PERMAS

Version 17

Markus Ast

Development Team

INTES GmbH, Stuttgart

Fellbach, April, 12. 2018

Some Highlights

PERMAS V17

- Thread contact on cylindrical mesh
- New MPC-Update in nonlinear statics
- Improved MLDR procedure
- Elements with visco-elastic material
- Revision of laminate handling
- Optimization of laminates
- New stress & error indicators
- Performance improvements
- etc.

VisPER V6

- New wizards:
  Pressfit / Sampling / Design / Assembly
- Improved wizards:
  Contact / Topo / Shape
- Extended Postprocessing and Visualization
- Widely improved comfort & functionalities
- etc.

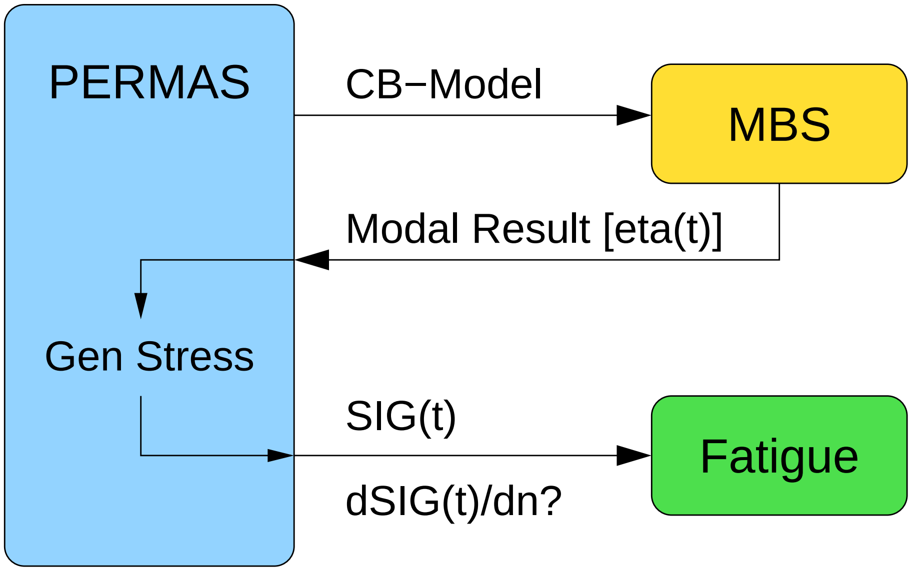
1 General Functionality

General Functionality
Interfaces

- **Medina-Door**
  - Support of BOF records with more than 2^{31} data values (e.g. element stresses for models with more than 24 Mio TET10)
- **Abaqus-Door**
  - Support of substructure generation procedure
  - Support of general material model description for gaskets
  - New translation method of connector actuations available
- **Simpack** Version 9 interface
- New API-based interface for **Excite** (ExbAPI_V2.0)
- New **EMA** interface (Experimental Modal Analysis):
  Read and process measured eigenmodes,
  e.g. for MAC correlation between FE model and measurement.

FE-Stresses from MBS-Displacements (post-calculation with imported results)

GENERATE IMPORT RESULTS can now read results of a MODAL/DIRECT TIMEHISTORY.



This also enables the import of results from a Multi-Body-System and the successive generation of secondary results (e.g. stresses) based on corresponding modes or responses, e.g. for a lifetime analysis.

Export / XYDATA

- **HDF export**
  - Model data in gzip format
  - 64-bit results with DEFAULT SET HDF_DATASIZE = 64

- Permas POST & DATO: PARAM { MODEL / RESULT } DIGITS = 6

- RSYS = ROTB option for XYDATA command

```
SELECT
 ! Extract all dofs of set SET_N1 to a set of curves with standard name
 ACTIVE NSET = SET_N1
 XYDATA ITEM DISPlacement RSYS = ROTB
```

IF Conditions in UCI

$$\textbf{IF} \quad \left\{ \begin{array}{c} (\; op\_a \left\{ \begin{array}{c} < \\ <= \\ == \\ /= \\ >= \\ > \end{array} \right\} op\_b \;) \quad \text{THEN} \end{array} \right\}$$

ENDIF

```
TASK  LOOPS = 12
  ACTivate SITuation = CAR ; STATIC
  ACTivate SITuation = OPT ; OPTIM
! > Print for each second iteration
  IF ('MOD(%(tloop),2)' == 0) THEN
     PRINT; ITEM XYDATA TYPE = OHIS
  ENDIF
TASK END
```

Extended Name/Text Tokens

- Max. length of **name tokens** increased from 40 to **80** characters.
  E.g. improved compatibility in Abaqus-Door.
- Max. length of **text strings** increased from 80 to **256** characters.
  E.g. for descriptions.
- Splitting long names or text over several DAT lines with &
- DEFAULT SET MAXCOLEXPORT = 160
  improves readability of DAT[o]-Files.

Generalize Models by Naming

- '*Named*' model description, e.g.:
  - NIDs or NSETs as simple alternative to coordinate values:
    E.g. rotation axis of $INERTIA or $VELOSTATE definition with
    $$x_1 \quad y_1 \quad z_1 \quad x_2 \quad y_2 \quad z_2 \quad \longrightarrow \quad nid_1 \quad nid_2 \quad \longrightarrow \quad \boldsymbol{nset_1} \quad \boldsymbol{nset_2}$$
  - '*Speaking*' parameters instead of integer control values, e.g.:
    AXES = ±X,Y,Z (instead of just ±1,2,3) for $CONTACT & $SURFACE

- Surface/Line nodes as alternative to nodal point set:
  - $SUPPRESS { <u>NODE</u> / **SURFNODE** / LINENODE }

Harmonization of Options
E.g. STATIC ⇔ NLMATERIAL

Harmonization of STATIC & NLMATERIAL:
(resp. INERTIA & NLINERTIA)

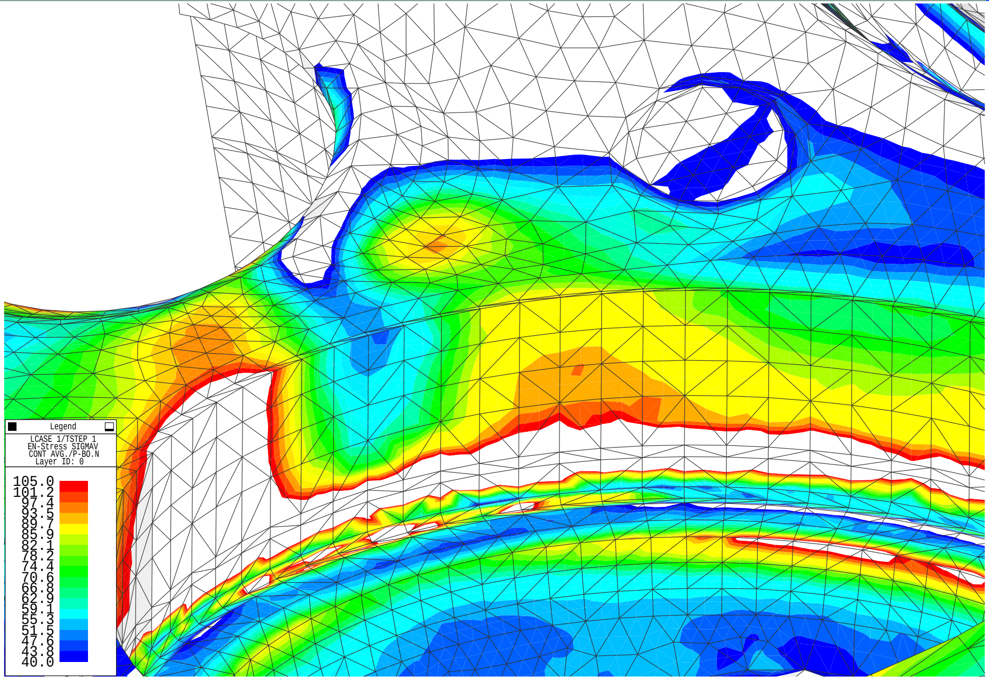
- **TSTART/TEND** parameter **in STATIC** and INERTIA command
  (as in NLMATERIAL)
- NLMATERIAL/NLINERTIA with **negative TSTART/TEND, too**
  (as in STATIC with negative time values in $NLLOAD / $NLRESULTS)
- **NLMATERIAL**/NLINERITA **to RESIDUAL** as default option, i.e.
  GENerate REACtion and GENerate RESidual done by default
  (as in STATIC)

