Assembly reports
Assembly reports
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Welcome to self paced training for Solid Edge. This course is designed to educate you in the use of Solid Edge. The course is self-paced and contains instruction followed by activities.

**Solid Edge self-paced courses**

- **spse01510**—Sketching
- **spse01515**—Constructing base features
- **spse01520**—Moving and rotating faces
- **spse01525**—Working with face relationships
- **spse01530**—Constructing treatment features
- **spse01535**—Constructing procedural features
- **spse01536**—Modeling synchronous and ordered features
- **spse01540**—Modeling assemblies
- **spse01541**—Explode-Render-Animate
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- **spse01655**—Revising assemblies
- **spse01660**—Assembly reports
- **spse01665**—Replacing parts in an assembly
Lesson 1  

Introduction

- spse01670—Designing in the context of an assembly
- spse01675—Assembly features
- spse01680—Inspecting assemblies
- spse01685—Alternate assemblies
- spse01690—Virtual components in assemblies
- spse01695—XpresRoute (tubing)
- spse01696—Creating a Wire Harness with Harness Design
- spse01424—Working with Solid Edge Embedded Client

Start with the tutorials

Self-paced training begins where tutorials end. Tutorials are the quickest way for you to become familiar with the basics of using Solid Edge. If you do not have any experience with Solid Edge, please start by working through the tutorials for basic part modeling and editing before starting this self-paced training.
Lesson

2  Assembly reports
Assembly reports

Types of assembly reports

Solid Edge creates reports that provide information about the components that make up an assembly. You can create:

• Assembly reports—For assembly and structural frame models, assembly reports list the parts and subassemblies in the model. You can generate these types of reports:
  – Bill of Materials—This report supports item number levels that match the assembly structure.
  – Exploded Bill of Materials
  – Summary of Atomic Reports
  – Parts List—This report is a flat list that shows the quantity of each element.

• Pipe reports—For pipe and tubing models, lists the pipes and fittings in an assembly.

• Harness reports—For wire harness models, lists the harness components and connections in an assembly. You can create the following types of wire reports:
  – Top Level
  – Expanded

You can create assembly reports for managed documents and unmanaged documents.

• For unmanaged documents, the report information is collected from the local file properties of the referenced part or assembly documents.

• For managed documents, the property information is collected from the list of properties defined in the SharePoint managed document profile, not the local file properties.

Producing assembly bill of material reports

To produce an assembly bill of materials, you can run the Reports command from the Solid Edge Assembly environment, from Revision Manager, or from Windows Explorer. In Explorer, the Reports command is available on the shortcut menu of an assembly document (.asm).

Note

The Reports command is not available in Windows Explorer for assemblies that are in a managed library. To create a report for a managed assembly, you must open the assembly in Solid Edge or Revision Manager.
Producing pipe reports

For pipe parts, you can use the Pipe Report command in XpresRoute to create a report for the pipes and fittings in an assembly. This displays the Piping Report dialog box for you to specify report type and content:

- Report all the pipes in the assembly.
- Report only the currently selected pipes.
- Include pipe fittings.
- Create a total length or cut length report.

The Pipe Report command is not available if there are no pipes in the file.

Producing harness reports

For wire harness parts, you can run the Harness Reports command from Harness Design to create a report for the components and connections in an assembly.

When you run the Harness Reports command in Harness Design, you can specify which harness elements to include in the report:

- All the harness components (or connections) in the assembly
- Only the currently selected components (or connections)
- Only the currently displayed components (or connections)

Note

- To create a report on selected parts, select the parts before selecting the Reports command.
- You can use the Show and Hide commands to control which harness elements are displayed.

Weldment assembly reports

You can use the Reports dialog box to specify whether a weldment assembly is treated as a single component, similar to a part; or as an assembly, where all its component parts are included in the report.

When you set the Expand Weldment Assemblies option, the component parts in the assembly are included in the report. When you clear this option, the weldment assembly is treated as a single part, and the component parts are excluded from the report.

