



## +FE Module Technical Datasheet

The +FE module is a fully integrated meshing module within ScanIP for the conversion of masks to volume meshes, contact surface definition and material property assignment as well generating rapid prototyping files.

### Mesh generation

#### » Multipart Volumetric Mesh

- › Mixed hexahedral/tetrahedral or pure tetrahedral element volume meshing
- › Choice of meshing algorithms:  
+FE-Grid and +FE-Free.
- › Part-wise mesh generation with smooth, conforming interfaces at boundaries
- › User defined minimum target element qualities
- › Option for pure hexahedral mesh, with unsmoothed boundaries/interfaces
- › Linear or quadratic (mid-side nodes) elements types available
- › Adaptive meshing for reduced mesh density away from boundaries
- › Automated feature based mesh refinement (+FE-Free)
- › Element quality statistics and histogram provided
- › User defined mesh density control through the use of spherical regions of high density and relaxation/transition zones.

#### » Material Properties: Volume element material properties can be set to:

- › Placeholder for later definition in FE package of choice
- › Homogeneous where mass density, Young's Modulus and the 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.

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

$$\rho = a + b \cdot GS$$

a, b are user defined variables which may be non-integer

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

$$E = a + b \cdot \rho^c + d \cdot \rho^e$$

a, b, c, d and e are user defined variables

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

$$\mu = a + b \cdot \rho^c + d \cdot \rho^e$$

a, b, c, d and e are user defined variables

*continued on next page*

- » Shell Mesh: Shell elements 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
- » 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.
- » 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)

## Export formats

- » FE Output: the +FE module exports can include nodes, hexahedral, tetrahedral and shell elements, material properties, contact surfaces and node sets:
  - › Abaqus
  - › Adina
  - › Ansys
  - › Ansys Workbench
  - › Comsol
  - › I-deas
  - › LS-Dyna
  - › VTK
- » CFD Output: FLUENT
- » RP Output Formats (STL files): Models for rapid prototyping can be exported as stl files (ascii or binary). These are exact replicas of the finite element models.

Visit our [website](http://www.simpleware.com) to download a fully functional 30-day free trial. Trial versions are fully supported by our technical team.



[www.simpleware.com](http://www.simpleware.com)

### About Simpleware Ltd.

Simpleware develops world-leading software solutions for the conversion of 3D image data into high-quality CAD, Rapid Prototype, CFD and Finite Element models. Simpleware customers range from international blue chip corporations to research institutes and universities world-wide.