

# How to sweep a sleeve along a wire or cable in Creo Elements/Direct Modeling

**Product:** Creo Elements/ Direct Modeling, Cabling module

**Release:** 18.0, 18.1

## Overview

This article describes how to add a sleeve or sheath along an existing Cabling conductor.

## Description

Periodically a protective wrapping or sleeve must be applied to a cabling conductor for visualization and placement purposes. However, with the current Cabling module, you cannot automatically add geometry along the length of an existing conductor. This article shows how to manually construct a sleeve along an existing conductor using its 3D Curve (Fig. 1).

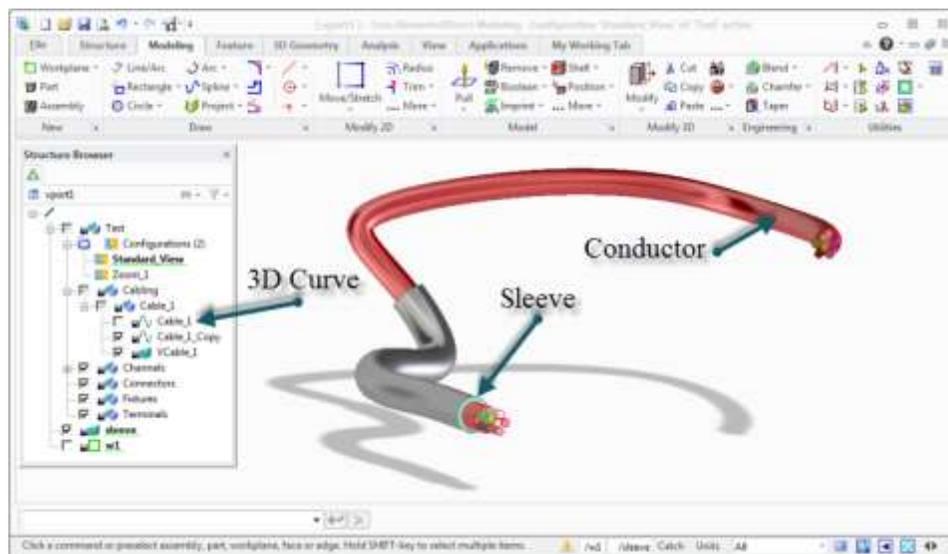


Figure 1. A sleeve added to a conductor using 3D Curve.

### To use the Modeling Sweep tools and the existing Cabling 3D Curve to generate a sleeve:

1. Switch from the Cabling module to Modeling.
2. Create a new Workplane at or near one end of the sleeve.
  - a. Click **Workplane > Project Construction** geometry
  - b. Select the end face.
3. Create the sleeve profile geometry. For example, in Fig. 2, the profile was created at the end point and then modified after the sleeve was created. We did this to add a small gap distance from the end as shown in Fig. 1 above.

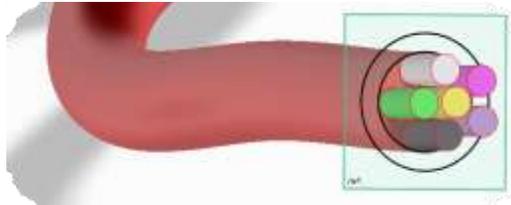


Figure 2. Sleeve profile with gap.

4. Make a copy of the existing 3D Curve for the Cabling conductor as follows: In the **Structure** tab, in the **Part & Assembly** group, click **Copy** (Fig. 3).
5. From the Structure Browser, select the 3D Curve. Or disable the display of the Conductor, and select from the viewport. Then click OK to complete the command.

