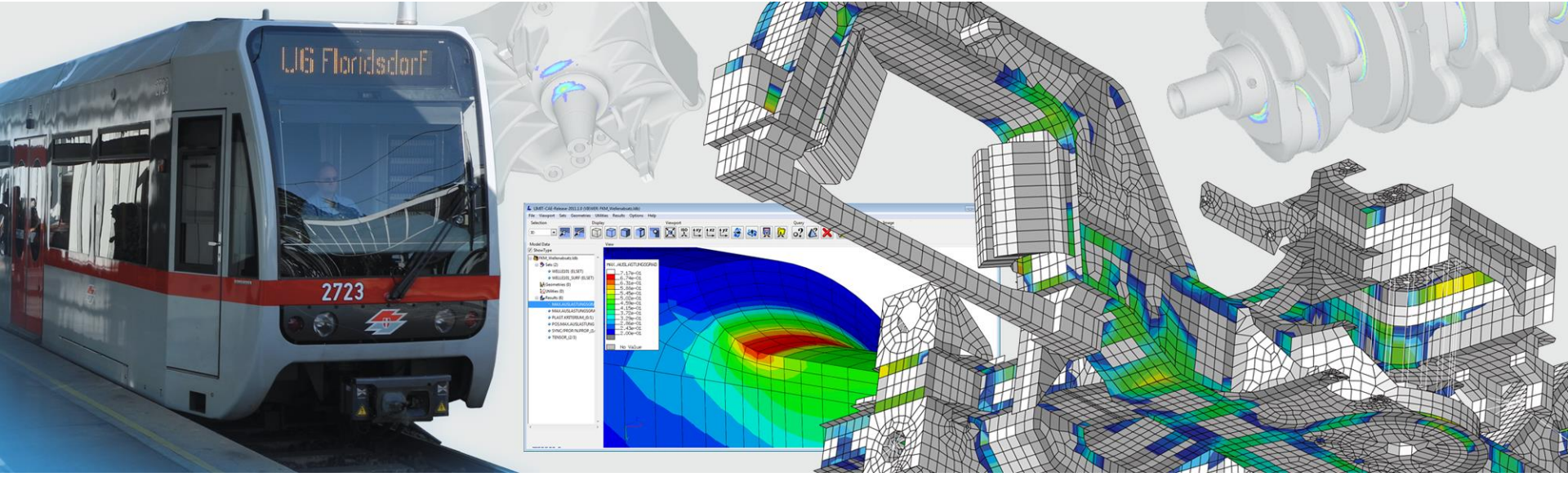

Interface LIMIT – Ansys

Supported Ansys Versions in Release Package

- **121**
- **130**
- **140**
- **145**
- **150**
- **160, 161, 162**
- **170, 171, 172**
- **180**

Note: Please make sure, that the LIMIT interface version is equal to the ANSYS version of your structural analysis. Otherwise wrong results might occure.

If you need a different version please contact LIMIT support (limit@cae-sim-sol.com)

Specification of the interface

- **Maximum nodenumber respectively elementnumber :**
  - Windows 64 bit (x64): 20000000
- **Maximum number of nodes :**
  - Windows 64 bit (x64): 3000000
- **Maximum number of elements :**
  - Windows 64 bit (x64): 4000000
- **These LIMITS can be changed by the user. See document LIMIT_2017, 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).

Following elements can be analyzed :

- **Solids:**
  - SOLID185 (lin. Hex-elements) (less suitable for stress assessment)
  - SOLID285 (lin. Tet-elements) (not suitable for stress assessment)
  - SOLID186 (quadr. Hex-elements) => stress gradiant available
  - SOLID187 (quadr. Tet-elements) => stress gradiant available
  - SOLID164 (lin. Hex-elements) (no gradients available)
  - SOLID45** (8 node Hex-element) (less suitable for stress assessment)
  - SOLID72** (4 node Tet-element) (less suitable for stress assessment)
  - SOLID73** (8 node Hex-element) (less suitable for stress assessment)
  - SOLID92** (10 node Tet-element) => stress gradiant available
  - SOLID95** (20 node Hex-element) => stress gradiant available

  *Note: ** Not available in Workbench. Can be used via geometry extraction from RST-file.*
- **Shells:**
  - SHELL93, SHELL150, SHELL281 (8 nodes)
  - SHELL43, SHELL63, SHELL181 (4 nodes)
- **Membranes:**
  - SHELL41 (4 nodes)

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 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.

Modifications for RST-Output:

- **The default RST-file can be read by the LIMIT interface**
- **Additional information on shells can be found on the next slide**

Important Information for Dealing with ANSYS SHELLS:

- For a correct weld analysis the stresses on the top and bottom side of the shells must be imported into LIMIT in the right order.
- The default shell elements in ANSYS Workbench are SHELL181 and SHELL281. These shells have different modes of stress output to the RST-File, defined by KEYOPTION(8).
  LIMIT supports only KEYOPTION(8)=0: „Store data for bottom of bottom layer and top of top layer". This is the default mode of Workbench.
- If you use these elements in APDL (ANSYS Classic) make sure the stresses are stored to the RST-file in the right order.
- All other elements have a fixed order of writing the stresses to the RST-file and no additional settings are necessary.
- The easiest way to check the stress data imported into LIMIT is shown in document: "*LIMIT-Checking-Stress-Import.pdf*".
- In case the stresses for top and bottom are inverted, the user can switch the values. See next slide: „Switching Top and Bottom Stresses During Import".

Switching Top and Bottom Stresses During Import :

- Using the JobManager/KeywordEditor the following command is needed:

```
*SWITCH_TOP_BOTTOM_STRESS_ANSYS_SHELL, OPTION
```

Valid Options are:

`ALL` … Will switch top/bottom stresses for all shell elements or

`ALL_SHELL181` … will switch top/bottom stresses for all SHELL181 elements or

`ALL_SHELL281` … will switch top/bottom stresses for all SHELL281 elements or

`ALL_SHELL43` … will switch top/bottom stresses for all SHELL43 elements or

`ALL_SHELL63` … will switch top/bottom stresses for all SHELL63 elements or

`ALL_SHELL93` … will switch top/bottom stresses for all SHELL93 elements or

`ALL_SHELL150` … will switch top/bottom stresses for all SHELL150 elements or

`ETYP#` … ELEMENT-TYPE-NUMBER: first integer in ET command: ET, 7,181 => here 7

With `ETYP#` individual regions can be switched

Examples see next page!

Switching Top and Bottom Stresses During Import

- Example 1, two lines are defined in KeywordEditor:

```
*SWITCH_TOP_BOTTOM_STRESS_ANSYS_SHELL, ALL
*SWITCH_TOP_BOTTOM_STRESS_ANSYS_SHELL, ALL_SHELL181
```

Result: All shells are being switched, all SHELL181 are being switched a second time, so they remain in their original order.

- Example 2, two lines are defined in KeywordEditor:

```
*SWITCH_TOP_BOTTOM_STRESS_ANSYS_SHELL, 7
*SWITCH_TOP_BOTTOM_STRESS_ANSYS_SHELL, 8
```

Result: All shells with ELEMENT-TYPE-NUMBER 8 or 11 are being switchee.

The ELEMENT-TYPE-NUMBER is defined with the Ansys-ET-command.

ET, 7,181 …. Element Type Number 7 uses SHELL181

ET, 8,281 …. Element Type Number 8 uses SHELL281

Last slide