



Suggested Technique for Using Envelopes to Create Simplified Representations

Procedure

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To make large assemblies more manageable, a tool called an envelope can be used to create geometry to represent selected components in the assembly. An envelope is a special type of part which contains its own geometry and a list of components that it represents. A new part file, with a ".prt" extension is generated when an envelope is created. Envelopes can be substituted into a simplified representation, so that only the envelope's geometry is visible, in place of the original geometry. The three major techniques involving envelopes are:

- [to make surface copies of the selected components](#)
- [to create simple solid features to represent the more complex selected components](#)

The use of shrinkwrap features will not be covered in this document. For more information on using simplified representations, refer to [Suggested Technique for Managing the Display of Assemblies Using Simplified Representations](#). To demonstrate the use of envelopes, `lighter.asm`, which is shown in Figure 1, will be used.

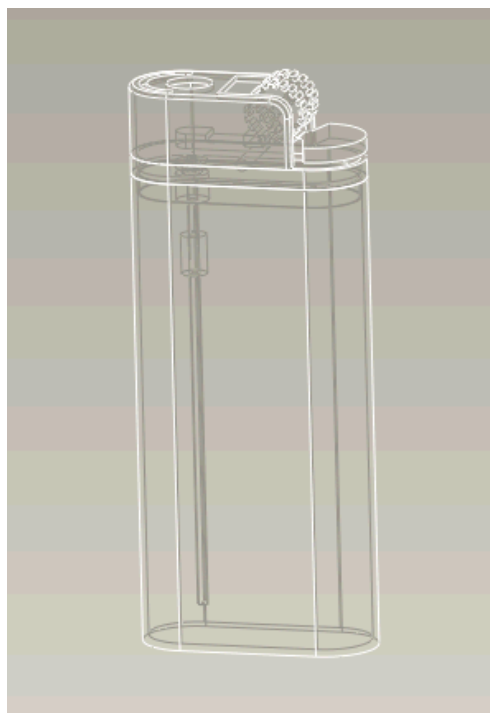


Figure 1

1. The [first technique](#) involving envelopes is to create surface copies of one or more components. The `flint_wheel` component on the top of the lighter has a very large number of features due to the radial pattern of grooves. Copying a few of the surfaces of the `flint_wheel` and using these surfaces in place of the actual component would reduce regeneration time without dramatically altering the appearance of the assembly.

Select View > Envelope Manager > New, and enter the name "wheel", pressing ENTER afterwards. Keep the default selection Include and pick "wheel," as shown in Figure 2, and select OK

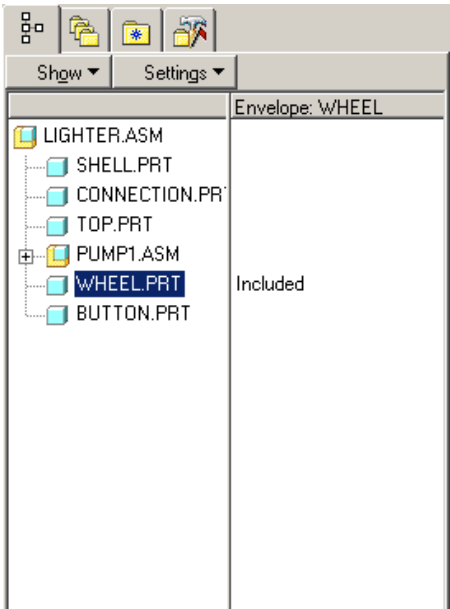


Figure 2

2. There are four options available when the Envelope Method dialog box opens. A new component can be created by selecting Create Envelope Part, an existing component can be used by choosing Select Existing Assembly Component, or a shrinkwrap feature can be created or used by picking either of the two shrinkwrap options. Using an existing component will simply represent the components that were selected to be members of the envelope with the existing component. In this example, click Create Envelope Part and name the part file which is generated "wheel_simp". The Creation Options menu will appear, and with it, the ability to create any solid, surface, or datum feature to represent the members. Select Create features > OK from the Creation Options dialog box. The wheel_simp part should activate. To create the surfaces select the wheel part, then one of the outer flat surfaces in wheel.prt and click Edit > Copy > Edit > Paste. While holding down CTRL, select the second outer surface and the cylindrical surface in the middle of the flint_wheel as shown in Figure 3. Then click the green checkmark in the Copy tool dashboard. Surface copies can only reference one part at a time. If there were other members being represented by this envelope, and additional surface features were required, the Geometry button could be chosen from the Envelope Definition dialog box, then repeat the process from above, selecting different surfaces this time. To finish the editing of the envelope click OK from the Envelope Definition dialog box. Select the envelope name and then click the Info button and notice that it lists the component that will be substituted and mentions that this envelope is not being used in any Simplified Reps. Close the Information window. To view the envelope, highlight the envelope name and right-click and select Show References. Click Close from the ENVELOPE dialog box to complete creation of the envelope.

