

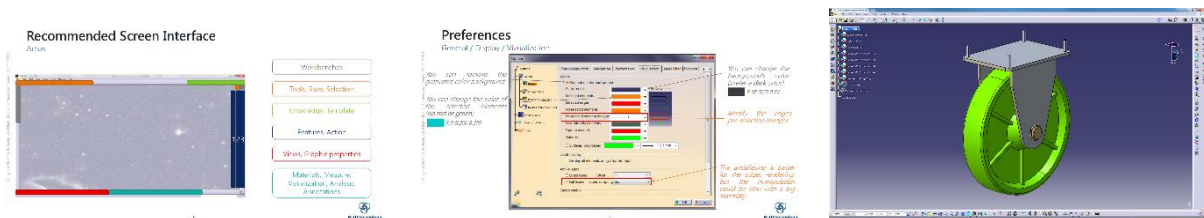
CATIA Essentials course (3 days): introduction to CATIA – global overview of Part, Assembly and Drawings

Introduction:

This course will give you a flying start with CATIA. It's perfect for beginners and CAD-Engineers who are looking to quickly become efficient in the CATIA environment. You'll get an introduction to: Part Design, Assembly Design, Drawings and System Customization

After 3 days, you will know best-practices for how to design parts, assemblies and drawings within your own CATIA environment.

We will also work together to help you answering your questions about your own projects.



Day 1

Morning (9:00-12:00): CATIA Essentials – Basics (glossary, interface, manipulation and concepts); CATIA Essentials – Introduction to the CATIA environment (Managing data, quick customization and useful shortcuts).

Afternoon (13:00-17:00): CATIA Essentials – Settings (set up the screen interface for each workbenches, customize CATIA and set the preferences); Introduction to Part Design – Basics (create simple solids and use basic commands).

Day 2

Morning (9:00-12:00): Introduction to Part Design – Basics (create more complex solid and use advanced commands like Boolean and parameters).

Afternoon (13:00-17:00): Introduction to Assembly Design – Basics (overview of the commands, understand the software's logic and learn the methodology to follow).

Day 3

Morning (9:00-12:00): Introduction to Drafting – Basics (overview of the features and set up a functional and basic drawing layout).

Afternoon (13:00-17:00): Summary exercise (reproduce a small assembly of a wagon wheel using Part, Assembly and Drawing workbenches).