

Table Of Contents

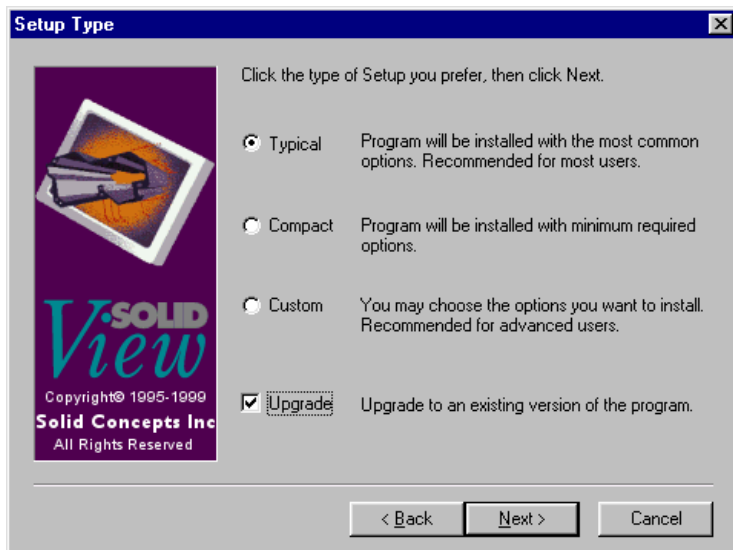
Overview	1
Eligibility	1
General 3.55 Changes	1
SolidView/Pro Changes	3
CAD Interfaces	3
Known Problems	4

Overview

SolidView Version 3.55 is a minor enhancement release that also incorporates a number of error corrections. SolidView/Lite, SolidView, SolidView/Pro and SolidView/Pro RP are all included in this new release. Version 3.55 includes changes that impact all the SolidView products, as well as changes that only impact SolidView/Pro and SolidView/Pro RP. In addition to the changes to the SolidView products, new CAD interfaces are now available in version 3.55.

Eligibility

Existing SolidView, SolidView/Pro, and SolidView/Pro RP users of version 3.3 or later are eligible for a no-cost upgrade to version 3.55. Users of previous versions of SolidView should contact their Authorized SolidView Reseller or Solid Concepts to determine their eligibility before attempting to upgrade to version 3.55. Eligible SolidView users with single-computer licenses can upgrade to 3.55 without needing a new SolidView Program Key by selecting the “Upgrade Option” during the installation process, as shown below.



Network license users will require a new Program Key from their Authorized SolidView Reseller or Solid Concepts in order to enable SolidView after the installation process. Users can obtain a free 15 day evaluation program key from <http://www.solidview.com/key.htm>.

General 3.55 Changes

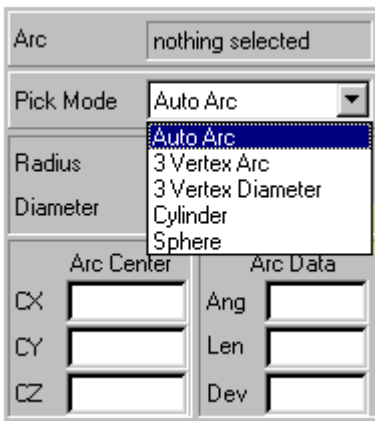
The changes to 3.55 that effect all SolidView products are:

SFX File Upload

The SolidView SFX Upload feature allows users to automatically upload an SFX file, complete with any changes or annotations to a specified Web server. For example, with this feature users of the FREE SolidView/Lite viewer can make changes to an SFX file, and while they will not be able to save their changes locally, they can upload the changes to a pre-determined Web server. In order for this feature to be activated, you must have purchased an SFX Upload from Solid Concepts or your authorized SolidView reseller and you must have the required Web server software to perform the actual upload. For more information on this feature, please contact Solid Concepts or your authorized SolidView reseller.

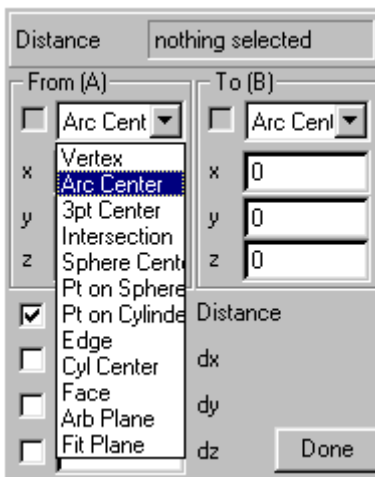
Measure Cylinder and Sphere

SolidView can now automatically identify and measure cylinders and spheres in the Measure Arc function. These two new options are available in addition to the existing Measure Arc options. For more information, click on the SolidView Measure Arc tool button, place your mouse pointer in the Measure Arc dialog and press F1 to display the SolidView online help for Measure Arc.