Simplification by More Powerful Commands

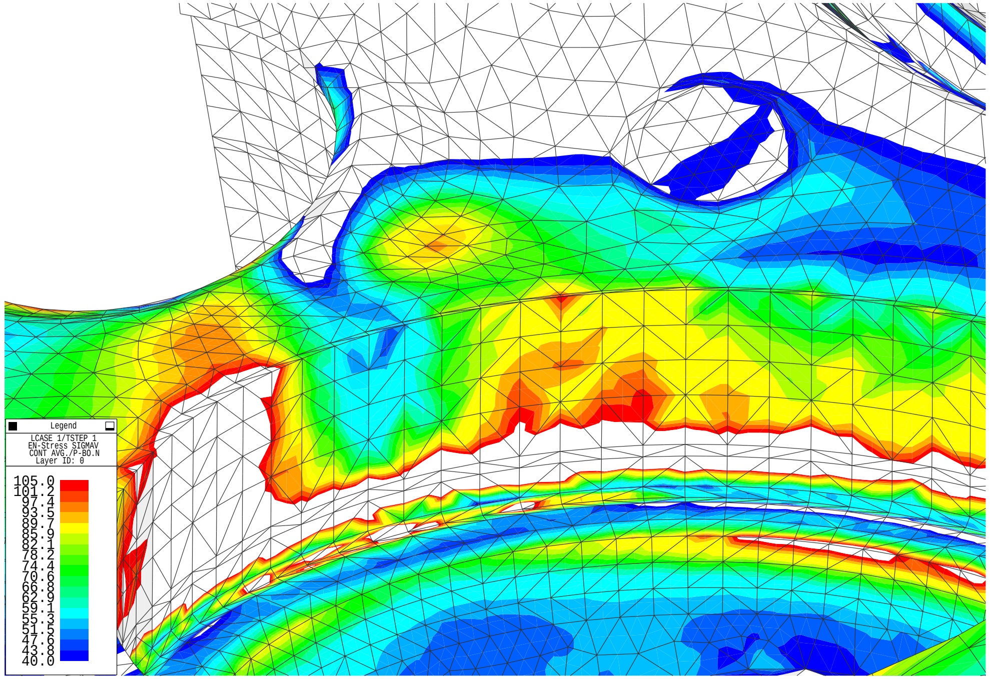
Obsolete statements (but still working)		Can be handled by V17 Extension to . . . (preference for new models)
~~NLINERTIA~~	⟶	NLMATERIAL with inertia relief loads (also inertia relief with NLGEOM=yes !)
~~$MPCUPDATE~~	⟶	DEFAULT SET MPCUPDATE = COEFF (just update all relevant MPCs)
$MPC ~~DEPCOOR=PROJECT~~	⟶	New MPC-Update with lever-arm correction (just keep original, true geometry)
$MPC ~~ISURFLIN~~	⟶	$SURFACE with MIDNODE=DEPENDENT (no need for MPC or 2nd surface)

$SURFACE MIDNODE=DEPENDENT with **Tria2Quad interpolation**

- Linearization of Tet10 surfaces in order to reduce contact DOFs
- Problem: Local stress evaluation (i.e. gradients for lifetime analysis)

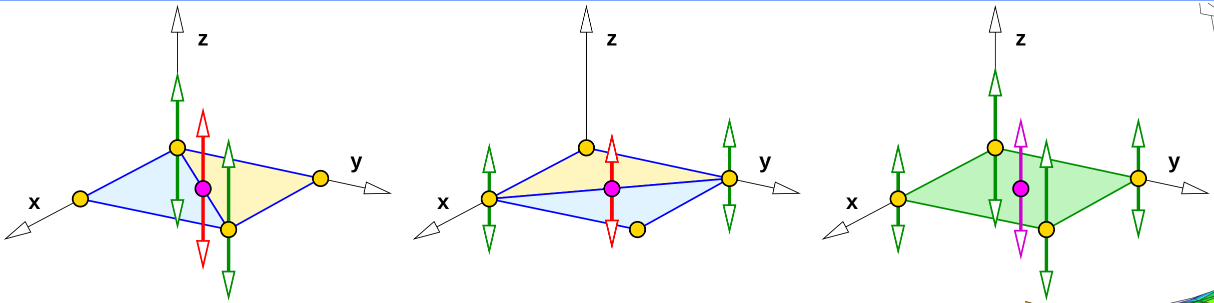


Reference (full quadratic)

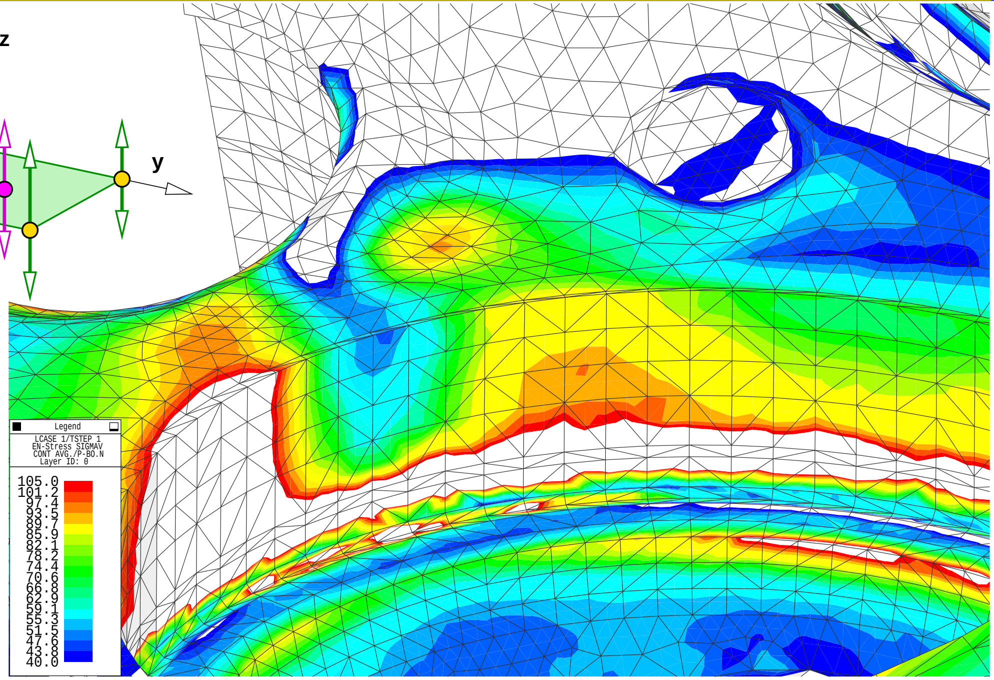


Surface linearization in V16 (without Tria2Quad interpolation)

