


**About SolidWorks Product Activation**

Version: 15.3.1.0127  
Computer ID: 5KH2HF59  
Computer Name: englab01

OK

**SolidWorks Product Activation**

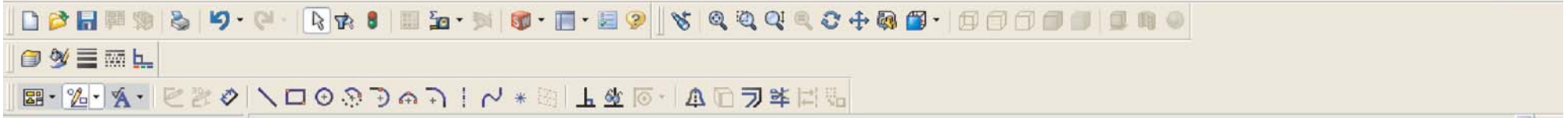


Currently activated products:

Product	Expires
SolidWorks Office Professional	Never

Refresh

About... OK Cancel Help



**Print**

Document Printer

Name: VLS4.60 Properties

Status: Ready Page Setup...

Type: VLS4.60

Where: UCP-1

Comment:

System Options

Line Weights... Margins...

Document Options

Header/Footer...

Print range

All  Selection

Pages:

Enter page numbers/ranges.  
For example: 1,3,5-8,10

Number of copies: 1

Print background

Print to file

Convert draft quality drawing views to high quality

OK Close Help



**Print**

Document Printer  
Name: VLS4.60 Properties  
Status: Ready Page Setup...

**Line Weights**

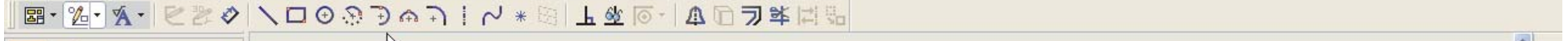
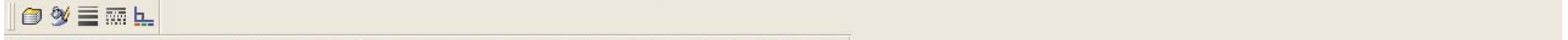
Thin: 0.05mm	Thick(3): 0.7mm
Normal: 0.25mm	Thick(4): 1mm
Thick: 0.35mm	Thick(5): 1.4mm
Thick(2): 0.5mm	Thick(6): 2mm

OK Cancel Help

Enter page numbers/ranges.  
For example: 1,3,5-8,10

Print to file  
 Convert draft quality drawing views to high quality

OK Close Help



Part1

- Blocks
- Design Binder
  - Design Journal.doc <Empty>
- Annotations
- Sheet1

TEST OF SOLIDWORKS

**Print**

Document Printer  
Name: VLS4.60 Properties  
Status: Ready Page Setup...  
Type: VLS4.60  
Where: UCP-1  
Comment:

System Options  
Line Weights... Margins...  
Document Options  
Header/Footer...

Print range  
 All  Selection  
 Pages:   
Enter page numbers/ranges.  
For example: 1,3,5-8,10