Cylinder and Sphere Options in Measure Distance

The SolidView Measure Distance function now includes options for Point on Sphere, Point on Cylinder, and Cylinder Center. For more information, click on the SolidView Measure Distance tool button, place your mouse pointer in the Measure Distance dialog and press F1 to display the SolidView online help for Measure Distance.



Modify Units

The SolidView Modify Units function has been changed to clarify the function and to minimize the possibility of converting geometry to the wrong units. In previous versions of SolidView, it was possible to set the From Units option to any setting, leading to the potentially incorrect modification of the geometry. With version 3.55, if the units of the selected objects is known, the From Units option will be disabled. If all the selected objects have the same units, then the From Units will be shown, but the user will not be able to modify the From Units. If multiple objects are selected, and the objects are in different units, no From Units will be displayed. Objects that are already in the units specified by the To Unit option will not be modified.

Face Colors

SolidView now supports different colors on individual features on an object. This is supported in SolidWorks, SDRC IDI files, and Polyline files.

SolidView/Pro Changes

The following changes are included in version 3.55 of SolidView/Pro and SolidView/Pro RP. Note that the availability of some features may be dependent on licensing optional SolidView/Pro features.

Network Log File

SolidView/Pro can keep a log file of Network license usage for planning and tracking purposes. To enable this feature, you must have a SolidView/Pro Network License and you must insert a file named "sldview.log" in the SolidView/Pro network folder (sviwnet) created during the network installation. The Network Log file will contain an "In" and "Out" entry for each SolidView/Pro and CAD Interface license. This information is saved in fixed length records, as shown below:

```
Wed Sep 13 08:50:11 2000 (SCI) IN: "SolidView" @SHOP01
Wed Sep 13 08:50:36 2000 (SCI) IN: "C:\SV355\PLUGINS\psfi.dll" @SHOP01
Wed Sep 13 08:50:43 2000 (SCI) OUT: "C:\SV355\PLUGINS\psfi.dll" @SHOP01
Wed Sep 13 08:50:47 2000 (SCI) OUT: "SolidView" @SHOP01
```

The SolidView Network Log file can be read into any spreadsheet application to determine minimum, peak, average, and other SolidView/Pro usage trends.

Save SFX As

The SolidView/Pro Save SFX As function now includes options for saving SFX data in a four view .BMP file as well as in an animated .GIF file.

Polygon Files

SolidView/Pro now supports direct access to Polygon files in .ply format.

CAD Interfaces

The following CAD interface additions and changes are available in SolidView/Pro 3.55:

SolidEdge

This is a new optional interface for SolidEdge part, assembly and sheet metal files. Specifically, the SolidView SolidEdge interface supports .par, .asm, and .psm files from SolidEdge.

SolidWorks Geometry

This is an optional interface for SolidWorks part geometry that allows the user to specify the tessellation tolerance when importing SolidWorks part files. The standard SolidWorks interface that is included in all SolidView products uses a fixed tessellation tolerance that is determined by the display setting used when the

part or assembly was saved in SolidWorks. Often, this tessellation tolerance was not suitable for rapid prototyping and FEA applications. With the SolidWorks Geometry interface, users can import the geometry with a specified tessellation tolerance and then output .STL for rapid prototyping or FEA applications.

SDRC Web Access

The SolidView/Pro SDRC Web Access has been enhanced to support the IDI files generated by SDRC Master Series version 8.

Unigraphics

The SolidView/Pro Unigraphics interface no longer requires access to any UG license, however this interface only supports Unigraphics V12 – V15 part files. Support for V16 part files is expected to become available in the near future. Drawing and assembly support will not be available with this interface.

ACIS/SAT

The SolidView/Pro ACIS/SAT interface now supports SAT data in DWG format.

Known Problems

The following problems are known to be in SolidView 3.55.

Pro/E Import

In some instances, surfaces are missing from imported Pro/E geometry. A temporary solution to this problem is to decrease the import tolerance. For example, if surfaces are missing at a tolerance of .005 inches, they may be imported correctly at a tolerance of .0005 inches.

IGES Import

In some instances, the SolidView IGES import option will fail, due to conflicts with changes needed to support the Pro/E and Catia import options. Users can download an older version of the libgeom.dll file from www.solidview.com/support.htm and install this file in place of the one distributed with version 3.55 to resolve this problem. Installing this .dll will render the Catia and Pro/E interfaces inoperable unless the 3.55 .dll is re-installed.