FUNDAMENTALS

OBJECTIVE: Develop with Open CASCADE Technology to build, handle and view CAD models in an application

CONTENT

Day 1
Introduction
✓ General presentation
✓ Documentation and samples

DRAW
✓ Presentation & Functionalities
✓ Definition of new commands

Handles
✓ Notion & use of handles
✓ Definition of a new handled class

Utilities and services
✓ Collections and exceptions
✓ Units, messages and resources

Day 2
Geometry
✓ Basic and advanced geometry
✓ Methods of construction
✓ Constraint geometry in 2D

Day 3
Topology
✓ Definition and purpose
✓ Structure of a shape
✓ Collection of shapes
✓ Exploration tools

BRep Model (Boundary Representation)
✓ Geometry and precision in BRep
✓ BRep tools overview

Day 4
Modeling algorithms
✓ Introduction
✓ Packages of modeling algorithms (BRepBuilderAPI, BRepPrimAPI, BRepAlgoAPI, BRepFilletAPI, BRepOffsetAPI)
✓ History of modifications, error handling

Features
✓ Basic Concepts
✓ Mechanical features
✓ The Gluer and the SplitShape classes

Day 5
Visualization
✓ Interactive context and object
✓ Neutral point & Local context
✓ Selection management
✓ Filters

CAD files Import/Export
✓ Import/export to STEP, IGES and BRep formats

EXERCISES
✓ Create a DRAW command
✓ Create a 2D constraint sketch
✓ 3D shape modeling
✓ Displaying

PREREQUISITES
✓ C++

DURATION, LOCATION
✓ 5 days
✓ At the Customer’s site or at the premises of OPEN CASCADE (Guyancourt, Lyon - FRANCE)

REGISTRATION
https://www.opencascade.com/contact