Alternate assembly reports

If the assembly has been converted to an alternate assembly, in Solid Edge the report is created for the active member. In Windows Explorer and Revision Manager you can use the FOA Member Names dialog box to specify which member you want to create a report for.
Virtual components

If the assembly for which you are creating a report contains virtual components, use the Reports command on the Tools menu in the Assembly environment to generate the report. Teamcenter-managed virtual components are included in the report.

Caution

If you run the Reports command from Windows Explorer on an assembly that has virtual components, the virtual components will not be contained in the report. In a case where the assembly contains only virtual components, a message may be displayed that states that no parts are in the file.

Showing assembly item numbers in reports

You can show assembly item numbers in assembly and structural frame reports, and in pipe reports. Assembly item numbers are saved with the assembly and are available for downstream use, for example, in balloons placed on a model and in ballooned parts lists placed on an assembly drawing. This ensures consistent part numbering is propagated across all applications of the model.

- To create item numbers in the assembly, select the Maintain item numbers check box in the Item Numbers page (Solid Edge Options dialog box).

- To use these item numbers in assembly reports, select the Use assembly generated item numbers check box, which is available in the Report dialog box.

Alternatively, you can leave this option unchecked and have the assembly Report command generate item numbers on the fly.

Formatting a report

You can format the report using the Format option on the Reports dialog box. Using the Format Report dialog box, you can set the following:

- Font

- Column headings (based on available properties)

- Sorting method

Each property you include in the report will be a new report column. Standard properties you can choose from include Quantity, Document Number, Revision, and Author. You can also include custom file properties that you have recorded in the part and subassembly documents.

After you have formatted the report, you can preview the report output by selecting the OK button on the Format Report dialog box.
Outputting the report

After previewing the report, you have the following options:

• Save the current report.
• Print the current report.
• Copy the current report to the Clipboard.
• Create another report.

Saving a report

The Save As option allows you to define a storage location, document name, and output type. You can output the report as a text file (.txt) or rich text file (.rtf).

Adding properties to managed documents

If you add new custom properties to managed Solid Edge documents, you must also add these properties to the managed document profile on the SharePoint server to make them available in an assembly report. Managed document profiles are stored in the Document Profiles folder on the SharePoint server. To properly synchronize the Solid Edge file properties and the SharePoint managed document profile properties, the property names must match exactly.
Item numbers in assemblies

Item numbers for parts and subassemblies can be created and maintained automatically in an assembly model. Item numbers are based on the assembly structure, that is, the order in which parts, subassemblies, and assemblies are displayed in Assembly PathFinder.

Once created, you can use assembly item numbers in assembly reports, including pipe and tubing reports produced in XpresRoute, and structural frame design reports. The item numbers are saved with the assembly and are available for downstream use, for example, in PMI balloons placed on the model and in ballooned parts lists on an assembly drawing. This ensures consistent part numbering is propagated across all applications of the model.

Creating item numbers in assemblies

You can create item numbers in an assembly model by selecting the Maintain item numbers check box in the Item Numbers page (Solid Edge Options dialog box). As new items are added to the assembly, new item numbers are created. If items are removed from the assembly, their item numbers are not available for reuse.

You can remove item numbers from the assembly by deselecting the Maintain item numbers check box.

Item number creation can be enabled and disabled at any time.

- Item numbers can be generated as the assembly is built by enabling item number creation before items are added to the assembly.

- In some cases, it may be better to enable item numbers after most of the assembly structure has been created.

Editing item numbers

You can edit item numbers in the model by selecting the top level assembly occurrence in PathFinder and choosing the Occurrence Properties command. The Item Number column in the Occurrences Properties dialog box shows the assembly item numbers. You can:

- Change existing item numbers.

- Add missing item numbers using the Next Available Number command on the shortcut menu.

- Use the Reset Item Numbers command on the shortcut menu to cancel your edits and restore the item numbers generated by the assembly structure.

