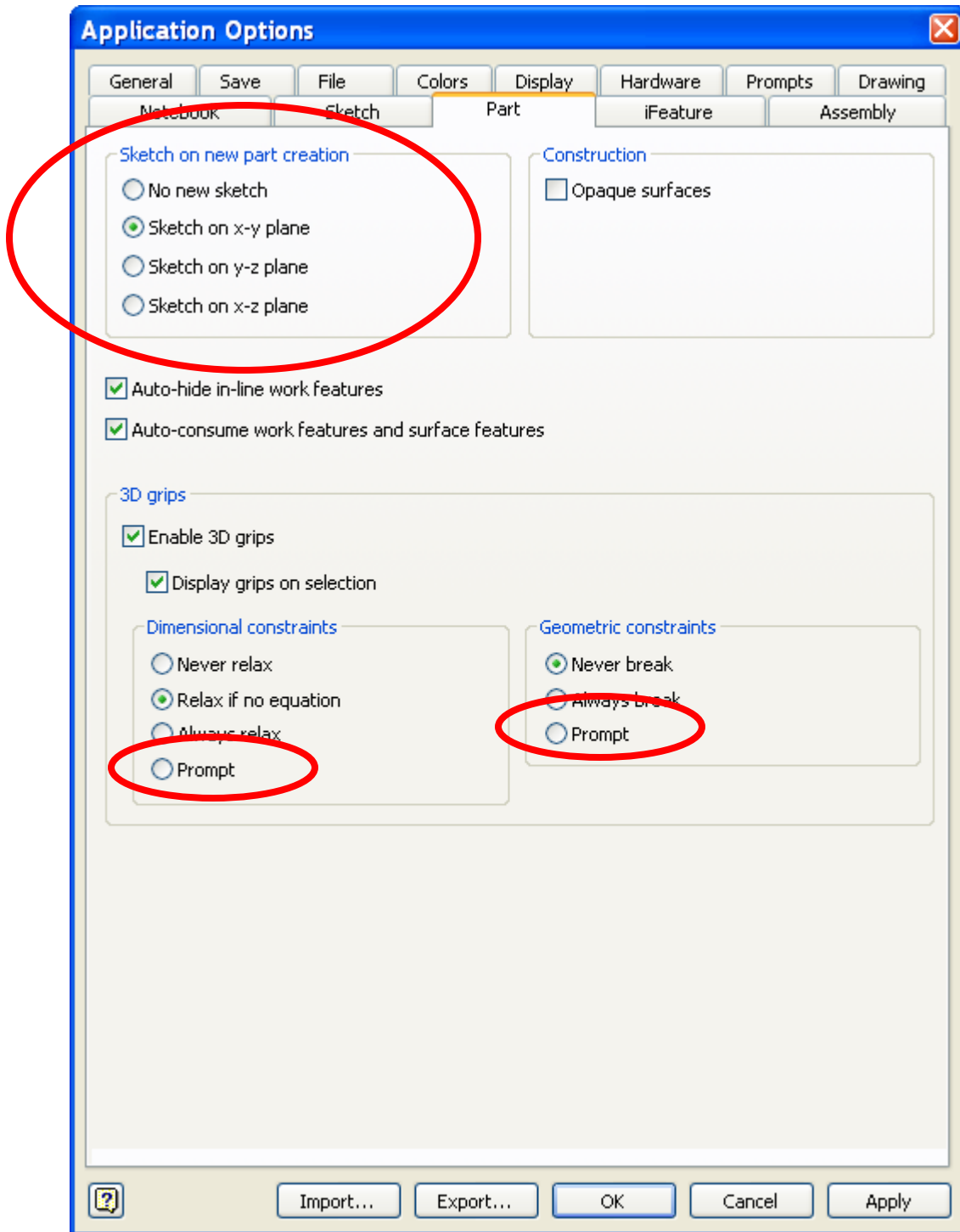
**Autoproject part origin on sketch creation** will give a projected reference point on any sketch.

Note: this can cause problems for adaptive sketches. Delete the projected point if the sketch is going to be adaptive.

Turn off the display of the **Grid lines, Minor grid lines, Axes** and **Coordinated system indicator**. This will clean up the sketch display.

Increasing the **Constraint Toolbar Scale** can make reading the constraint indicators much easier.