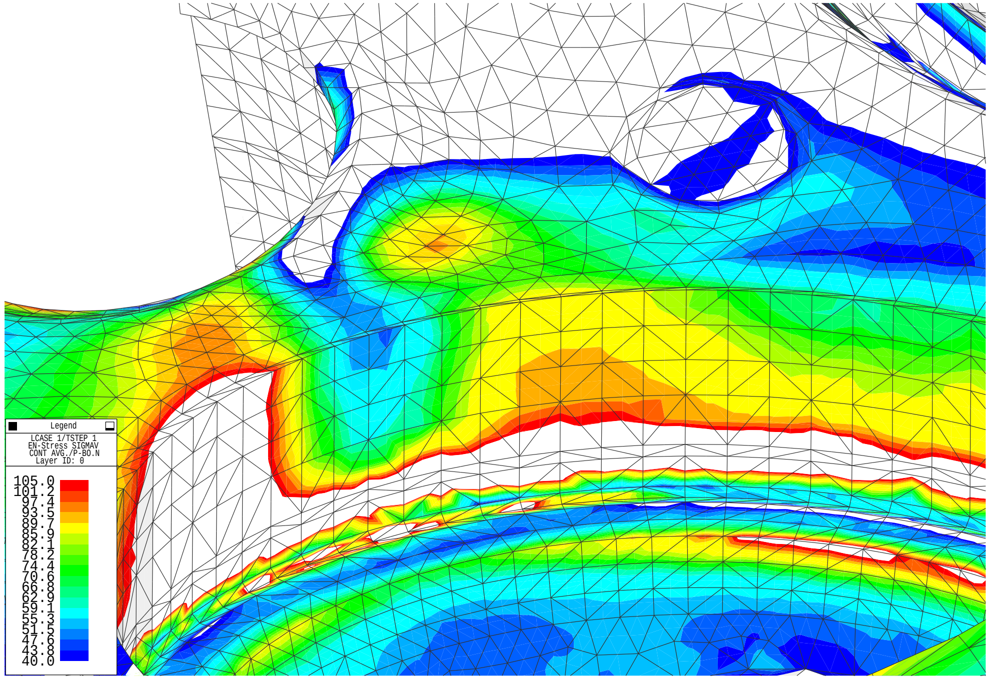
$SURFACE MIDNODE=DEPENDENT with **Tria2Quad interpolation**



- Improvement for MPCs & Contact
- Default in PERMAS V17



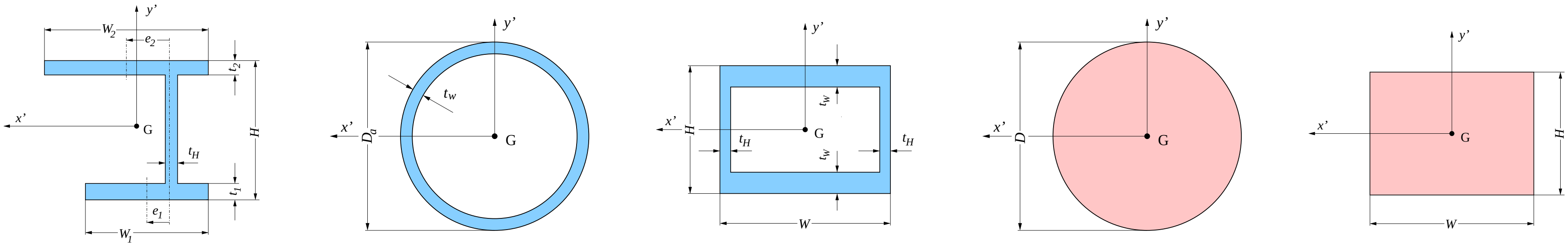
Reference (full quadratic)



Surface linearization in V17
with Tria2Quad interpolation

General Functionality
Miscellaneous

- Completed standard beam cross-sections by solid circle and rectangle:



- $REFVAL TEMP with new default LPAT=ALL ensures an identical reference temperature $T_0(eid)$ for all load patterns or time steps respectively.

```
  $REFVAL   TEMP    LPAT=ALL
   MYESET   20.0
```

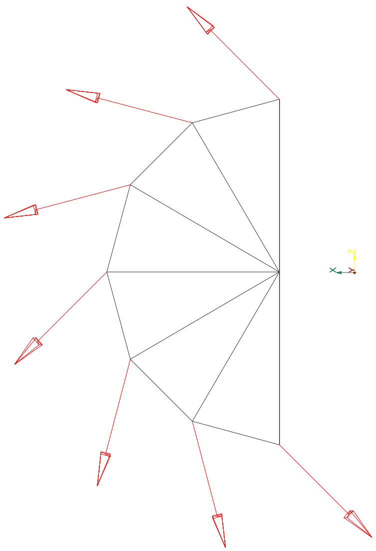
- $COORGEN BARYCENTER: Generate a center node for a given node set.

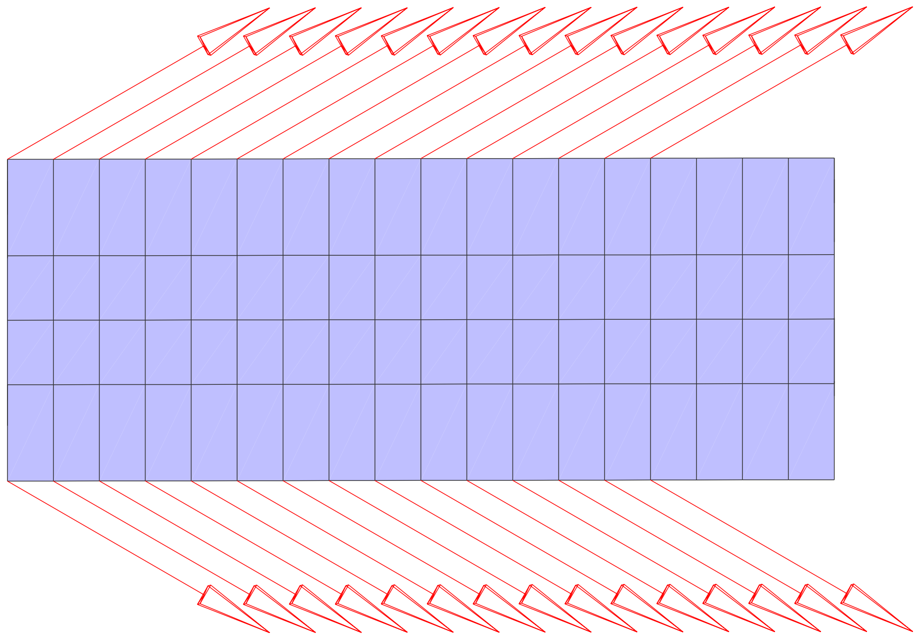
$RSYS with extra Euler rotations (for all local displacement systems)

$$\textbf{\$RSYS} \quad rsystype \quad \left[\text{FORM} = form \right]$$

$$sysid \quad \left\{ \begin{matrix} nid_1 \\ x_1\, y_1\, z_1 \end{matrix} \right\} : \left\{ \begin{matrix} nid_2 \\ x_2\, y_2\, z_2 \end{matrix} \right\} : \left\{ \begin{matrix} nid_3 \\ x_3\, y_3\, z_3 \end{matrix} \right\} \left[: \ \textbf{list(} \ iaxis, angle \ \textbf{)} \right]$$

```
$RSYS    CYL  ! RSYS 17 plus successive Euler rotations around z' and x''
 17      0.0  0.0 1.0  :  1.0 0.0 1.0 : 1.0 0.0 2.0  :  3,45.0  1,15.0
```

