Simpleware FE
Release Version U-2022.12

December 5, 2022

Export Formats

- FE Output: the FE exports can include nodes, hexahedral, tetrahedral and shell elements, boundary layer prism elements, material properties, contact surfaces and node sets
  - Abaqus (*.inp)
  - Adina (*.in)
  - Ansys (*.ans)
  - Ansys Workbench (*.inp, preferred)
  - Ansys Workbench (*.cdb)
  - COMSOL Multiphysics (*.nas, recommended)
  - COMSOL Multiphysics (*.mphtxt)
  - COMSOL Multiphysics (*.mph, unsupported)
  - LS-DYNA (*.key)
  - Nastran (*.nas)
  - Patran (*.out)
  - VTK (*.vtk)
  - MATLAB (*.mat)
- CFD Output: the CFD exports can include nodes, hexahedral and tetrahedral elements, boundary layer prism elements, material type and boundary conditions
  - FLUENT (*.msh)
  - Abaqus (*.inp)
  - OpenFOAM
  - COMSOL Multiphysics (*.nas, recommended)
  - COMSOL Multiphysics (*.mphtxt)
  - COMSOL Multiphysics (*.mph, unsupported)
  - VTK (*.vtk)
  - MATLAB (*.mat)
- Option to store a mesh for each model in the ScanIP project file

Multipart Volumetric Meshing (options available for FE and CFD meshes)

- Mixed hexahedral/tetrahedral or pure tetrahedral element volume meshing
- Element types: Linear, straight quadratic or curved quadratic
- Choice of meshing algorithms:
  - FE Grid: image resolution-based mesh
  - FE Free: progressively decimated (or refined) mesh from the image resolution
- Advanced FE Grid options:
  - Smooth or voxel cell (pure hexahedral with unsmoothed boundaries/interfaces) mesh
  - Surface smoothing
  - Mesh quality optimization
  - Mesh adaptation
  - Smooth masks against exterior
- Advanced FE Free options:
  - Coarseness slider to automatically adjust advanced parameters relating to mesh coarseness
  - Per-part meshing settings
  - Meshing mask and surface objects in the same model (without voxelization)
  - Adaptive surface remeshing
  - Characteristic element edge lengths (min/max)

Simpleware FE generates volume and surface meshes from segmented image data, and can be used to define contact surfaces, boundary conditions and material properties.

synopsys.com/simpleware

Multipart Volumetric Meshing (options available for FE and CFD meshes) cont.

- - Surface elements constraints: surface change rate, number of elements across a layer
  - Volume elements constraints: internal change rate, number of quality optimization cycles
  - Other options: snap to parent surface, snap surface object nodes to boundaries, smooth masks against exterior, refine near slits and cavities
- Additional mesh quality improvement: User-defined minimum target element qualities. Quality metrics include:
  - In-out aspect ratio
  - Tetrahedron volume skew
  - Tetrahedron minimum dihedral angle
  - Jacobian
  - Jacobian and in-out ratio
  - Jacobian, in-out ratio and edge length aspect ratio
  - Jacobian, in-out ratio and min weighted sine of dihedral angle
- Allow element nodes to be slightly off-surface to improve mesh quality
- Optimize mesh performance for dynamic simulations by avoiding small edge lengths
- Preserve mesh periodicity at periodic faces (if any) of the geometry
- Part-wise mesh generation with smooth, conforming interfaces at boundaries
- Adaptive meshing for reduced mesh density away from boundaries
- Automated feature-based mesh refinement (FE Free)
- Boundary layer meshing: generate triangular prism cells close to solid boundaries
- Element quality statistics provided in the log history
- Mesh refinement regions: User-defined mesh density control through the use of regions of interest (ROIs) of high density and relaxation/transition zones. Available ROI options include:
  - Rectangular frustum
  - Polygonal frustum
  - Lasso frustum
  - Sphere/spheroid
  - Cube/cuboid
  - Cylinder
  - Finite plane
  - Cone
- Support for independently meshing objects (specialist option)

Multipart Volumetric Meshing (options available for FE meshes only)