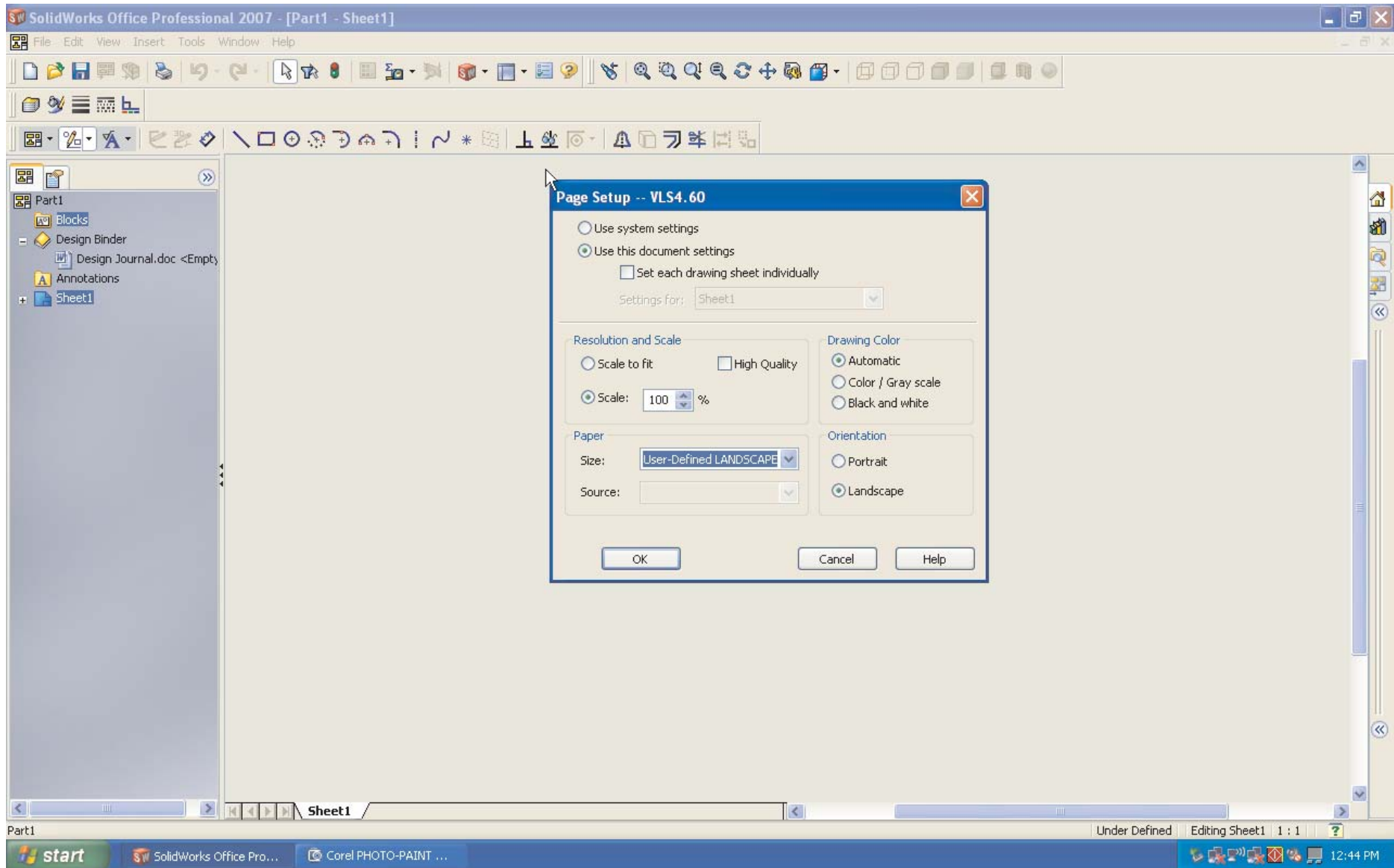
Number of copies: 1  
 Print background  
 Print to file  
 Convert draft quality drawing views to high quality

OK Close Help

**Line Weights**

Thin:	0.05mm	Thick(3):	0.7mm
Normal:	0.25mm	Thick(4):	1mm
Thick:	0.35mm	Thick(5):	1.4mm
Thick(2):	0.5mm	Thick(6):	2mm

OK Cancel Help




- Information
- Feature Creation Tools
- System Tools
- Reference Geometry
- Features
- Parts
- Assemblies
- Configurations
- SolidWorks Toolbox
- Drawings
  - Drawings Overview
  - Getting Started in Drawin
    - Setting Options for Dr
    - Creating a Drawing
    - Sheet Format/Size
    - The Drawing Window
    - Sheet Formats, Sheet:
    - Customizing Sheet Fo
    - Saving Sheet Formats
    - Sheet Properties
    - Multiple Drawing Shee
    - Renaming Sheets
    - Linking Notes to Docu
    - Views of Parts and As:
    - View Boundaries
    - Scales in Drawings
    - 2D Sketching in Drawi
  - Types of Drawing Docum
  - Standard Drawing Views
  - Derived Drawing Views
  - Drawing View Alignment
  - Drawing Tools
  - Drawing Outputs
- Detailing
- SolidWorks Design Checker
- Import/Export
- ScanTo3D
- DXF/DWG Import Wizard
- FeatureWorks

## Creating a Drawing

Drawings consist of one or more views generated from a part or assembly. The part or assembly associated with the drawing must be saved before you can create the drawing. You can create a drawing from within a part or assembly document.

Drawing files have the **.slddrw** extension. A new drawing takes the name of the first model inserted. The name appears in the title bar. When you save the drawing, the name of the model appears in the **Save As** dialog box as the default file name, with the default extension **.slddrw**. You can edit the name before saving the drawing.