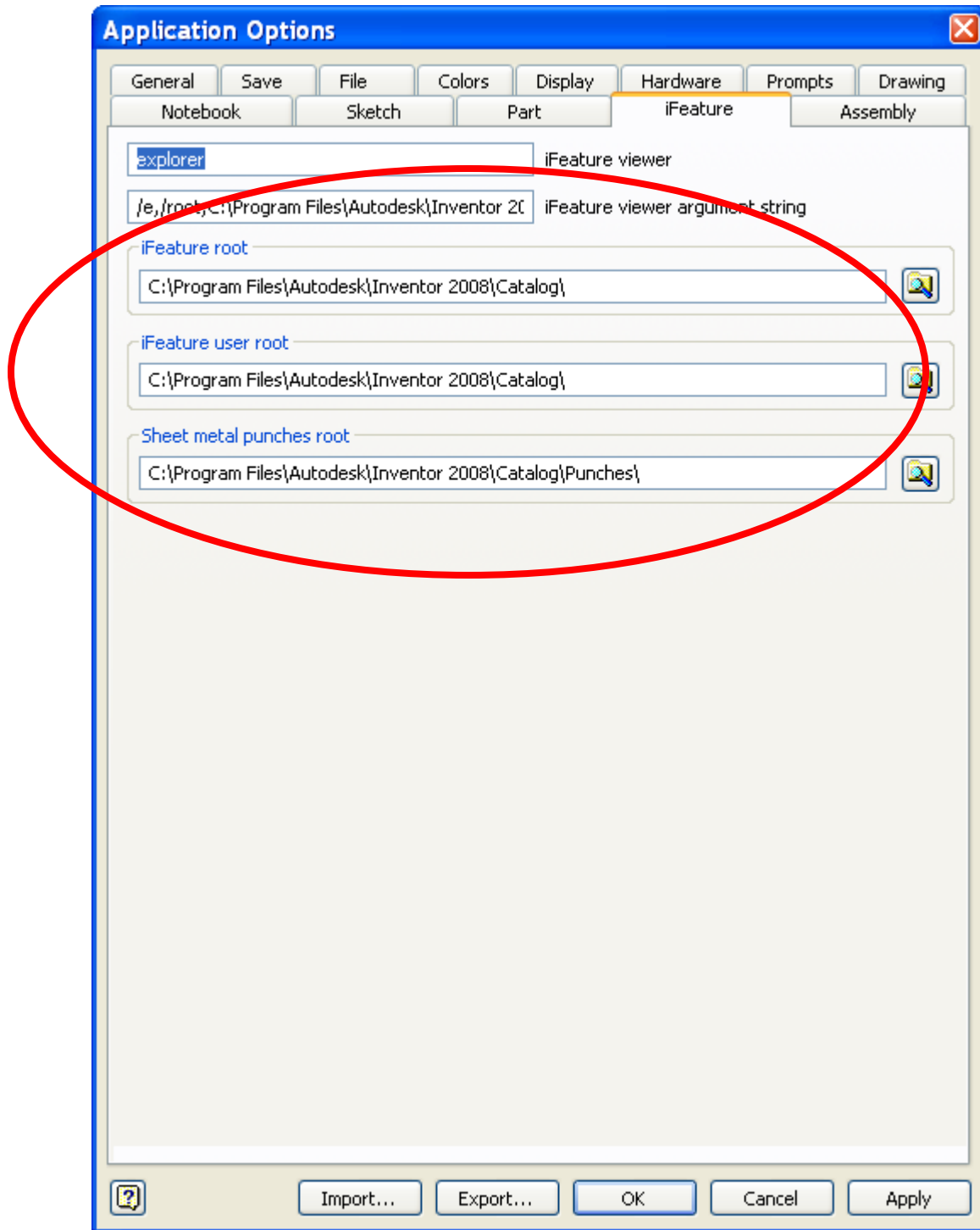
## Part



For many designs the **Sketch on x-y plane** (front view) is a logical starting point, but for designs such as factory equipment **Sketch on the x-z plane** is a better choice. This choice has the part's elevation as the default extrusion direction.