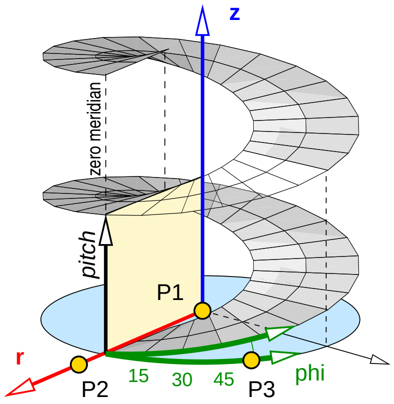




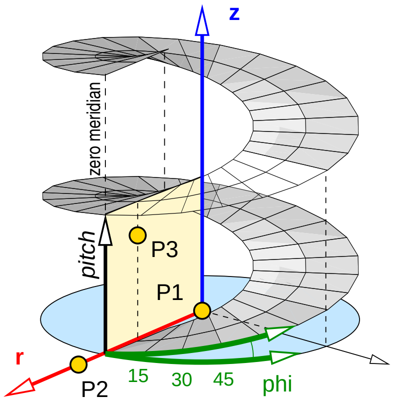
$RSYS HELIX & THREAD
(e.g. for '*thread-less*' thread contact)

PERMAS Users' Conference Stuttgart 2018

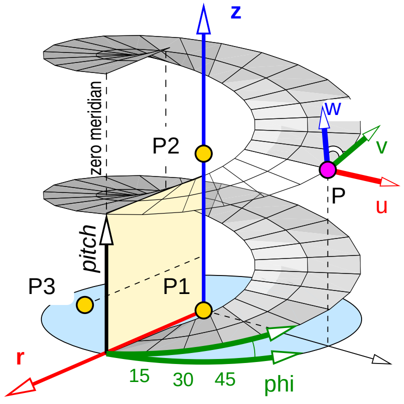
INTES



FORM = RPHI

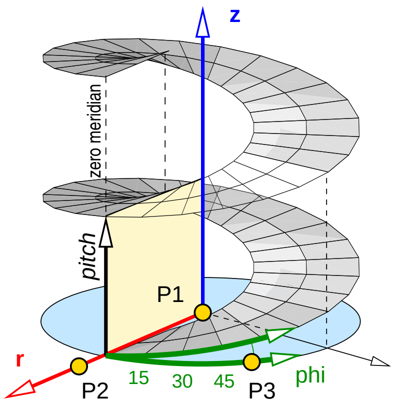


FORM = <u>RZ</u>

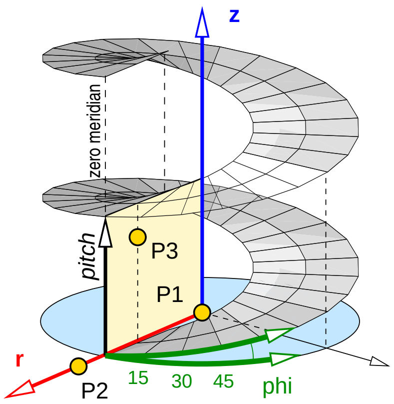


FORM = ZR

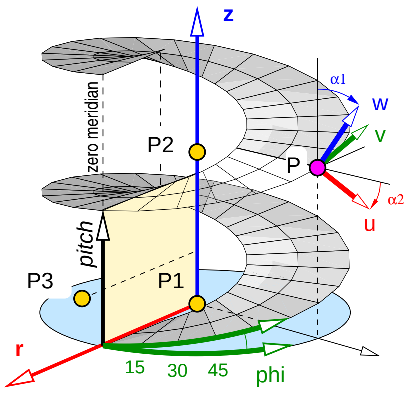
$RSYS **HELIX**



FORM = RPHI



FORM = <u>RZ</u>



FORM = ZR

$RSYS **THREAD**

$\widehat{=}$ HELIX with Euler rotations for ALPHA & PITCH

2 Nonlinear Statics

Surface-Contact with Non-Normal (pre-defined) Contact Direction

Up to now the normal direction of a $CONTACT SURFACE definition was strictly coupled to the normal direction of the underlying surface.

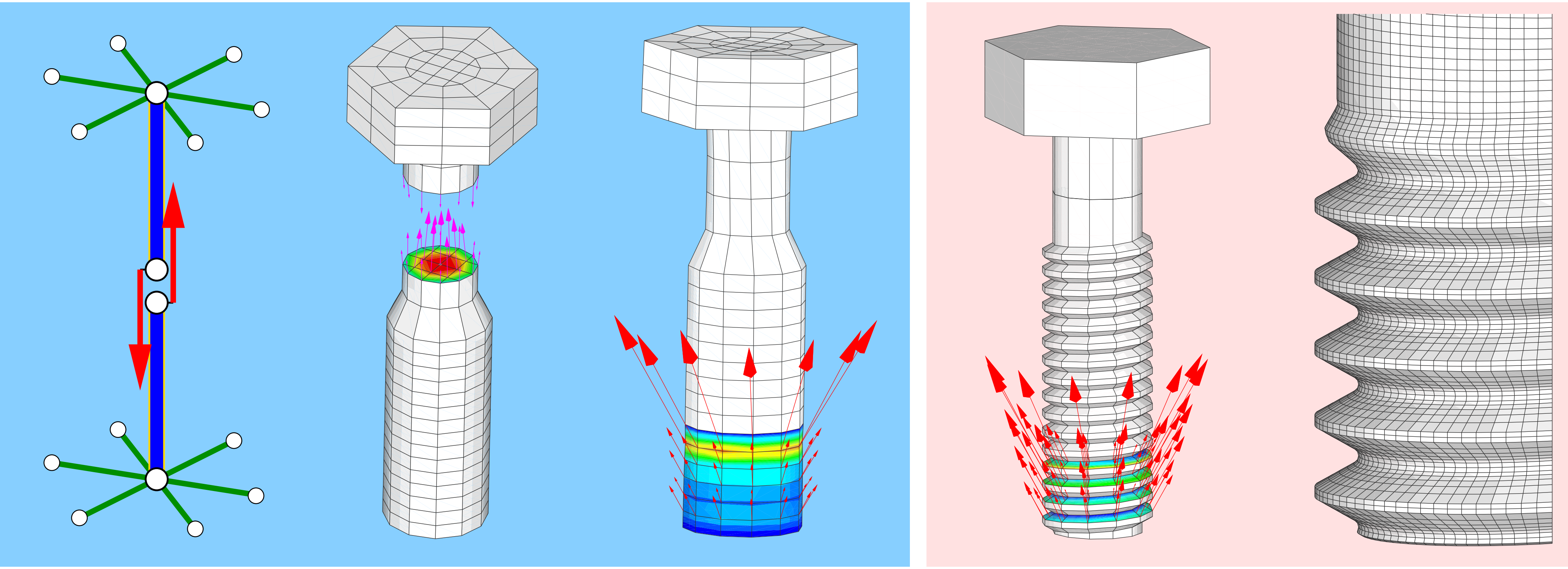
$\Rightarrow$ **New in PERMAS V17**:

$$
\textbf{\$CONTACT}\quad\textbf{SURFACE}\quad \left[\,[\text{TO}]\left\{\begin{array}{c}\underline{\text{NODE}}\\ \text{SURFNODE}\\ \text{SURFACE}\\ \text{SELF}\\ \text{GROUND}\end{array}\right\}\right]\quad\left[\text{FRICTION} = \left\{\begin{array}{c}\underline{\text{NO}}\\ \text{Y}\\ \text{Z}\\ \text{BOTH}\end{array}\right\}\right]
$$

$$
\left[\,\textbf{CONTSYS =}\left\{\begin{array}{c}\underline{\text{NORMAL}}\\ \\ \boldsymbol{sysid}\end{array}\right\}\right]\quad\begin{array}{l}\left[\text{YREF} = plane\,[,\gamma]\,[,sysid]\right]\\ \left[\textbf{AXES =}\ \boldsymbol{idofn}\,[,\boldsymbol{idofy}]\right]\end{array}
$$

