

[Home \(/support\)](#) > [Support \(/support\)](#)

This is a PDF version of Article CS155622 and may be out of date. For the latest version click [here](https://www.ptc.com/en/support/article?n=CS155622&language=en&posno=2&q=harness%20part%20units&source=search) (https://www.ptc.com/en/support/article?n=CS155622&language=en&posno=2&q=harness part units&source=search)

Article - CS155622

How to assign units to a harness part in Creo Parametric and Creo Elements/Pro 5.0

Created: 19-Dec-2013 | Modified: 20-Apr-2018

Applies To

- Creo Parametric 1.0 to 5.0
- Pro/ENGINEER and Creo Elements/Pro Wildfire 5.0

Description

- How to assign units to a harness part.
- Unable to change the units of an harness model.

Resolution

In **Creo Elements/Pro 5.0**:

- You can assign Length units with **Edit > Setup > Units** after activating the Harness in Cabling application.

In **Creo Parametric 2.0** and later releases:

- You can assign Length units with **Setup > Units** menu after activating the Harness from Cabling tab in ribbons.
- Also you may change the system of units activating the Cabling assembly:
 1. **File > Prepare > Model Properties**, then change **Units**.
 2. **Set** a System of Units and tick the "**Include Submodels**" box to propagate to harness.

[Legal Policy \(/documents/policies\)](#)