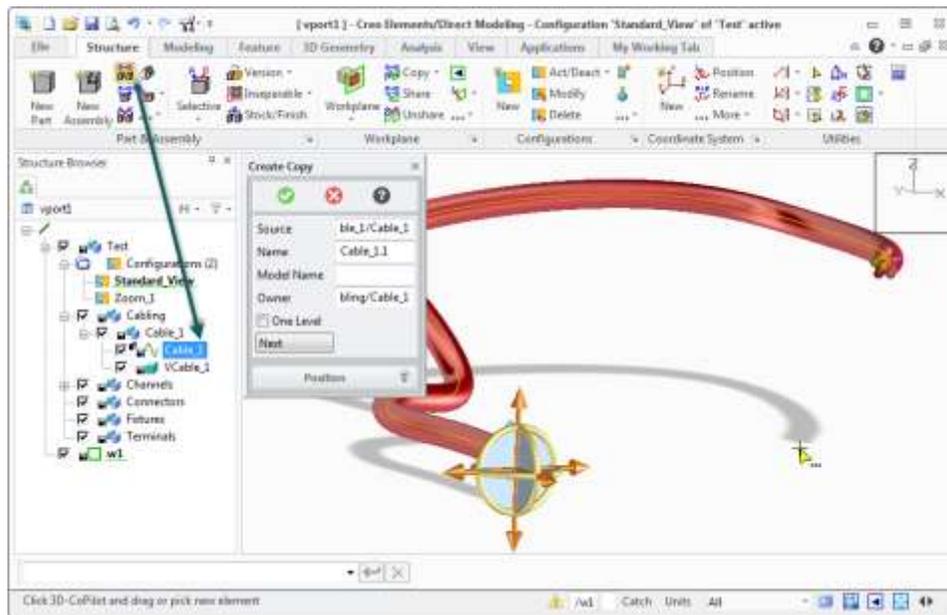


Figure 3. Select cable and create a copy.

6. Split the new 3D Curve to define the end point of the new sleeve by clicking the **3D Geometry** tab. Then click **3D Curve > More > Split Edge**.
7. Select the copy of the **3D Curve** from the viewport (not from the Structure Browser)
8. Select a point along the curve to be the end point for the sleeve. Notice that you can now mouse over the two different sections and they highlight (preselect) independently.

Note: Selecting the 3D Curve from the Structure Browser results in an error: *"String "/..." does not correspond to item of desired type"*

9. Sweep the sleeve profile along the specified 3D Curve section by clicking the **Modeling** tab. Then in the **Model** group, click **More > Sweep Add**.
10. Type a new part name (e.g., "Sleeve").
11. Mouse over and select the 3D curve segment from the viewport for the spine, and then click OK to complete the command.

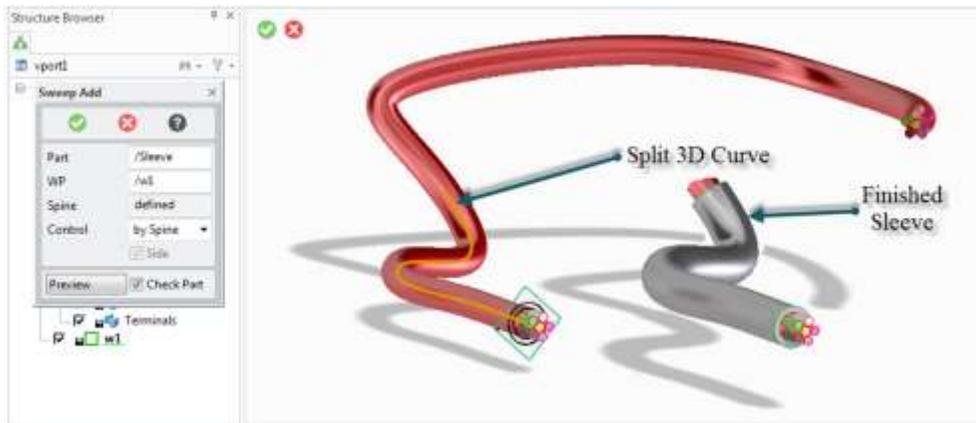


Figure 4. Finished sleeve.

See also an [online video demonstrating](#) the process (no sound).