- Shell elements: Shell element sets can be defined at the outer surfaces and interfaces of parts. This shell mesh can either be exported on its own or as a layer of elements encasing the volumetric mesh.
- The ROI tools can also be used to define shell element sets.
- Contact Surfaces and Node Sets: Contact surfaces and node sets can be automatically defined between any parts in contact as well as between a part and the surrounding medium and a part and the borders of the bounding box. The ROI tools can also be used to define surface and node sets. When ROIs are used for node and surface sets, the mesh can be aligned to these.
- Multi-label mesh (and contact surfaces) export available for Abaqus (*.inp)

Multipart Volumetric Meshing (options available for CFD meshes only)

- CFD Export Options—Material definitions can be set to either fluid or solid. It is possible to define any identifiable contact surface as a specific boundary type:
  - Wall
  - Symmetry
  - Inlet (velocity, pressure, mass-flow, vent or fan intake)
  - Outlet (pressure, exhaust, vent or outflow)
  - Interior boundaries (interface, porous jump, fan or radiator)
  - Generic patch (to be assigned in CFD package)
- Define high-quality arbitrary inlets/outlets for CFD using mesh clipping planes, either automatically (using centerlines) or manually
- Boundary layer meshing: generate triangular prism cells close to solid boundaries to model high fluid gradients in the viscous regions

Material Properties (volume element material properties can be set to)

- Placeholder: for later definition in FE package of choice
- Homogeneous: where the mass density, Young's Modulus and Poisson's ratio are constants defined by the user
- Greyscale: where material properties within parts can be automatically assigned from the image data, based on user-defined functions to relate greyscale value of the original data to density

Material Properties (volume element material properties can be set to) cont.

- Density: as a function of element parent voxel greyscale (GS) is given by:

  $\rho = a + (b \times GS)$

  (where a and b are user-defined variables which may be non-integer)

- Young's Modulus: as a function of density is given by:

  $E = a + (b \times \rho c)$

  (where a, b and c, are user-defined variables)

- Poisson's ratio: as a function of density is given by:

  $\mu = a$

  (where a is user-defined)

- Map to mesh:
  - Assign fiber orientation information directly to each mesh cell in a FE model
  - Average orientation tensor, eigenvector and eigenvalue data calculated for each mesh cell

Mesh Quality Inspection Tool

- Inspect FE/CFD mesh quality based on a variety of metrics
- Inspect elements/cells individually or in groups (violating the same quality metric)
- Show mesh errors (self-intersections) and/or warnings (distorted elements/cells)
- Visualize quality metric distributions via histogram plots
- Click on an entry in the quality inspection dialog to zoom into the specific element/cell
- Also pin-points location within the image data/segmentation
- User-defined thresholds on quality metrics
- Available mesh quality metrics:
  - Negative Jacobian
  - Volume mesh self-intersection
  - In-out aspect ratio
  - Edge length aspect ratio
  - Angular skew
  - Volume skew
  - Shape factor
  - Min/max dihedral angle
  - Jacobian
  - Minimum edge length
  - Orthogonal skew
  - Face area
  - Element volume
  - Boundary layer edge length ratio (for CFD meshes only)
  - Boundary layer wedge triangle face edge length ration (for CFD meshes only)

Volume Mesh Statistics

- Statistics template for Model statistics
- Count information for elements, nodes, edges, faces
- Volume statistics for elements
- Length statistics for edges
- Area statistics for faces

Volume Mesh Import and Remeshing

- Import a variety of commonly used volume mesh formats:
  - Ansys databases (*.cdb)
  - Abaqus input files (*.inp)
  - Nastran files (*.bdf, *.dat, *.nas, *.nastran)
  - VTK files (*.vtk, *.vtu)
- Adjust color and toggle visibility of individual element sets
- Position and orientate meshes using interactive tool
- Scale volume meshes
- Extract outer part surfaces of element sets for use with surface tools
- Add imported volume meshes to FE and CFD models
- Remesh imported volume meshes to different levels:
  - Leave mesh unchanged: no remeshing
  - Improve element qualities, preserve element connectivity
  - Regenerate volume mesh, preserving surface
  - Regenerate surface and volume mesh
- Add material properties: see "Material Properties: Volume element material properties can be set to" section above
- Inspect mesh quality: see "Mesh Quality Inspection Tool" section above
- Mesh export formats: see "Export Formats" section above