### To create a drawing from within a part or assembly document:

1. Click **Make Drawing from Part/Assembly**  on the Standard toolbar.


2. Select a [Sheet Format/Size](#), then click **OK**.




If you click **Cancel**, no sheet format is used.

3. Drag views from the [View Palette](#) into the drawing sheet, and set options in the [PropertyManager](#).

### To create a new drawing:

1. Click **New**  on the Standard toolbar, or click **File, New**.

2. In the **New SolidWorks Document** dialog box, select **Drawing** , then click **OK**.

3. Select a [Sheet Format/Size](#), then click **OK**.



If you click **Cancel**, no sheet format is used.

4. In the [Model View PropertyManager](#), select a model from **Open documents** or browse to a part or assembly file.

5. Specify options in the [PropertyManager](#), then place the view in the graphics area.

- Information
- Feature Creation Tools
- System Tools
- Reference Geometry
- Features
- Parts
- Assemblies
- Configurations
- SolidWorks Toolbox
- Drawings
  - Drawings Overview
  - Getting Started in Drawing
    - Setting Options for Drawing
    - Creating a Drawing
    - Sheet Format/Size
    - The Drawing Window
    - Sheet Formats, Sheet Properties
    - Customizing Sheet Formats
    - Saving Sheet Formats
    - Sheet Properties
    - Multiple Drawing Sheets
    - Renaming Sheets
    - Linking Notes to Documents
    - Views of Parts and Assemblies
    - View Boundaries
    - Scales in Drawings
    - 2D Sketching in Drawing
- Types of Drawing Documents
- Standard Drawing Views
- Derived Drawing Views
- Drawing View Alignment
- Drawing Tools
- Drawing Outputs
- Detailing
- SolidWorks Design Checker
- Import/Export
- ScanTo3D
- DXF/DWG Import Wizard
- FeatureWorks



## Sheet Format/Size


You select a sheet format when you open a new drawing. The standard sheet formats include [links](#) to system properties and custom properties.

Sheet formats help create drawings with a uniform format. Drawing sheet formats are OLE documents in which you can embed objects such as bitmaps.

You can also create a drawing with no sheet format.

### To select a sheet format:

1. Click **New**  on the Standard toolbar.
2. Select **Drawing** , then click **OK**.
3. Select from the following, then click **OK**. If you click **Cancel**, no sheet format is used.
  - **Standard sheet size.** Select a standard sheet size, or click **Browse** and locate a [custom sheet format file](#).
  - **Display sheet format** (available for standard sheet sizes). Display border, title block, and so on.
  - **Custom sheet size.** Specify a **Width** and **Height**.

 To select a different sheet format in an existing drawing document, right-click in the graphics area and select [Properties](#). To save a sheet format, click **File**, [Save Sheet Format](#).

Hide Back Print

Contents Index Search

- Information
- Feature Creation Tools
- System Tools
- Reference Geometry
- Features
- Parts
- Assemblies
- Configurations
- SolidWorks Toolbox
- Drawings
  - Drawings Overview
  - Getting Started in Drawings
    - Setting Options for Drawing
    - Creating a Drawing
    - Sheet Format/Size
    - The Drawing Window
    - Sheet Formats, Sheet Properties
    - Customizing Sheet Formats
    - Saving Sheet Formats
    - Sheet Properties
    - Multiple Drawing Sheets
    - Renaming Sheets
    - Linking Notes to Documents
    - Views of Parts and Assemblies
    - View Boundaries
    - Scales in Drawings
    - 2D Sketching in Drawings
  - Types of Drawing Documents
  - Standard Drawing Views
  - Derived Drawing Views
  - Drawing View Alignment
  - Drawing Tools
  - Drawing Outputs
- Detailing
- SolidWorks Design Checker
- Import/Export
- ScanTo3D
- DXF/DWG Import Wizard
- FeatureWorks



## Sheet Format/Size

You select a sheet format when you open a new drawing. The standard sheet formats include [links](#) to system properties and custom properties.


Sheet formats help create drawings with a uniform format. Drawing sheet formats are OLE documents in which you can embed objects such as bitmaps.

You can also create a drawing with no sheet format.

### To select a sheet format:

1. Click **New**  on the Standard toolbar.
2. Select **Drawing** , then click **OK**.
3. Select from the following, then click **OK**. If you click **None**, you create a drawing with no sheet format.


- **Standard sheet size.** Select a standard sheet size.
- **Display sheet format** (available for standard sheet sizes).
- **Custom sheet size.** Specify a **Width** and a **Height**.


