

PTC CREO

11 November 2020



Agenda

09:00 – 09:30	Welcome
09:30 - 12:30	<p>Creo Parametric Capabilities - (3D Part and Assembly Design, Drawings, Manufacturing)</p> <p>Workshop 01 - Fundamentals</p> <ul style="list-style-type: none"> Creo Design Process; Understanding Solid Modeling Concepts – Exercise <p>Workshop 02 - Assembly Design</p> <ul style="list-style-type: none"> Overview Assembly Design, Standard & Mechanism Constrains - Exercise <p>Workshop 03 - Drawings</p> <ul style="list-style-type: none"> View Types, View Properties, Driving vs. Driven Dimension – Exercise Create Drawings using Drawing Template – Exercise Assembly Drawings, Table, Bill of Materials – Exercise
12:30 – 13:00	Quick Lunch
13:00 – 17:00	<p>Workshop 04 - Manufacturing</p> <ul style="list-style-type: none"> Overview Manufacturing Process – Exercise <p>Workshop 05 - About the Multibody – Creo 7.0</p> <ul style="list-style-type: none"> Creating the multibody parts – Exercise