

# TRAINING PROGRAM

*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 Healign*

- 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*

- 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

- Diagnostic and logs
- Face orientation modification
- Hole obstruction

## PREREQUISITES

- C++
- Basic OCCT concepts
- (Open CASCADE Technology Fundamentals training module)

## DURATION, LOCATION

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