



Interface LIMIT – Ansys

Supported Ansys Versions in Release Package

- ✦ 130
- ✦ 140
- ✦ 145
- ✦ 150
- ✦ 160, 161, 162
- ✦ 170, 171, 172
- ✦ 18x (version 180 can import data from all 18x sub releases)
- ✦ 19x (version 190 can import data from all 19x sub releases)
- ✦ 2019 (version 193)

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

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): 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_2019, 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 gradient available
- SOLID187 (quadr. Tet-elements) => stress gradient 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 gradient available
- SOLID95** (20 node Hex-element) => stress gradient 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 switched.
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