AXE :	1
TITRE :	*CATIA V5 (Base)*
DUREE :	5 days
PUBLIC CIBLE et PRE-REQUIS :	New CATIA V5 Mechanical Designers Mechanical design experience, Experience with the Windows operating system
OBJECTIFS :	- Understand the CATIA V5 interface - Plan the construction of a part in order to properly convey its visual and functional aspects - Create simple parts in CATIA V5 - Understand how to manage parts in the context of an assembly - Produce simple drawings and assembly layouts
CONTENU :	This course will introduce you to CATIA V5. It will teach you how to build simple parts and assemblies in CATIA, and how to make simple drawings of those parts and assemblies. - Introduction to CATIA - Profile Creation - Basic Features - Additional Features - Dress-Up Features - Reusing Data - Finalizing Design Intent - Assembly Design - Design in Context - Drafting ISO - Drafting ANSI