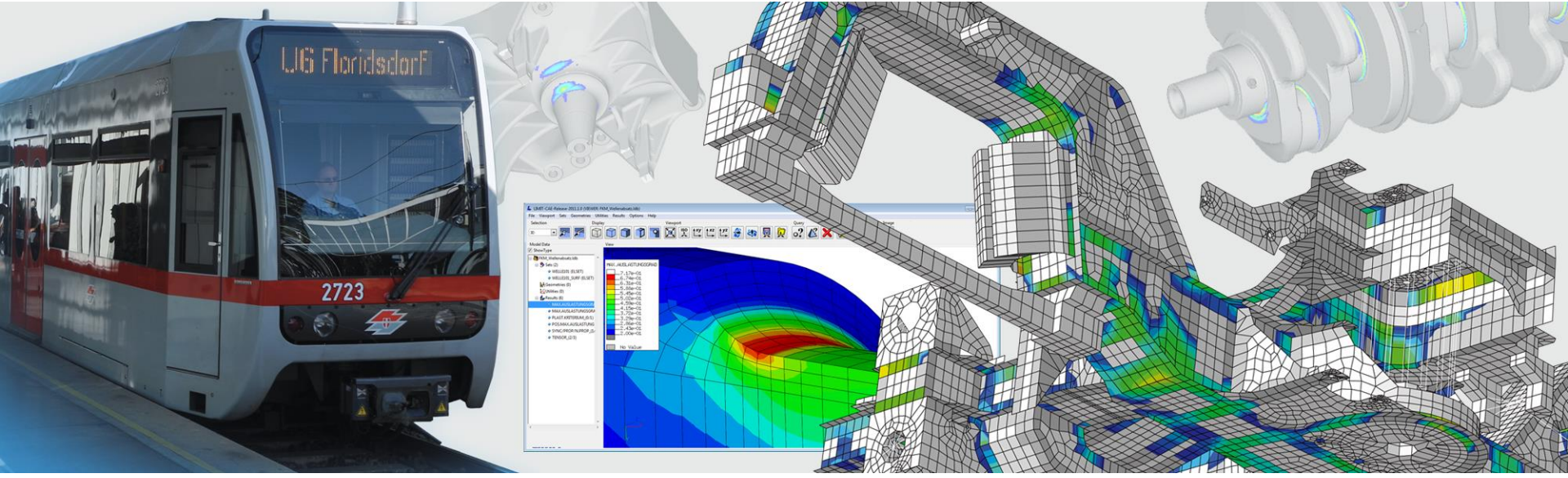



# *Interface LIMIT* – Nastran/Optistruct/Radioss

## Importing the.bdf-file into *LIMIT-CAE*:

- **File ending must be .bdf or .dat**
- **Optistruct:**
  - Export model with standard format
  - Import .fem
- **Radioss:**
  - Export model with bulk data standard format
  - Import .fem
- **Nastran (.bdf; must be ‚sorted bulk data' in the ‚short' 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_2019, section: *Redimensioning of Arrays***
- **Coordinate systems:**
  - Nodes
    - Definition in the global coordinate system
    - Definition using CORD1R or CORD2R (RECTANGULAR)
  - Result data must exist in the global system (Solids) respectively in the default element system (shells).

## Following elements can be analyzed:

- **Solids:**
  - CTETRA (4 nodes) (not suitable for stress assessment)
  - CPENTA (6 nodes) (less suitable for stress assessment)
  - CHEXA (8 nodes) (less suitable for stress assessment)
  - CTETRA (10 nodes) => stress gradiant available
  - CPENTA (15 nodes) => stress gradiant available
  - CHEXA (20 nodes) => stress gradiant available

- **Shells:**
  - CQUAD4
  - CQUAD8
  - CTRIA3
  - CTRIA6

## 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 of stress gradient => conservative
  - This leads to considerable less data
- **Supporting effect of stress gradient is only possible with solids!**
  - Results of a 3D analysis with good element quality and fine mesh are more precise than results of 2D-skin elements.

## Modifications for OP2-Output:

**(without these adjustments the assessment doesn‘t work!)**

- **sort: SORT1**
- **Data format: REAL**
- **Yield criteria: VONMISES**
- **Position: BILIN (necessary for analyzing the gradient) or CORNER**

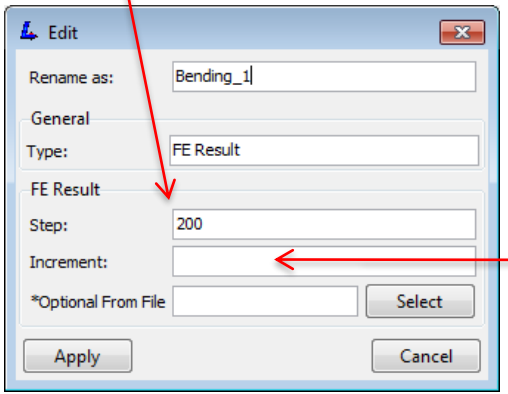
**e.g. STRESS(SORT1,REAL,VONMISES,CORNER)=ALL**

