AVENG - Core update to Creo 12.0 - Special: CZ Standards and Tools
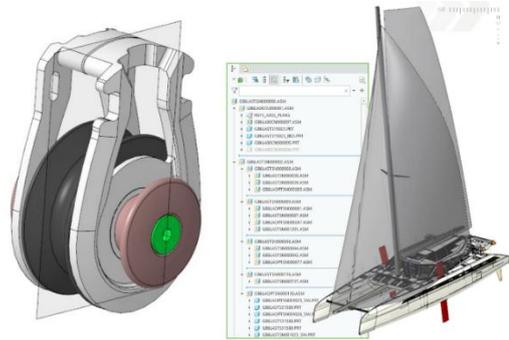
Training Code: **S-CPUPD12**

Duration: **1 to 3 days**

Target Audience

The training is designed for all users working with an older version of Creo who are transitioning to Creo 12.0.

It is intended for participants who have completed the following courses: Creo Parametric Basic Training, Creo Parametric Drawing Training, Creo Parametric Advanced Modeling Training and Creo Parametric Basic Surface Training.

Course Content

- Introduction to the modernized user interface of Creo 12.0.

- Simplified work with the Model Tree to understand the model structure through Snapshots and Design Items organized into groups.

- New features in the Sketcher environment – sketch verification, offset, entity trimming, enhanced curve editing options, new sketching tools, and more.

- New features in 3D modeling – new settings and feature presets for reuse.

- Surface modeling – new functions, improved curve and transition handling, greater flexibility in surface extension, offsetting multiple surfaces simultaneously, and more.

- Modernized Sheetmetal environment – improved creation of Flat Walls, multibody in Sheetmetal, enhanced mirroring, and improved wall connection options.

- Enhanced Multibody functionality across all Creo Parametric modules.

- New features in Assembly mode – overview and evaluation of regeneration performance, improvements in working with simplified representations.

- Intuitive creation and editing in Model-Based Definition, simpler selection of reference surfaces, table creation and formatting, new options for exporting 3D drawings.

- New features in Drawing mode – symbol gallery in drawing documentation, creation and editing of hatch styles directly in drawings.

- Updates in specialized modules.

- Tips and tricks, best practices, how to do things right the first time, and insights from real-world projects.

Objective

- The objective of the training is to master the transition to the new version of the Creo Parametric CAD system with its updated user interface, focusing primarily on part modeling, assembly creation, and drawing documentation.