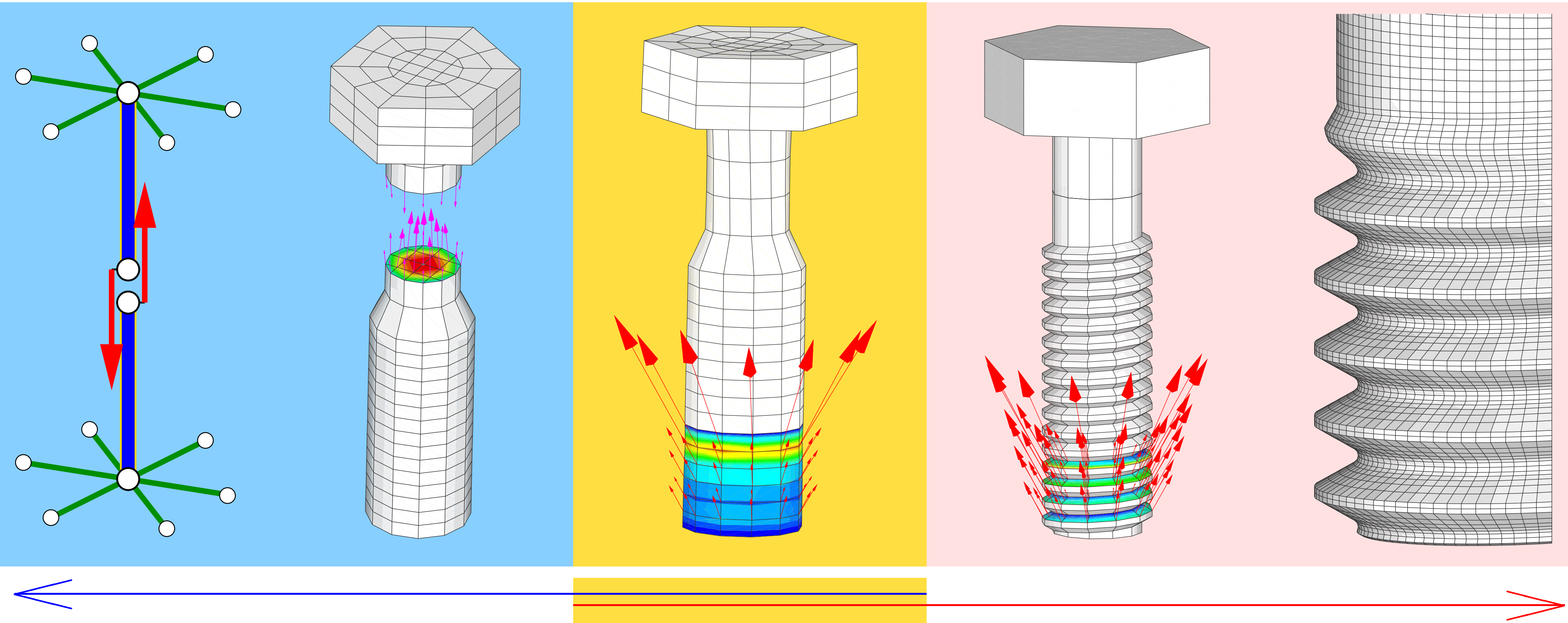
$\Rightarrow$ E.g. (by referring $RSYS THREAD) incompatible thread contact on cylindrical meshed screw – incl. pitch, flank and slip-stick friction !

Pretension/Screw Models – V16 (Type of Analysis and Geometric Details)



1) Beam model with pretension (cut & contact with FORCE_LOCK)
2) Cylindrical volume model with $PRETENSION PLANE (PERMAS **V12**)
3) Cylindrical volume model with $PRETENSION THREAD (PERMAS **V12**)
4) Detailed thread model with contact in flank normal direction
5) Submodelling with extra thread details and e.g. plasticity

Pretension/Screw Models – V17
(Type of Analysis and Geometric Details)