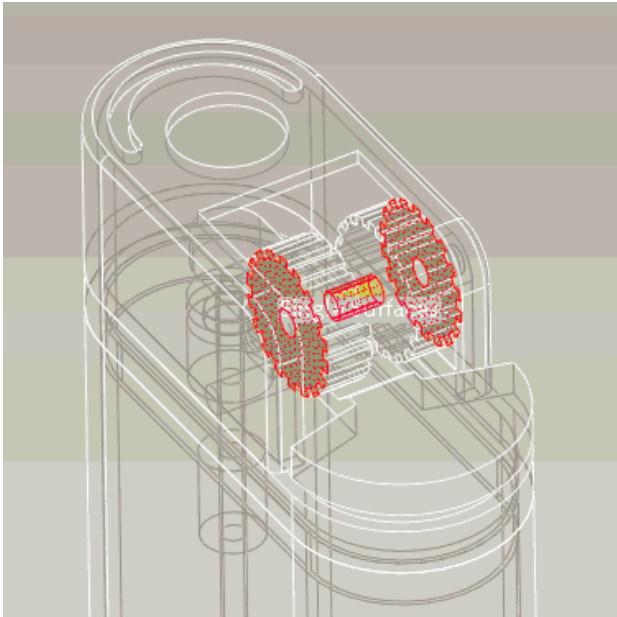


Figure 3

3. Now that the envelope has been created, it needs to be utilized in order to simplify the assembly. Click View > View Manager, and in the Simp Rep tab create a new simplified representation. Click New and name the representation "Surf_Wheel" and press ENTER. The default rule should be set to Master Rep in the Include tab. In the Substitute tab, click By Envelope from the METHOD menu, and pick "wheel" from the Select Envelope(s) section. This will automatically replace the component wheel.prt with the envelope WHEEL_SIMP as shown in Figure 4. Accept the specified changes and exit the EDIT menu. Close the View Manager. An alternate method to creating the simplified representation is to simply click View > Substitute > Envelope and double-click on WHEEL in the Select Envelope dialog

box. Then click View > View Manager and under the Simp Rep tab click New and enter "surf_wheel" and press ENTER.

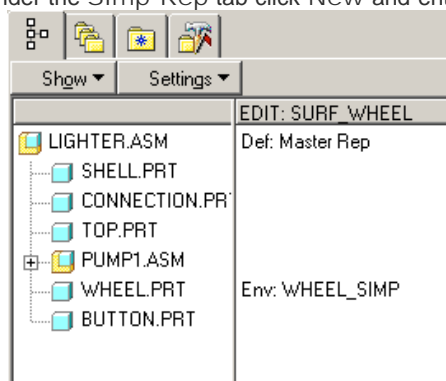


Figure 4

4. The results of substituting the "wheel" envelope can be seen in Figure 5.

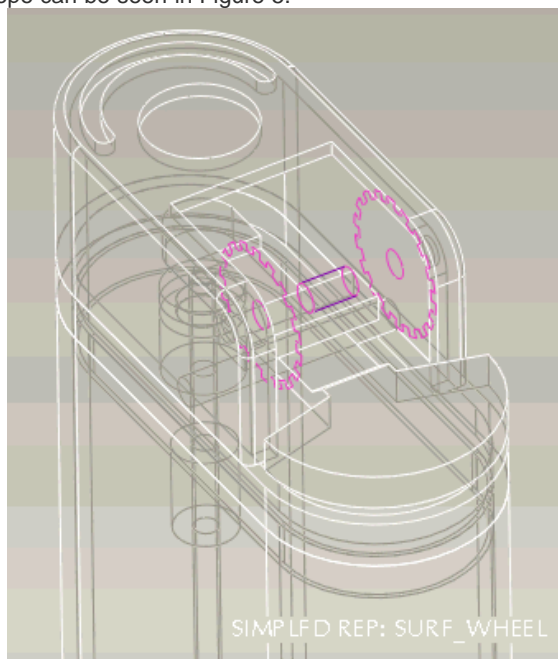


Figure 5

5. The [second major technique](#) involving envelopes is to create some very simple geometry to represent one or more complex parts. The front part of the lighter contains many small parts, as shown in Figure 6. The display of the assembly would be simpler if those parts were represented by one cylindrical protrusion. Click View > Envelope Manager > New and create an envelope called "tubes". This envelope will be substituted for the small components in a simplified representation.