 To select a different sheet format in an existing drawing, click **Format**.

## Saving Sheet Formats

### To save the sheet format:

1. Click **File, Save Sheet Format**.
2. Edit **File name**.
3. Click **Save**.

 You can overwrite standard formats or create custom formats. Sheet format files have extension **.slddrt** and are located in `<install_dir>\data`.

 Custom properties in the document are saved with the sheet format and added to any new documents that use the format.


Related Topics

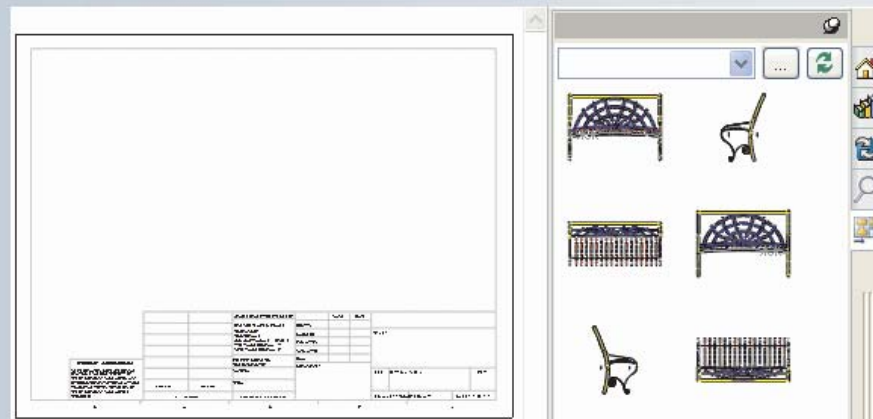


Contents Index Search


- ? View Palette
- ? PhotoWorks Items
- ? Preview Window
- ? 3D ContentCentral
- Status Bar
- Display
- SolidWorks Fundamentals
- SolidWorks Options
- Sketching
- 2D to 3D Conversion
- Moving from AutoCAD
- 2D Emulator
  - ? Overview of 2D Comman
  - ? View Toggle
  - ? Abbreviations
  - ? Repeating Commands
  - ? Transparent Commands
  - ? Function Key Toggles
  - ? Command Options
  - ? Selecting Objects
  - ? Specifying Coordinates
  - ? Previews
  - ? PropertyManager
- Drawing Entity Creation
- Other Drawing Tools
- View Tools
- Entity Properties
- Information
- Feature Creation Tools
- System Tools
- Reference Geometry
- Features
- Parts
- Assemblies
- Configurations
- SolidWorks Toolbox
- Drawings
  - ? Drawings Overview
  - Getting Started in Drawin

## View Palette

Use the View Palette , located in the **Task Pane**, to insert drawing views. It contains images of standard views, annotation views, section views, and flat patterns (sheet metal parts) of the selected model. You can drag views onto the drawing sheet to create a drawing view.



The View Palette populates when you:

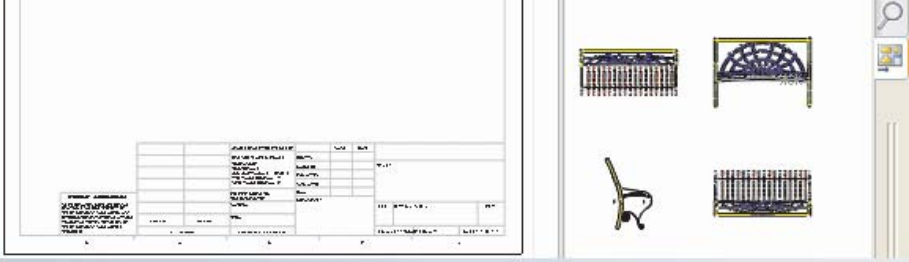
- Click **Make Drawing from Part/Assembly** 
- Browse to a document from the View Palette.
- Select from a list of open documents in the View Palette.

