Develop with

Open CASCADE Technology

OPENCASCADE

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