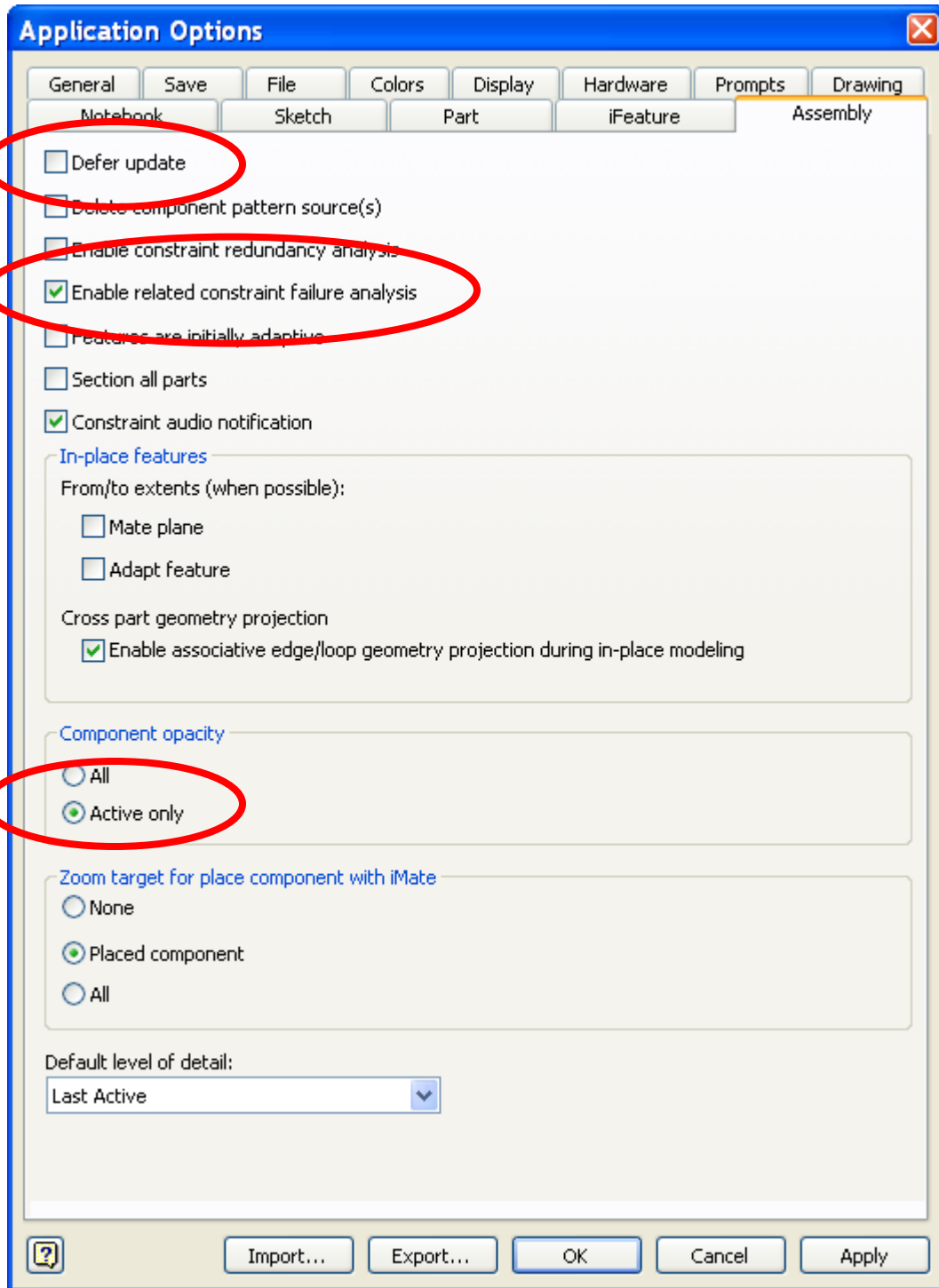
If you frequently use 3D grips consider setting the dimension and geometric constraint behaviors to **Prompt**.

## iFeature



These three settings can be changed to point to a common location on a company network. This way typical features and sheet metal punches don't have to be reloaded onto each machine when changes are made.

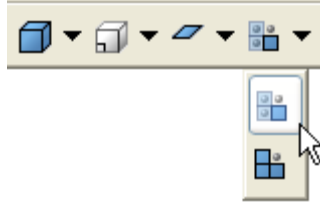
## Assembly



The **Defer update** option can be turned on when working on very large assemblies. This allows work to be done in several areas before a possibly time-consuming update is performed.

The **Enable related constraint failure analysis** option can be useful when working with complicated assemblies.

Default **Component opacity** is **Active only**. Check to make sure this is the current setting. This can also be set with a button on the Inventor Standard toolbar.



These suggestions incorporate many of the changes in the user interface. Options are added and removed in each new release. When new versions are installed take a look at the Options dialog box and see what new tools might help you.