

SHAPE HEALING: Analysis & Repair Tools

OBJECTIVE: Use Open CASCADE Technology to analyze and repair models imported from other CAD systems

CONTENT

Day 1

Introduction to Shape Healing

- ✓ Motivation and objectives
- ✓ Review of geometry and topology (curve and surface continuity, tolerance and precision)
- ✓ C++ API of Shape Healing

Common problems after import

- ✓ Orientation of 3D and parametric curves
- ✓ Coincidence of 3D and parametric curves
- ✓ Bad orientation of edges in a wire
- ✓ Bad connection of adjacent edges
- ✓ Self-intersection of edges in a wire
- ✓ Missing degenerated edges
- ✓ Lacking edges

Day 2

Common problems after import (continued)

- ✓ Healing small faces
- ✓ Healing free bounds
- ✓ Healing tolerances

Shape upgrade

- ✓ Modifying surface continuity
- ✓ Shape splitting
- ✓ Converting geometry to Bezier
- ✓ Approximating curves and surfaces
- ✓ Converting surfaces to B-Splines

EXERCISES

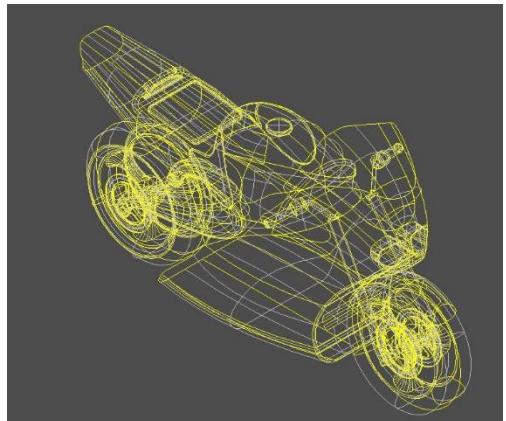
- ✓ Diagnostics and logs
- ✓ Face orientation modification
- ✓ Hole obstruction

PREREQUISITES

- ✓ C++
 - ✓ Basic OCCT concepts
- (OPEN CASCADE Fundamentals training module)

DURATION, LOCATION

- ✓ 2 days
- ✓ At the Customer's site or at the premises of OPEN CASCADE (Guyancourt, Lyon - FRANCE)



REGISTRATION

<https://www.opencascade.com/contact>