### To insert drawing views from the View Palette:


In a drawing with no parts or assemblies open:

Contents Index Search

- View Palette
- PhotoWorks Items
- Preview Window
- 3D ContentCentral
- Status Bar
- Display
- SolidWorks Fundamentals
- SolidWorks Options
- Sketching
- 2D to 3D Conversion
- Moving from AutoCAD
- 2D Emulator
  - Overview of 2D Commands
  - View Toggle
  - Abbreviations
  - Repeating Commands
  - Transparent Commands
  - Function Key Toggles
  - Command Options
  - Selecting Objects
  - Specifying Coordinates
  - Previews
  - PropertyManager
- Drawing Entity Creation
- Other Drawing Tools
- View Tools
- Entity Properties
- Information
- Feature Creation Tools
- System Tools
- Reference Geometry
- Features
- Parts
- Assemblies
- Configurations
- SolidWorks Toolbox
- Drawings
  - Drawings Overview
  - Getting Started in Drawing




The View Palette populates when you:

- Click **Make Drawing from Part/Assembly** 
- Browse to a document from the View Palette.
- Select from a list of open documents in the View Palette.



**To insert drawing views from the View Palette:**

In a drawing with no parts or assemblies open:

1. Click **Browse**  in the View Palette.
2. Browse to a part or assembly, then click **Open**.
3. Drag a view into the graphics area.

In a drawing with parts or assemblies open:

1. Select the part or assembly that you want to insert from the list in the upper left corner of the View Palette.
2. Drag a view into the graphics area.

 If you delete a view from a drawing, you can re-insert the view from the View Palette by clicking **Refresh**  at the top of the palette.

- [-] SolidWorks Options
- [-] Sketching
- [-] 2D to 3D Conversion
- [-] Moving from AutoCAD
- [-] 2D Emulator
  - [?] Overview of 2D Comman
  - [?] View Toggle
  - [?] Abbreviations
  - [?] Repeating Commands
  - [?] Transparent Commands
  - [?] Function Key Toggles
  - [?] Command Options
  - [?] Selecting Objects
  - [?] Specifying Coordinates
  - [?] Previews
  - [?] PropertyManager
- [-] Drawing Entity Creation
- [-] Other Drawing Tools
- [-] View Tools
- [-] Entity Properties
- [-] Information
- [-] Feature Creation Tools
- [-] System Tools
- [-] Reference Geometry
- [-] Features
- [-] Parts
- [-] Assemblies
- [-] Configurations
- [-] SolidWorks Toolbox
- [-] Drawings
  - [?] Drawings Overview
  - [-] Getting Started in Drawin
    - [?] Setting Options for Dr
    - [?] Creating a Drawing
    - [?] Sheet Format/Size
    - [?] The Drawing Window
    - [?] Sheet Formats, Sheet:
    - [?] Customizing Sheet Fo
    - [?] Saving Sheet Formats
    - [?] Sheet Properties

## Saving Sheet Formats

### To save the sheet format:

1. Click **File, Save Sheet Format**.
2. Edit **File name**.
3. Click **Save**.

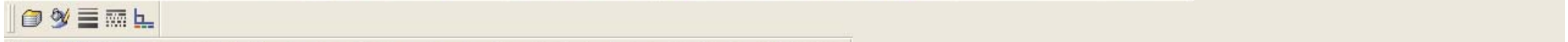


You can overwrite standard formats or create custom formats. Sheet format files have extension **.slddrt** and are located in `<install_dir>\data`.



Custom properties in the document are saved with the sheet format and added to any new documents that use the format.


Related Topics



Part1

- Blocks
- Design Binder
  - Design Journal.doc <Empty>
  - Annotations
  - Sheet1

### About SolidWorks

 SolidWorks® Office Professional 2007  
SP3.1  
Serial No.: 0000 0019 2970 5947 PFBZ KJGK

Copyright © 1995-2007 Dassault Systemes  
U.S. Patents 5,815,154; 6,219,049; 6,219,055; 6,603,486; 6,611,725; 6,844,877;  
6,898,560; and 6,906,712; and certain other foreign patents, including EP 1,116,190 and JP  
3,517,643.

Portions of this software are copyrighted by and are the property of UGS Corp. ©  
2007.  
Portions of this software © 1997-2002 Macromedia, Inc.  
Portions of this software © 1990-2007 D-Cubed Limited.  
Portions of this software © 1998-2007 Geometric Software Solutions Co. Limited.  
Portions of this software © 1995-2004 Spatial Corporation.  
Portions of this software © 1996-2007 Microsoft Corporation. All rights reserved.  
Portions of this software © 1999-2004 Viewpoint Corporation.  
Portions of this software © 1997-2007 Tech Soft America.  
Portions of this software © 1997-2007, Structural Research & Analysis Corporation.

License Agreement...    Adobe Copyrights...

Click Connect to visit the SolidWorks web site and learn more about the  
Company and Product.    Connect...

OK