

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

CONTENT

Preliminaries

- OCCT basic concepts
- Literature
- Performance tips
- OCCT's command line application (DRAW)
- Useful built-in DRAW commands
- Connection between DRAW and C++
- OCCT Standard types
- Smart pointers in OCCT
- Operation system tools
- Collections
- Exception mechanism of OCCT
- Message mechanism
- Progress indicator

Geometry

- OCCT's classes design idea
- Non-parametric geometry
- Parametric geometry
- Trimming concept
- Constraint geometry
- Geometric tolerance concept
- Geometric API

Topology

- Geometry limitations
- Why boundary representation?
- Topological data structure
- Boundary representation
- Precision in boundary representation
- Topological objects usage in geometric algorithms
- Boundary representation problems
- Topological primitives' construction
- Changing location of topological object
- Topological orientation concept
- Topology API
- History of modifications

Mesh

- Mesh storage
 - Mesh construction
 - Mesh manipulation
- The following exercises are available:
- Topological model meshing
 - Calculation of area occupied by mesh
 - Normals construction and saving them in mesh

TRAINING PROGRAM

Fundamentals

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Data exchange

- Mesh, b-rep, and mixed formats
- Mesh formats: STL, VRML, OBJ, and PLY
- B-rep formats: BRep, IGES, STEP

Visualization

- Visualization techniques: rasterization vs ray-tracing
- Rasterization pipeline
- OCCT visualization framework
- New presentation creation: displaying, selection
- Presentation building blocks
- Presentation customization
- Manipulator
- Animation
- Shading models
- GD&T
- Hidden line removal (optional)

Bounding volume hierarchy

- Typical use-cases
- Why BVH?
- BVH: idea, structure, implementation

EXERCISES

- Each section above contains exercises to consolidate the course material

DURATION

- 5 days

LOCATION

- On-line, at the Customer's site or at the premises of Open Cascade (Guyancourt, Lyon - FRANCE)