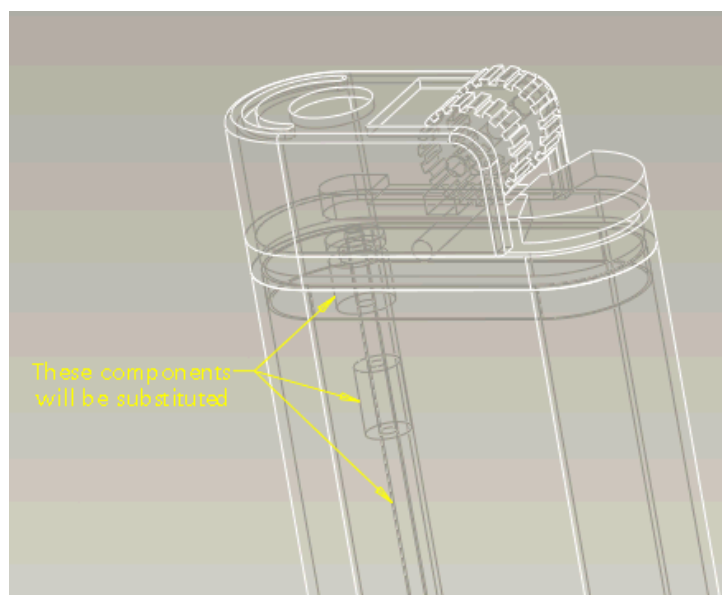


Figure 6

6. From the Model Tree, select `pump1.asm`, and expand it. With Include highlighted pick `pump_tube1.prt`, `pump_tube2.prt`, `rod.prt`, as shown in Figure 7. Note that in Figure 7 the component `pump1.asm` is not listed as Included. If `pump1.asm` is listed as Included, select the Exclude button and pick this component in the Model Tree. These three highlighted components will be represented by the envelope. Click Envelope Part and name the part file "tubes", then click OK.

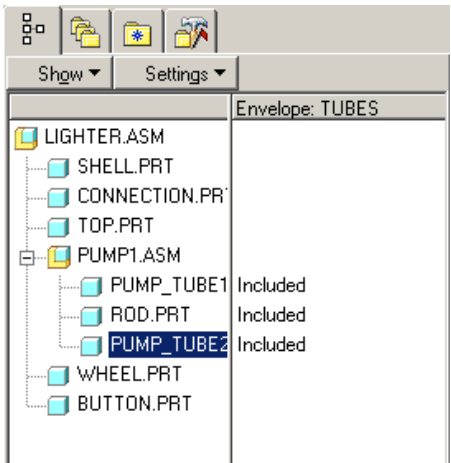


Figure 7

7. Select the Create Features radio button and OK from the Creation Options dialog box. Click Insert > Extrude > Placement > Define. Select DTM1 from `button.prt` as the sketching plane and RIGHT as the sketching reference with a Right orientation, as shown in Figure 8. Point the arrow downwards for the direction of feature creation.

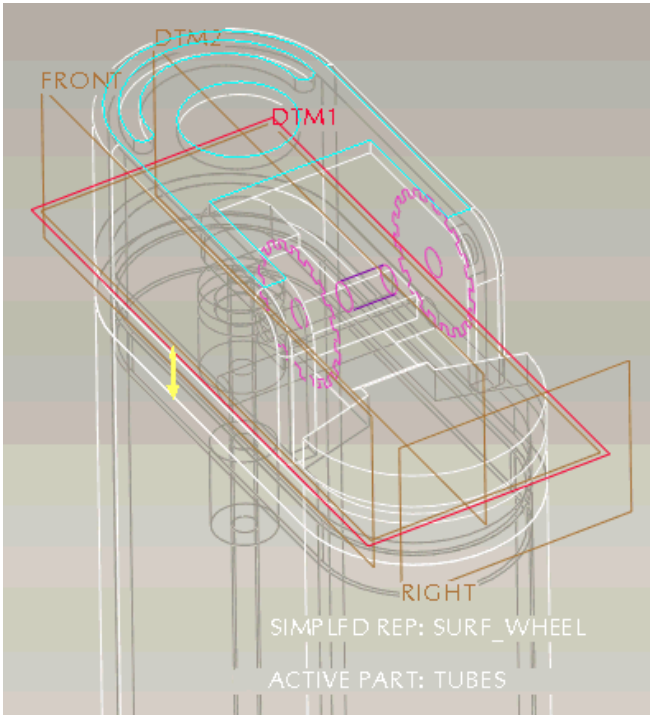


Figure 8

8. Create the sketch that is shown in Figure 9, selecting edges of `pump_tube.prt` using the Sketch > Edge > Use tool. Give the feature a Blind depth of 2.00, and apply and save the changes in this feature and close the tool dashboard. Click OK from the Envelope Definition dialog box, then select TUBES in the envelope list and click Info. Notice the component in this envelope will substitute. Close the Information window. This will complete the creation of the "tubes" envelope. Click Close to exit the ENVELOPE dialog box.