Using assembly item numbers in reports

You can use the assembly-generated item numbers in assembly reports when you select the Use assembly generated items number check box in the assembly Report dialog box. The item numbers are shown when you select the Format button to open the Format Report dialog box or in the Report Output dialog box.

See the help topic, Create a report.
Using assembly item numbers in PMI balloons

You can display assembly item numbers in PMI balloons placed on the model.

When you select the PMI tab→Annotation group→Balloon command, you can use these options on the Balloon command bar to retrieve the item numbers from the assembly and display them in the balloon:

- Item Number button
- Item Count button

Using assembly item numbers on a drawing

On an assembly drawing, you can:

- Review the assembly generated item numbers on the Item Number page (Parts List Properties dialog box). The numbers shown in the first column correspond to the assembly structure.

- Show the assembly generated item numbers in a ballooned parts list by selecting the Use assembly generated items number check box on the Options page (Parts List Properties dialog box).
Lesson 2  Assembly reports

**Item numbering schema**

Use the Item Numbers page (Solid Edge Options dialog box) to select an item numbering schema for the parts and subassemblies in the assembly model. Refer to the following table for a comparison of item numbering options.

**Example**

The gray shaded cells below indicate the assembly structure, where A1 and A2 are subassemblies, and P1, P2, P3, P4, and P5 are parts.

The unshaded cells show how flat list item numbering is applied to the assembly structure.

The last column shows the level based item number option, which is available only for Exploded-Top down item numbers.

```
<table>
<thead>
<tr>
<th>Level 1</th>
<th>Level 2</th>
<th>Level 3</th>
<th>Top level only</th>
<th>Atomic Top down</th>
<th>Exploded Top down</th>
<th>Use level based numbers</th>
</tr>
</thead>
<tbody>
<tr>
<td>A1</td>
<td></td>
<td></td>
<td>1</td>
<td>1</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>P1</td>
<td></td>
<td>1</td>
<td>2</td>
<td>1.1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>P2</td>
<td></td>
<td>2</td>
<td>3</td>
<td>1.2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>A2</td>
<td></td>
<td>4</td>
<td>1.3</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>P3</td>
<td>3</td>
<td>5</td>
<td>1.3.1</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>P4</td>
<td>4</td>
<td>6</td>
<td>1.3.2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>P6</td>
<td></td>
<td>5</td>
<td>7</td>
<td>1.4</td>
<td></td>
</tr>
<tr>
<td></td>
<td>P7</td>
<td></td>
<td>6</td>
<td>8</td>
<td>1.5</td>
<td></td>
</tr>
<tr>
<td>P2</td>
<td></td>
<td></td>
<td>2</td>
<td>2</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>P4</td>
<td></td>
<td>3</td>
<td>4</td>
<td>6</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>A2</td>
<td></td>
<td>4</td>
<td>4</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>P3</td>
<td>3</td>
<td>5</td>
<td>4.1</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>P4</td>
<td>4</td>
<td>6</td>
<td>4.2</td>
<td></td>
</tr>
<tr>
<td>P5</td>
<td></td>
<td></td>
<td>5</td>
<td>7</td>
<td>9</td>
<td>5</td>
</tr>
</tbody>
</table>
```

**Formatting item numbers**

You can format the item numbers shown in the assembly model by selecting the following options on the Item Numbers page (Solid Edge Options dialog box):

- Use level based numbers
- Start number
- Increment by
Exploded parts lists

You can use the Parts List command and options in the Parts List Properties dialog box to define and place an exploded parts list on an assembly drawing.

Use the Options page to:

• Show the item numbers that were created in the assembly.

Using item numbers from the model ensures the parts list item numbers do not change unless the model does. Otherwise, item numbers generated on-the-fly by the Parts List command.