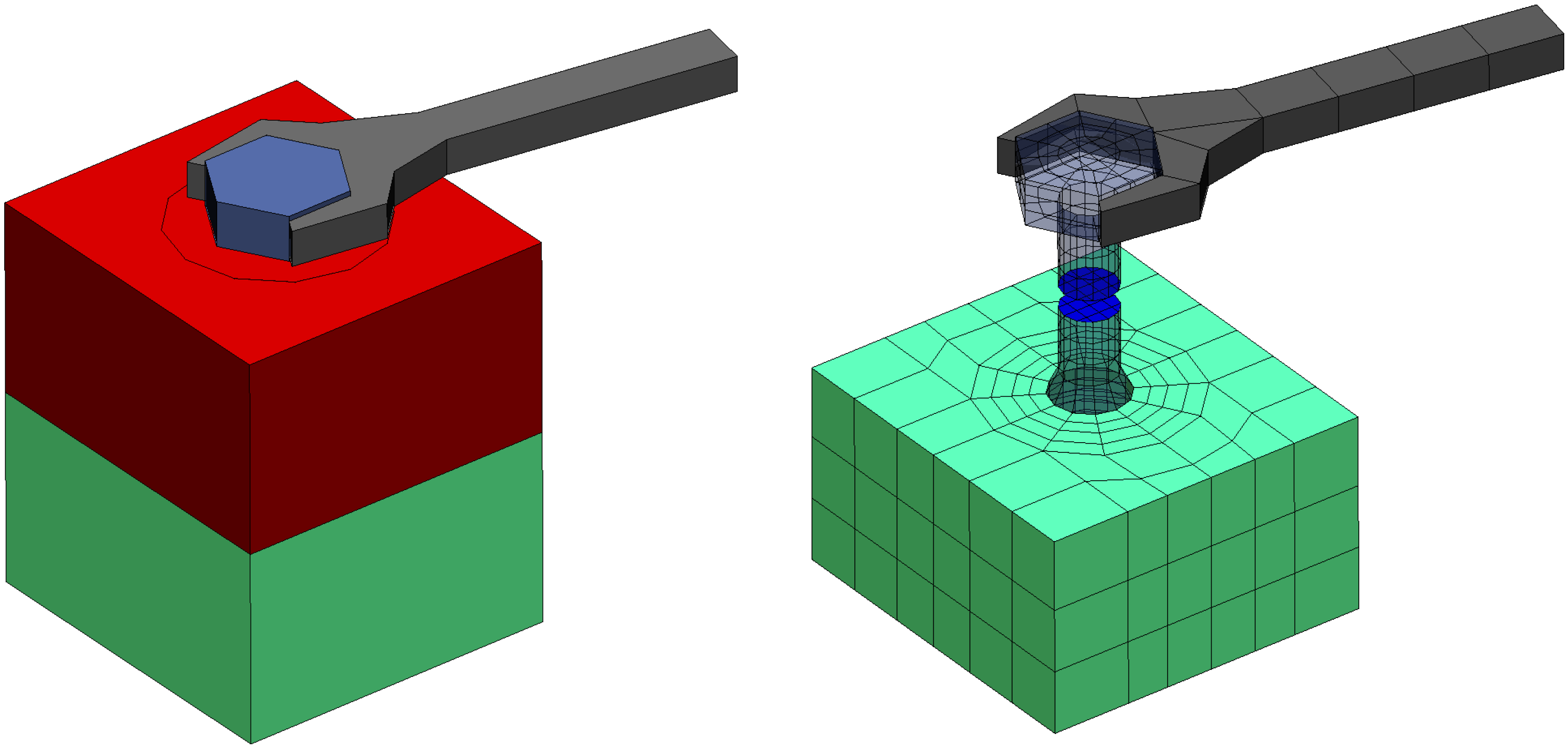


Pretension models
with MPCs and 1 CA-DOF
no local contact, no friction

Contact models
many CA-DOFs, no MPC
local contact and friction

CA20: Tightening/Unscrewing Torque

The necessary tightening and unscrewing torque for a screw with given pretension force can be obtained from a simple cylindrical model – **incl. flank/pitch friction in virtual thread** and under head.

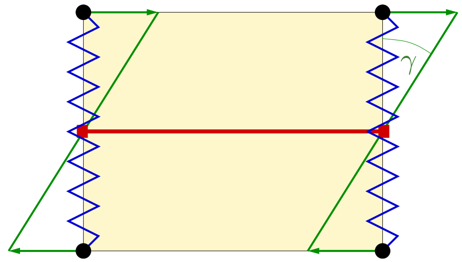


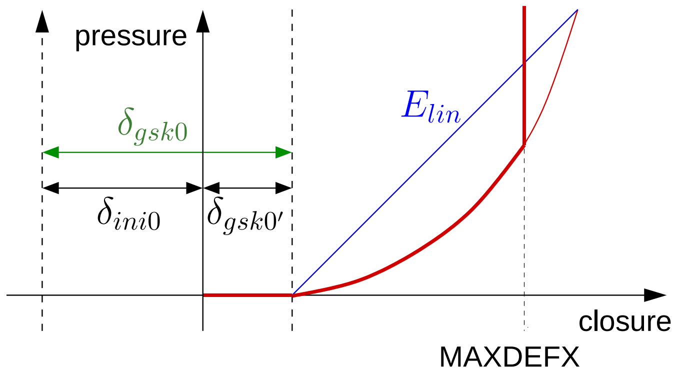
Also: No CAU or NLGEO=YES is needed - just simple STATIC !

Gaskets

New in **PERMAS V16**:

- Cleaner Gasket formulation (membran with decoupled springs)/
- **$GEODAT INIGAP** ($\delta_{ini0}$ shift in material curve)
- Joint handling of outer contact gap and internal offset: $\delta_{uni0} = \delta_{ca0} + \delta_{gsk0} = \delta_{ca0} + (\delta_{ini0} + \delta_{gsk0'})$

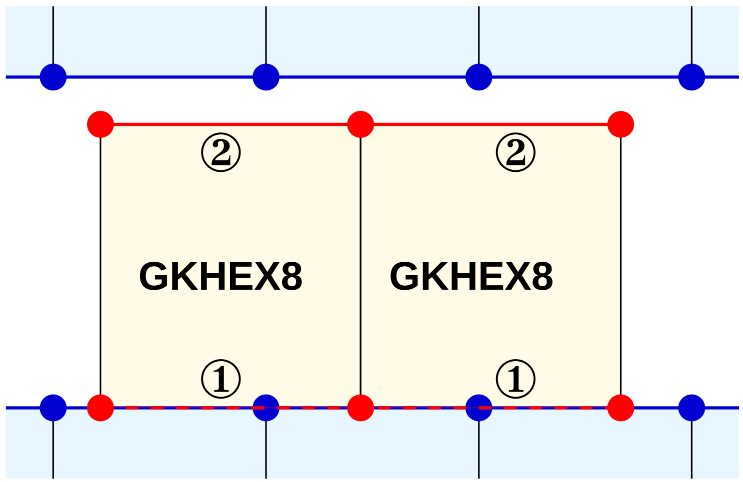




New in PERMAS V17:

- $ELSYS optional (default from node sequence)
- **Surface-To-Surface contact with Gaskets.**

  Attention: Gaskets must be 2nd partner !
- ① $MPC ISURFACE ...
  SURF1 SURF2
- ② $CONTACT SURFACE SURFACE/NODE ...
  SURFACE NODE



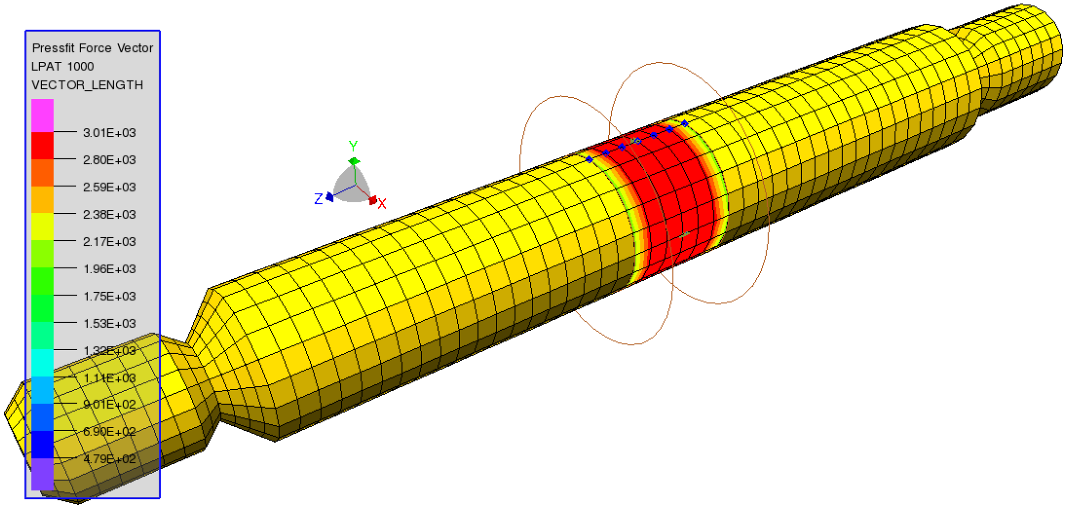
Contact Analysis Miscellaneous

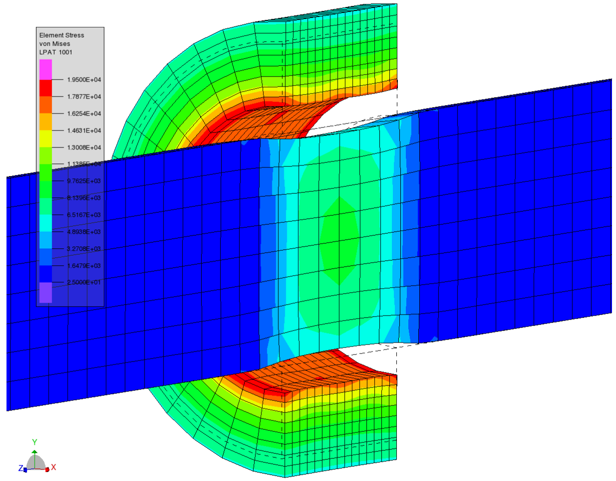
- Improved reuse of **CASO-File** (e.g. for $VELOSTATE and $CONTLOCK)
- New default option SET CASFILES = AUTO
  (always write caso-files for larger models as backup)
- SET CASINAME=*inpre* to change the prefix for reading CAS[O] files
  (SET CASNAME=*pre* changes prefix for read & write)
- Max/average initial distance in CA-Neighborhood statistics:

