Interface LIMIT - Marc

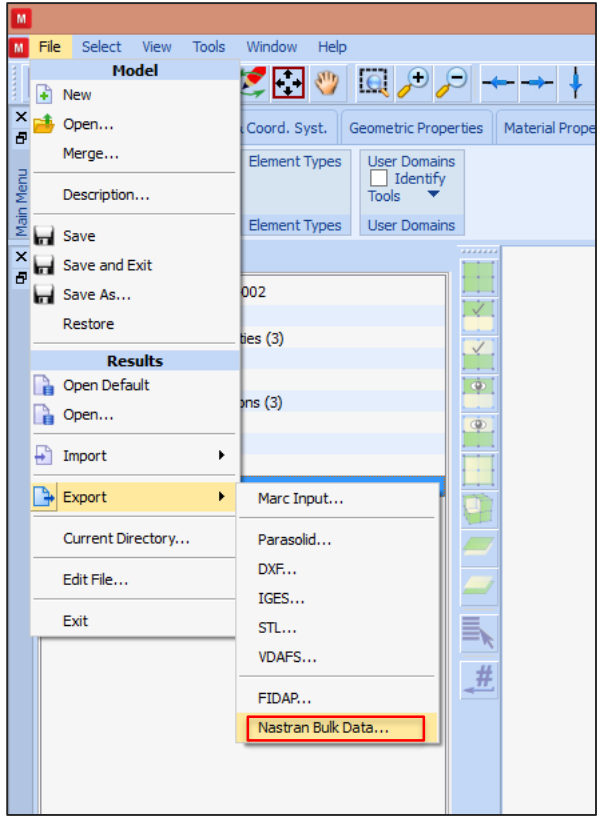
Supported Marc Versions in Release Package

- **14.2**
- **14.2 works also for newer versions.**
  - 16.0 has been successfilly tested!
  - 18.0 has been successfilly tested, but users have to change the style of the t16-file to revision 12! This is done in Mentat, while defining the results.

If you have questions please contact LIMIT support (limit@cae-sim-sol.com)

Import into LIMIT-CAE via Nastran-BDF

- In the current release the Marc-dat format can‘t be imported directly! The Finite Element model must be imported into the LIMIT-GUI using the standard Nastran Bulk Data format (.bdf).
- Exporting the BDF from Mentat is shown on right side. Export is performed directly in the first Mentat menue „File“
- Only the element topology and the nodes are needed from the Finite Element model. Thus no essential information is lost converting to the BDF-format.



Specification of the Interface

- **Maximum nodenumber respectively elementnumber :**
  - Windows 64 bit (x64): 50000000
- **Maximum number of nodes :**
  - Windows 64 bit (x64): 6000000
- **Maximum number of elements :**
  - Windows 64 bit (x64): 6000000
- **These LIMITS can be changed by the user. See document LIMIT_2020, section: *Redimensioning of Arrays***
- **Coordinate systems:**
  - Nodes must be defined in the global coordinate system
  - Result data must exist in the global system (Solids) respectively in the default element system (shells, membranes).

Supported Element Types:

- **Solids:**
  - Element 7, 84, 117, 120 (lin. Hex-elements)
  - Element 134, 157 (lin. Tet-elements)
  - Element 136 (lin. Wedge-element)
  - Element 21, 35, 57, 61 (quadr. Hex-elements)
  - Element 127, 130 (quadr. Tet-elements)
  - Element 202 (quadr. Wedge-element)
- **Shells:**
  - Element 138 (3 nodes)
  - Element 75, 139, 140 (4 nodes)
  - Element 49 (6 nodes)
  - Element 22, 72 (8 nodes)
- **Membranes:**
  - Element 158 (3 nodes)
  - Element 18 (4 nodes)
  - Element 200 (6 nodes)
  - Element 30 (8 nodes)
- **2D Solids, Axial Symmetry:**
  - Element 10, 82 (lin. Quad 4)
  - Element 28, 33 (quadr. Quad 8)
  - Element 2, 156 (lin. Tria 3)

Solid Assessment:

- **Goal of a LIMIT FKM proof of strength :**
  - Assessment of surface stresses (2D-tensors)
  - Popular method and conservative
- **Free surfaces :**
  - Are necessary for the consideration of stress gradients normal to the surface
  - Are identified by the software LIMIT
  - Can be generated by covering the solids with 2D-elements (skin) in the preprocessor.
- **2D-skin elements (membranes) can be assessed as well**
  - But without supporting effect => conservative
  - This leads to considerable less data
- **Supporting effect is only possible with solids!**
  - Results of a 3D analysis with good element quality and fine meshing are more precise than results of 2D-skin elements.