**Note: LIMIT can average corner values, which is activated with the key *CENTER_STRESS**

- **Geometry must be written to .op2-file:** **PARAM, POST, -1**
  **PARAM, OGEOM, YES**
- **Please don‘t use:** **PARAM, PATVER, 2.**

## Addressing FE Results from SOL101 (linear) in the LoadManager:

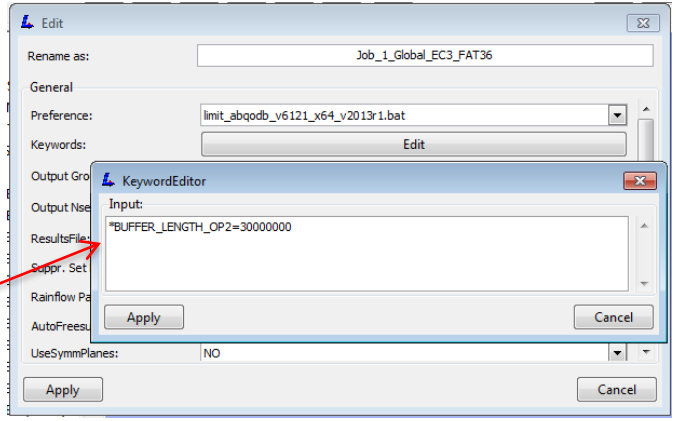
- The Step refers to the SUBCASE number in the input deck




```
$HMNAME LOADSTEP
1"Torsion"
SUBCASE      200
 LABEL= Torsion
 SPC =       1
 LOAD =      2
 ANALYSIS = STATICS
$
```

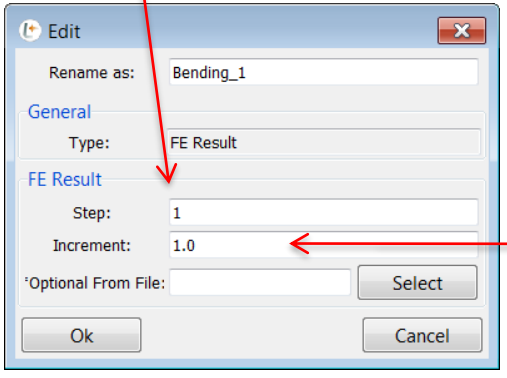
**Remains empty!**

- Maximum SUBCASE number is limited to **1000000** but can be increased in JobManager > Edit > Keywords > Edit:
  e.g. *BUFFER_LENGTH_OP2=30000000



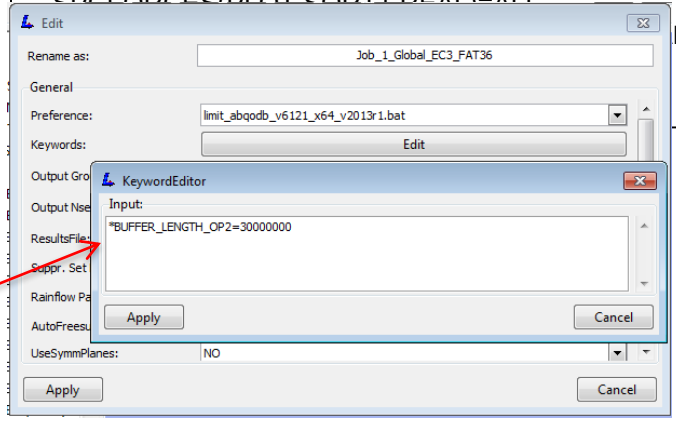
## Addressing FE Results from SOL400 (nonlinear) in the LoadManager:

- The Step refers to the SUBCASE number in the input deck




Total Analysis Time (Blank => last step of subcase!)

```
SUBCASE 1
 STEP 1
   SUBTITLE=Schraube-Force
   ANALYSIS = NLSTATIC
   NLSTEP = 1
   BCONTACT = 1
   SPC = 2
   LOAD = 18
   DISPLACEMENT(PLOT,SORT1,REAL)=ALL
```




- Maximum SUBCASE number is limited to **1000000** but can be increased in JobManager > Edit > Keywords > Edit:
  e.g. *BUFFER_LENGTH_OP2=30000000

## Possible reasons for errors:

- **If the .bdf-file contains the line ‚PARAMOMACHER=YES', parts of the .op2-file are written in double precision. This leads to an abortion during reading the .op2-file**
- **No geometry written to .op2-file. See previous slide.**

**Last slide**