**Note**

You can use the assembly model item numbers when the Maintain item numbers check box is selected in the assembly document on the Item Numbers page, Solid Edge Options dialog box. To learn more, see Help topic [Item numbers in assemblies](#).

Use the List Control page to:

• Display subassemblies and subassembly parts in an exploded parts list.

• Choose the item numbering format:
  – Level based item numbers, which indicate the hierarchy of an exploded parts list.
  – Flat list item numbers.

• Show the top level assembly in a row by itself.

Use the Columns page to:

• Select and format the Item Number data column.

• Select additional data columns—Mass (Item), Mass (Quantity), Miter Cut 1, and Miter Cut 2—when generating parts lists for assembly models containing frames, pipes, or tubes.

• Indent the item numbers or the content of any column.

Use the Sorting page to:

• Sort the item numbers in the same order that they are shown in Assembly PathFinder.

Instead of defining a custom exploded parts list style, you can select the default Solid Edge exploded parts list style from the Saved Settings list on the General page (Parts List Properties dialog box) or on the Parts List command bar.
Activity: Assembly reports

Overview

The objective of this activity is to demonstrate some of the options when generating an assembly report.

Activity

In this activity you will use different options to create assembly reports.

Turn to Appendix A for the activity.
A Activity: Generating a report in an assembly

Overview
This activity demonstrates to create and format a report in assembly.

Objectives
You will create different reports based on different options.
Open an existing assembly with all the parts active

The assembly you will open has assemblies and subassemblies with parts at all levels.

- From the Solid Edge start screen, click Open Existing Document. Browse for top.asm in the folder where the activity files are located.
Generate a bill of materials report

- Expand pathfinder to show all the parts and subassemblies.

- On the Tools tab in the Assistants group, click the Reports command.
Select the options shown and then click format.

- In the bill of materials format dialog box, click options.
- Use the add and remove buttons set the options as shown and click OK.
- The report is generated. Observe the results. Save the report as rich text format as `bom.rtf` in the folder where the activity files are located.
Generate an exploded bill of materials report

- Click New Report to generate an exploded bill of materials report.
- Select the options shown and then click format.

![Report dialog box]

- In the bill of materials format dialog box, click options.
- Use the add and remove buttons set the options as shown and click OK.

![Options dialog box]

- The report is generated. Observe the results. Save the report as rich text format as `exploded_bom.rtf` in the folder where the activity files are located.
Generate a summary of atomic parts report

- Click New Report to generate a summary of atomic parts report.
- Select the options shown and then click format.

![Image of report window]

- In the bill of materials format dialog box, click options.
- Use the add and remove buttons set the options as shown and click OK.

![Image of options window]

- The report is generated. Observe the results. Save the report as rich text format as atomic.rtf in the folder where the activity files are located.
Generate a parts list report

- Click New Report to generate a parts list report.
- Select the options shown and then click format.

![Report dialog box]

- In the bill of materials format dialog box, click options.
- Use the add and remove buttons set the options as shown and click OK.

![Options dialog box]

- The report is generated. Observe the results. Save the report as rich text format as parts_list.rtf in the folder where the activity files are located.
- Click Close
Assigning top level item numbers in an assembly

Item numbers can be assigned at the assembly level and used downstream in reports and on a drawing sheet.

- Click the Solid Edge application button.
- Click Solid Edge Options, then click the item numbers tab.
- Set the options as shown.

![Options](image)

**Note**

Item numbers are determined by the assembly pathfinder order. Item numbers can be controlled by editing the occurrence properties in pathfinder.

- Click apply then click ok to dismiss the Solid Edge Options.
Activity: Generating a report in an assembly

- To view the occurrence properties for the top level of the assembly, right click `top.asm` in pathfinder, then click occurrence properties.