Output Format:

Using Cauchy stress:

- For large deformations the stresses must be available in a coordinate system fixed to the material point:
  - For shells this is the default
  - For solids you should use post code 391 in combination with MATERIAL>ORIENTATIONS in MARC!

Possible Postcodes Combinations for Solids:

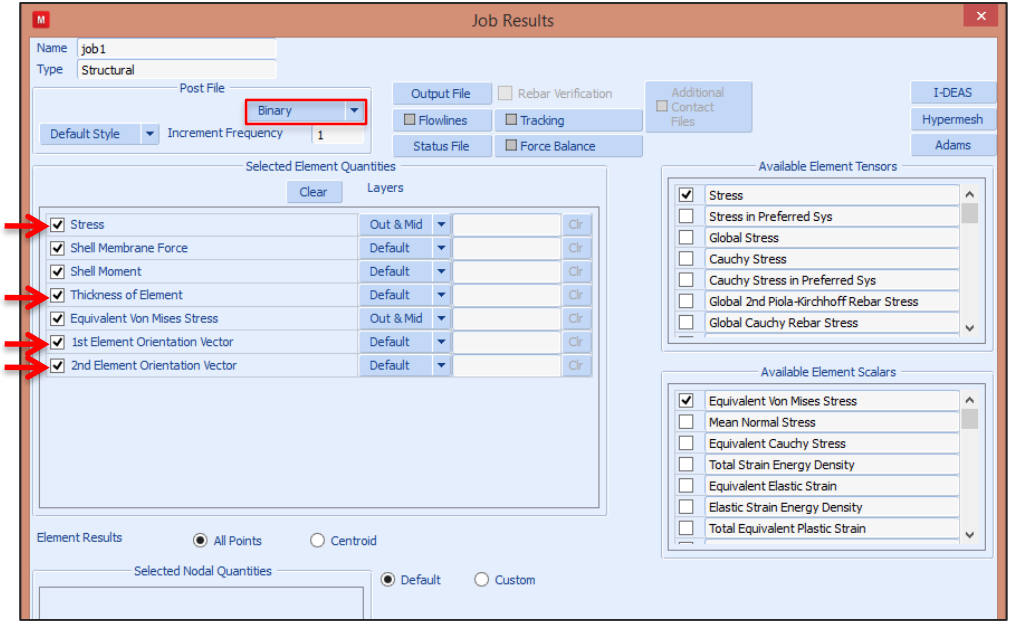
- 311 (Result type: **Stress** = stress tensor Cauchy or PKII, global coordinates, small displacements/rotations)
- 341 (Result type: **Cauchy Stress** = stress tensor Cauchy, global coordinates , small displacements/rotations)
- 391 & 691 & 694 (Result type: **Stress in Preferred System** = stresses in preferred direction tensor, Cauchy or PKII for small strain, always Cauchy stress for large strain, material orientations)

Possible Postcodes Combinations for Shells:

- 20 & 311 & 691 & 694 (= thickness, result type: **Stress =** stress tensor Cauchy or PK2, element orientation 1 & 2)
- 20 & 341 & 691 & 694 (= thickness, result type: **Cauchy Stress =** stress tensor Cauchy, element orientation 1 & 2)
- 20 & 391 & 691 & 694 (= thickness, result type: **Stress in Preferred System =** Cauchy or PK2, always Cauchy stress for large strain, element orientation 1 & 2)