> Statistics of contact neighborhood search for surface contacts in situation SITU_4 of component KOMPO_1

Contact Definition (contname/contid)	partner type	CA-Points (total)	CA-Pairs found	CA-Pairs found [%]	removed doublets	CA-Pairs used	max.dist (GapGeo)	avg.dist (GapGeo)	max.kink angle
54	Surface	2738	1268	46.3	634	634	1.60E+00	1.20E+00	0.13
55	Surface	5877	2896	49.3	1404	1492	2.71E+00	2.04E+00	28.41
56	Surface	2836	1310	46.2	655	655	3.60E+00	3.20E+00	1.84
57	Surface	5218	2548	48.8	1227	1321	2.61E+00	2.01E+00	20.21
58	Surface	11616	9546	82.2	4773	4773	9.90E-13	5.05E-14	28.41
59	Surface	10879	8788	80.8	4394	4394	9.81E-13	5.22E-14	30.17
68	Surface	3647	1449	39.7	479	970	7.36E-02	5.48E-02	0.79

V16 $PRESSFIT Definition



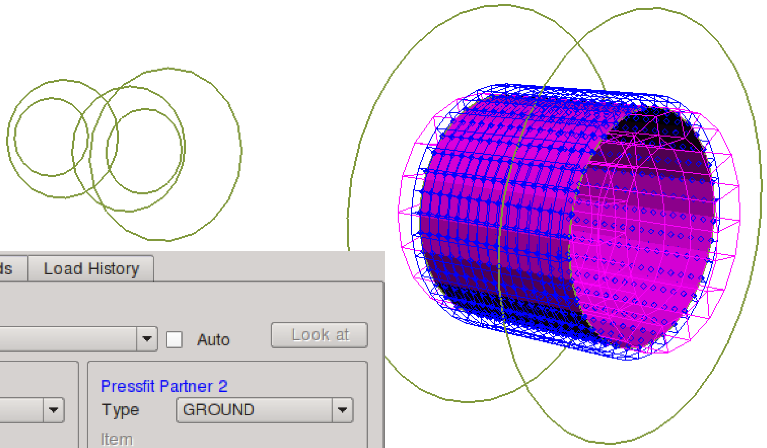


- **Press-fit as pure MPC coupling** – **without any CA-DOFs** !
- Permanent coupling with load variable interference,
  i.e. overlap = f (LPAT & $NLLOAD)

≈ $MPC ISURFACE with prescribe in surface normal direction

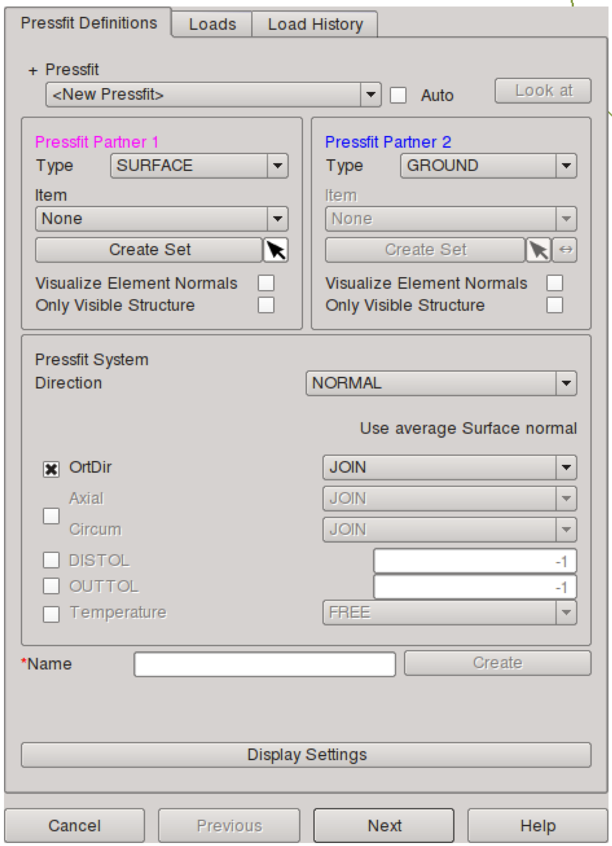
- Works for static & dynamic analyses (e.g. pre-stressed modes).
- Also applicable in sub-components.
- Specific results for: Verification, Precheck and Calculation available

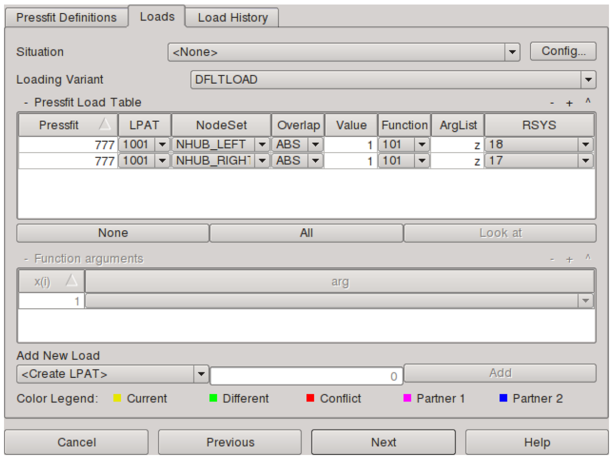
VisPER V6: Pressfit Wizard

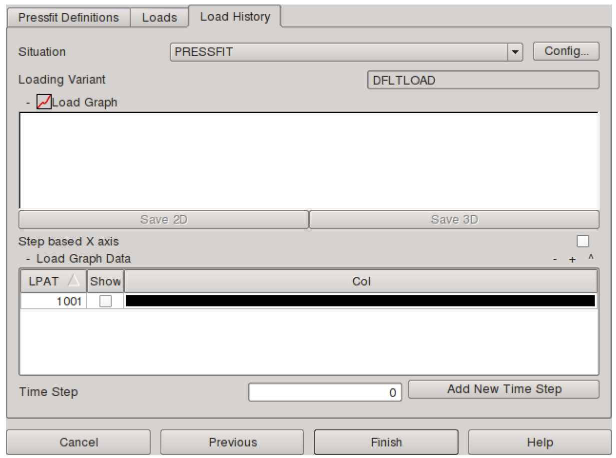


- Dedicated wizard for easy and complete definition of **$PRESSFIT** partners, load (incl. functions) and load history