<table>
<thead>
<tr>
<th>Placement Name</th>
<th>Item Num...</th>
</tr>
</thead>
<tbody>
<tr>
<td>top.asm</td>
<td></td>
</tr>
<tr>
<td>Part1.par:1</td>
<td>1</td>
</tr>
<tr>
<td>A1.asm:1</td>
<td>2</td>
</tr>
<tr>
<td>hanger.par:1</td>
<td></td>
</tr>
<tr>
<td>Part2.par:1</td>
<td></td>
</tr>
<tr>
<td>Part4.par:1</td>
<td>4</td>
</tr>
<tr>
<td>Part3.par:1</td>
<td></td>
</tr>
<tr>
<td>a2.asm:1</td>
<td>3</td>
</tr>
<tr>
<td>hanger.par:1</td>
<td></td>
</tr>
<tr>
<td>Part4.par:1</td>
<td></td>
</tr>
<tr>
<td>Post.par:1</td>
<td></td>
</tr>
<tr>
<td>Part5.par:1</td>
<td></td>
</tr>
<tr>
<td>a3.asm:1</td>
<td></td>
</tr>
<tr>
<td>hanger.par:1</td>
<td></td>
</tr>
<tr>
<td>Part6.par:1</td>
<td></td>
</tr>
<tr>
<td>Part7.par:1</td>
<td></td>
</tr>
<tr>
<td>Part8.par:1</td>
<td></td>
</tr>
<tr>
<td>Part2.par:1</td>
<td>4</td>
</tr>
</tbody>
</table>

- Click OK.

- On the PMI tab in the annotation group, click the balloon command 📌

- Set the balloon shape to be circular 🗓️

- Ensure that the item number and item count buttons are selected as shown.

- Click the balloon lock dimension plane 🗼
- Select the front reference plane as shown.
• Annotate the parts as shown. Placement is approximate.

• Click Save to save the assembly.
Placing the PMI dimensions on a drawing sheet.

- Click the application button.
- Click New→ISO draft.
- On the home tab in the drawing views group, click the view wizard command.
- Select the assembly *top.asm* and then click open.
- Click Next.
- Select the front view, click next and then click finish.
- Place the view on the drawing sheet as shown.

- On the home tab, in the dimensions group, select the retrieve dimensions command.
Activity: Generating a report in an assembly

- Select the view. The balloons created in the assembly are placed on the drawing sheet with the corresponding item numbers.

- Save the draft document as top_level.dft.
- Close the draft file. You will be returned to the assembly environment.
- Close the assembly.
Assigning Atomic top down item numbers in an assembly

- Open the assembly top_atomic.asm with all the parts active.

- Click the Solid Edge application button.

- Click Solid Edge Options, then click the item numbers tab.

- Set the options as shown.

- Click apply then click ok to dismiss the Solid Edge Options.

- On the PMI tab in the annotation group, click the balloon command.

- Set the balloon shape to be circular.

- Ensure that the item number and item count buttons are selected as shown.

- Click the balloon lock dimension plane.
Select the front reference plane as shown.
Activity: Generating a report in an assembly

- Annotate the parts as shown. Placement is approximate.

- Click Save to save the assembly.

- An optional step is to create a drawing sheet as you did for the top level item numbers and place the front view and the balloons on the drawing sheet.

- Close the assembly.
Assigning Atomic top down item numbers in an assembly

- Open the assembly *top_explode.asm* with all the parts active.

- Click the Solid Edge application button

- Click Solid Edge Options, then click the item numbers tab.

- Set the options as shown.

- Click apply then click OK to dismiss the Solid Edge Options.

- On the PMI tab in the annotation group, click the balloon command

- Set the balloon shape to be circular

- Ensure that the item number and item count buttons are selected as shown.

- Click the balloon lock dimension plane
• Select the front reference plane as shown.
Activity: Generating a report in an assembly

- Annotate the parts as shown. Placement is approximate.

- Click Save to save the assembly.

- An optional step is to create a drawing sheet as you did for the top level item numbers and place the front view and the balloons on the drawing sheet.

- Save and close the assembly. This completes the activity.
Summary

In this activity you learned some of the options available for generating assembly reports.