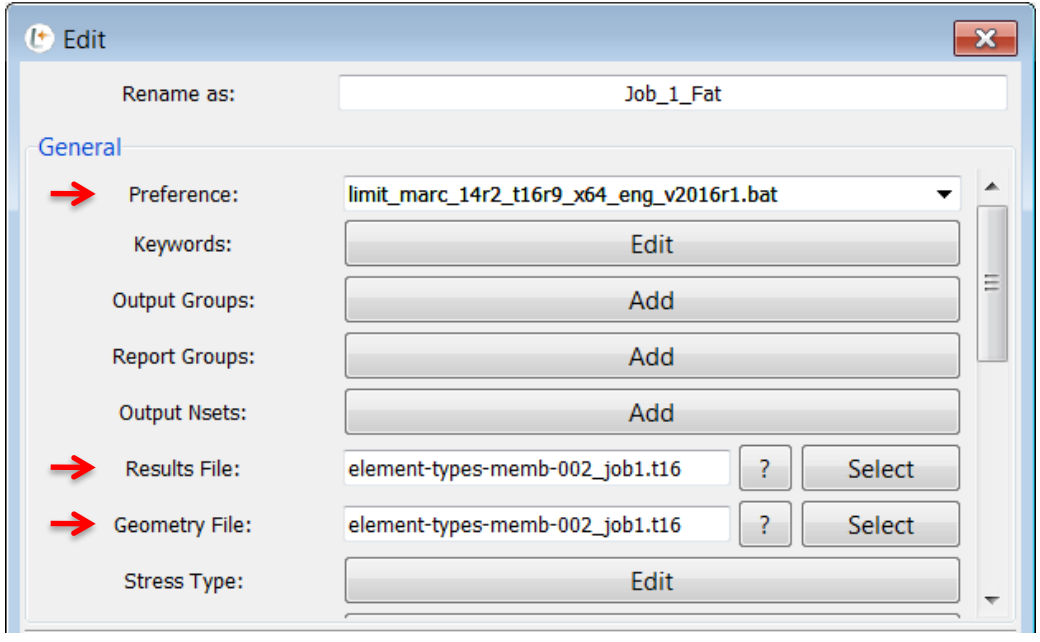
Output Format:

- The interface uses the Marc t16 binary result file
- For shell and membrane elements the local element coordinates as well as the shell thickness must be written to the t16 file. Typical settings (for all element types) in Mentat/Job Results:



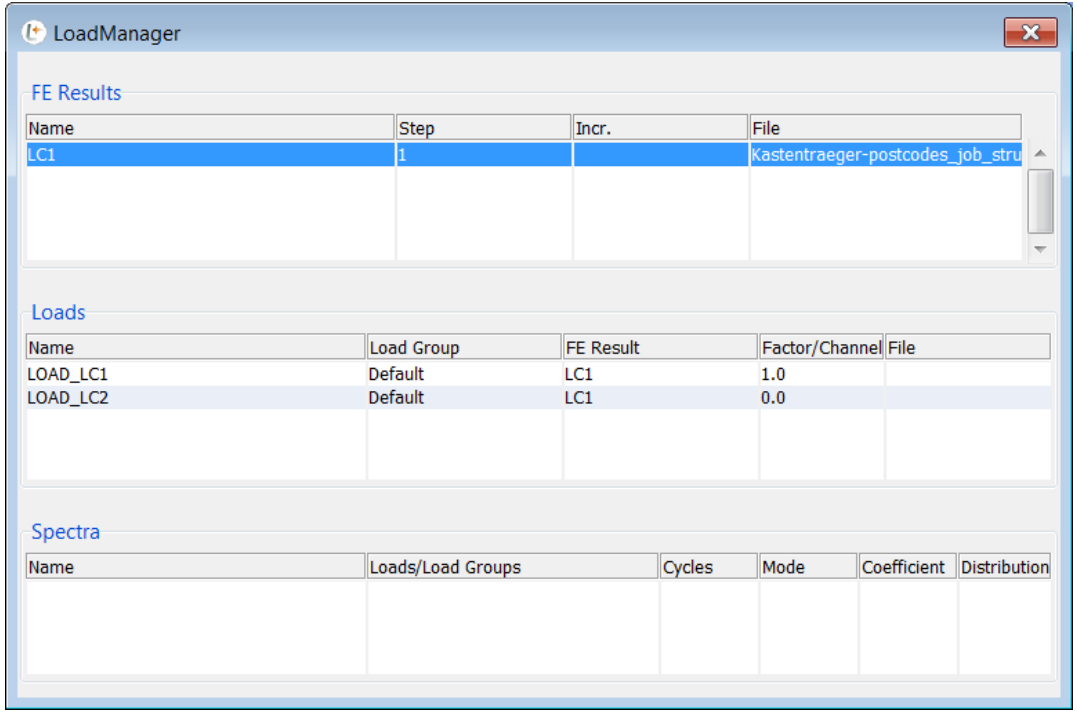
Setting Job Parameters within LIMIT:

- **LIMIT/Jobs/JobManager/Edit**
- **Preference: limit_marc_14r2…**
- **Result: [FE-job].t16**
- **Geometry: [FE-job].t16**
  Note: The FE model information (=Geometry) is taken from the t16-file



Addressing Loadcases via Step:

- **Last increment of step**
  - Step => Stepnumber
  - Incr. => remains empty



Addressing Loadcases via consecutive increment number:

- **Last increment of step**
  - Step => set to 0 (zero)
  - Incr. => consecutive number throughout all steps