BETA
SIMULATION SOLUTIONS

physics on screen

ANSA for Durability analyses pre-processing

Training	ANSA for durability
Duration	2 days (16 hours)
Level	Advanced
Who should attend	CAE analysts who perform durability analyses and have experience with ANSA.
Training description and objectives	This course introduces participants to the principles of pre-processing with ANSA for durability analyses, so that they become familiar with the relevant ANSA tools and techniques, and able to prepare a ready-to-run model for ABAQUS, NASTRAN, ANSYS or PERMAS. Upon course completion participants will be able to : – Manage large models, – generate and improve triangular surface mesh and unstructured tetra volume mesh for durability applications, – assemble the model, – set up load case for solution, – use tools related to durability applications such as pretension sections, – create reduced files in order to minimize the set up time and the analysis complexity, – obtain model information and generate reports, – output ready to be solved files for durability.
Prerequisites	Basic knowledge of durability principles and ANSA is necessary.
Suggestions	This course can be combined with any of the META for Durability trainings (“META basics for Durability analyses post-processing”, “Advanced post-processing with META for Durability analyses”). Participants should have followed the “Introduction to pre-processing with ANSA” training.

BETA CAE Systems International AG

D4 Business Village Luzern, Platz 4
CH-6039 Root D4, Switzerland

T +41 415453650
F +41 415453651

ansa@beta-cae.com
www.beta-cae.com

Language	English, German **ask for more languages*

Suggested topics
Day 1
– Introduction – model management – Surface meshing generation and improvement – Volume meshing generation and improvement – Shell – solid refinement techniques – Assembly tools (1d elements – spider elements – bolts – connections – seam welds – templates- connectors) – Solver files I/O – Materials and properties handling
Day 2
– Sets handling – Contacts creation – Pretension creation – Boundary and loads definition – ABAQUS step manager – NASTRAN header – ANSYS load case manager – PERMAS situations – Transformations – Model checks – Reporting – Includes management and configurations – Renumbering – Model cut – sub structuring – Results mapping – examples

Course content is subject to change without notice.
Course content may be adjusted to audience requirements or background.