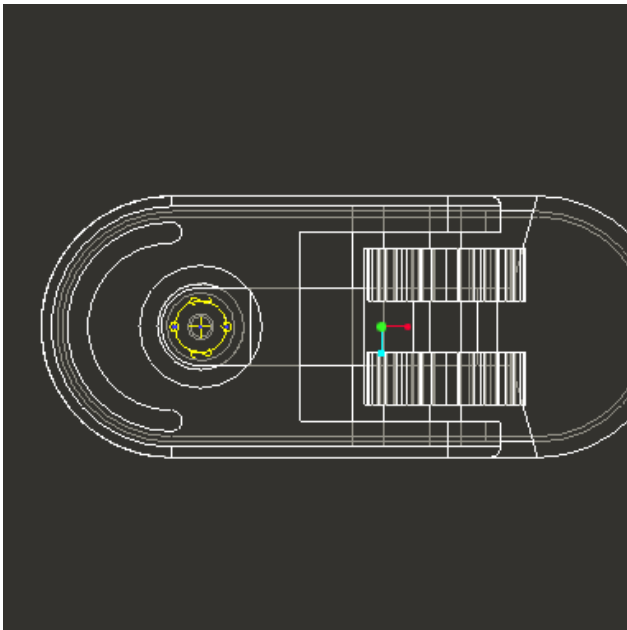
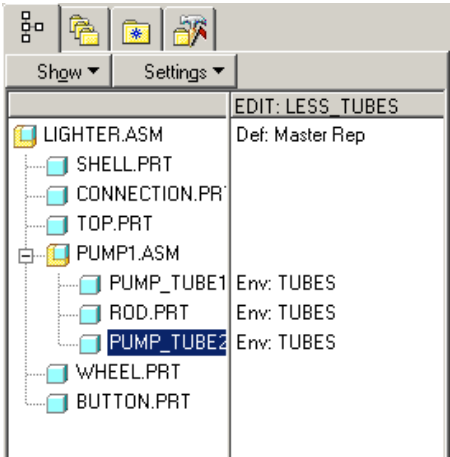


Figure 9

9. Click View > View Manager, and in the SIMP REP tab create a new simplified representation. Click New and name the representation "Less_Tubes" and press ENTER. Master Rep should automatically be set as the default rule under the Include tab. In the Substitute tab, pick By Envelope from the METHOD menu, and pick "tubes" from the SELECT ENVELOPE menu. This will automatically replace the components selected in Step 7 with the envelope "tubes," as shown in Figure 10. If the envelope "wheel_simp" appears with an eye symbol next to the name, then select "wheel_simp" again until the eye no longer appears. Accept the specified changes, exit the EDIT menu and close the View Manager. An alternate method to creating the simplified representation is to simply click View > Substitute > Envelope and double-click on TUBES in the Select Envelope dialog box. Then click View > View Manager and under the Simp Rep tab click New and enter "surf_wheel" and press ENTER.



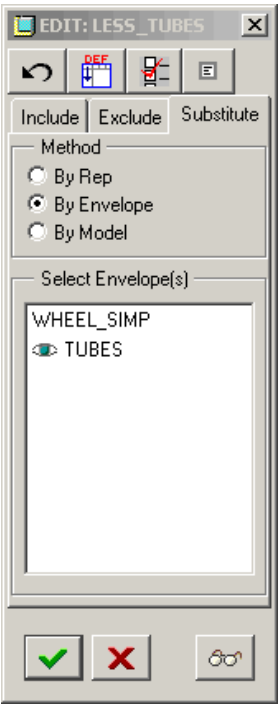


Figure 10

10. The results of substituting the "tubes" envelope can be seen in Figure 11.

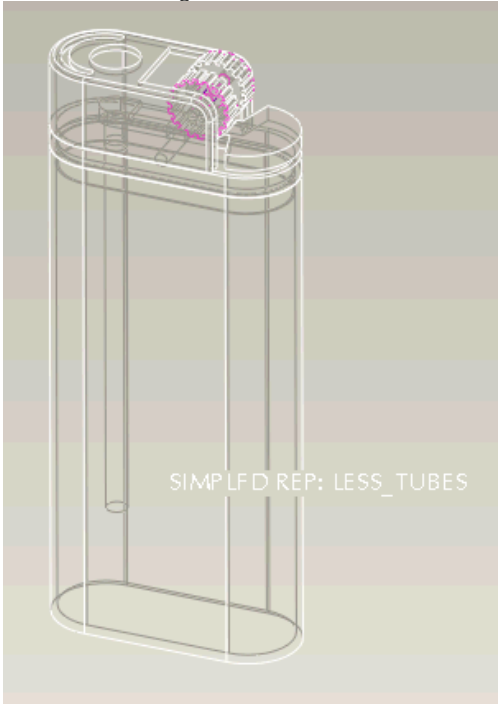


Figure 11

[Download Finished Files for this Technique](#)