- Special pressfit visualization







Nonlinear Statics
MPC-Update & NLGEOM=YES

- UCI: DEFAULT SET MPCUPDATE = **COEFF** as new default

  ⟶ **$MPCUPDATE** in DAT file **now obsolete !**

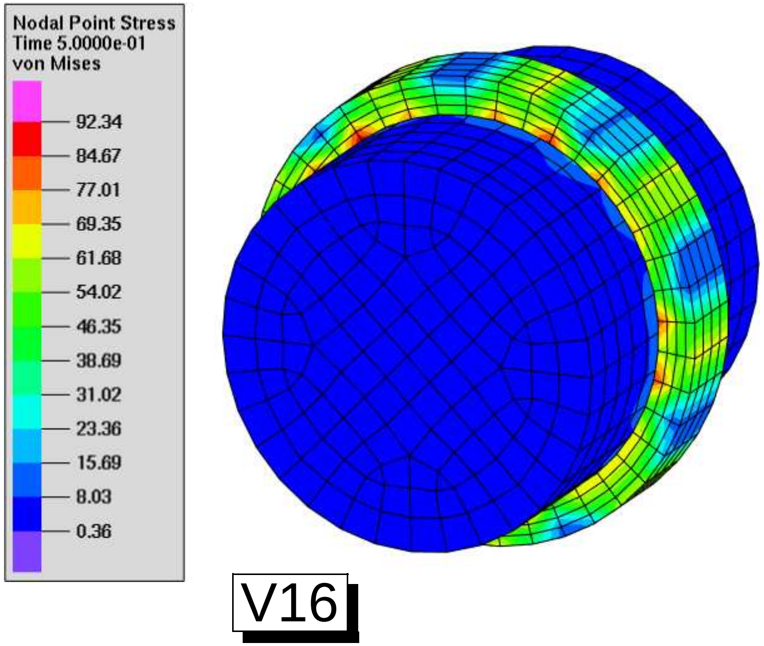
- Improved Handling of multiple rotations within MPC-Update

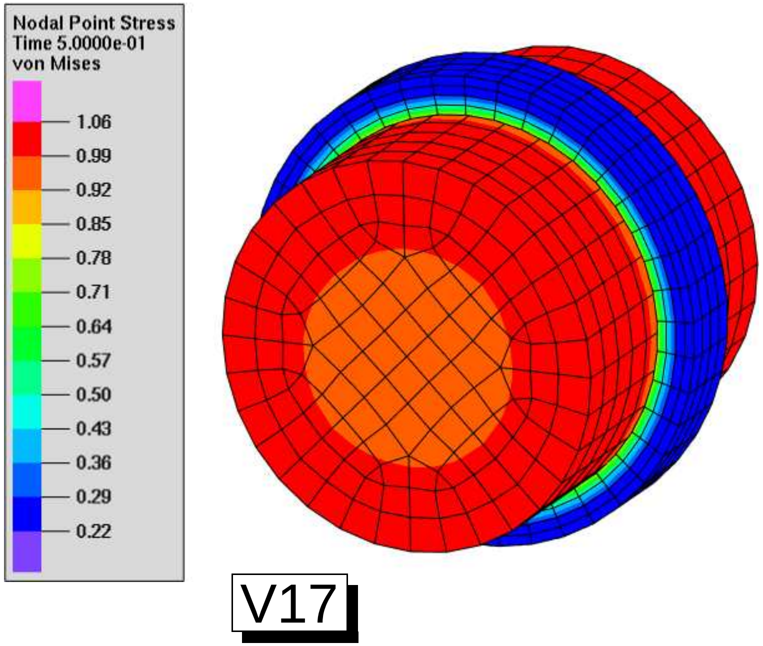
  ⟶ **More stable NLGEOM iterations !**

- **Lever arm correction** for $MPC **ISURFACE** / ITRIA / IQUAD with rotation

  ⟶ **DEPCOOR = PROJECT obsolete** ⟶ just use original geometry !

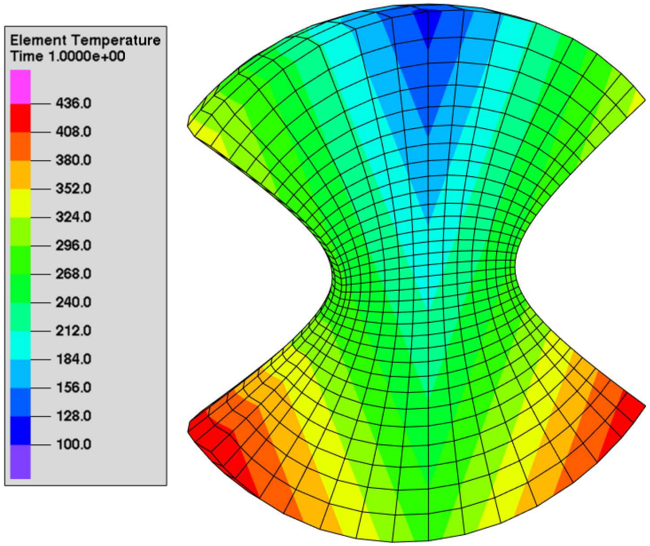
MPC-Update and $MPC ISURFACE Lever Arm Correction

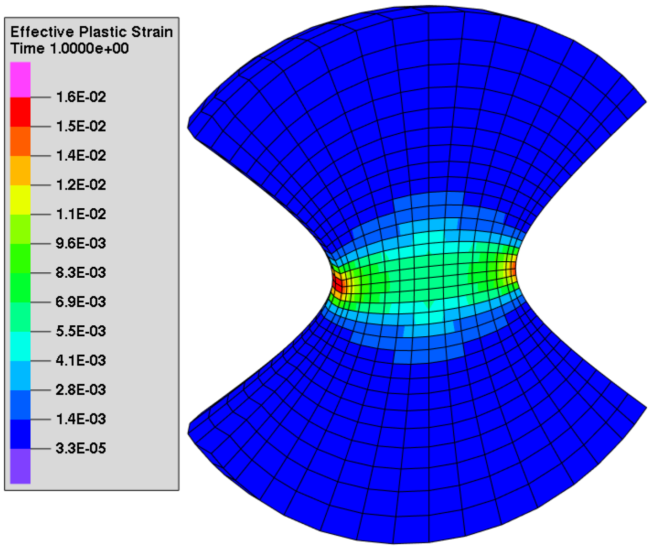




- incompatible mesh: 32/41 ring/cylinder elements along circumference
- coupling via $MPC ISURFACE, but without DEPCOOR=PROJECT
- axial pressure of 1.0 applied at cylinder sections
- rotation of 45 degree in 5 load increments

Cast Iron Plasticity
Improved convergence and stability





Notch CASTIRON Example:

- tension & inhomogeneous temperature
- temperature dependent yield curve

V16

Summary of NLMATERIAL Calculation

Increment	Time-Increment	Time	Iterations	Sol-Meth
0	0.20000	0.20000	13	RESTART
1	0.10000	0.10000	2	NR
1	0.14142	0.24142	13	RESTART
1	0.07071	0.17071	13	RESTART
2	0.02828	0.12828	2	NR
3	0.04000	0.16828	2	NR
3	0.05657	0.22485	13	RESTART
3	0.02828	0.19657	13	RESTART
3	0.01131	0.17960	13	RESTART

V17

Summary of NLMATERIAL Calculation

Increment	Time-Increment	Time	Iterations	Sol-Meth
1	0.20000	0.20000	2	NR
2	0.40000	0.60000	3	NR
3	0.40000	1.00000	4	NR

Nonlinear Statics Miscellaneous

- Smooth interpolation for tabular material data with two independent variables: E.g. improved convergence for temperature dependent plasticity

- Inertia relief loads now handled by NLMATERIAL, too:

  New Option: **RMDOFS**=SUPPRESSED in $RIGMODE definition

  $\longrightarrow$ Inertia Relief with material **and** geometric nonlinearities !

  $\longrightarrow$ **NLINERTIA now obsolete !**

- Combination of nonlinear statics with a cyclic symmetry analysis for eigenvalues $\longrightarrow$ Example NLS6V1

- Support of **pressure stiffness for quadratic elements**, too

- Additional parallelization of element operations (improved performance)

3 Dynamics

Vibration Analysis with MLDR (Multi-Level Dynamic Reduction)

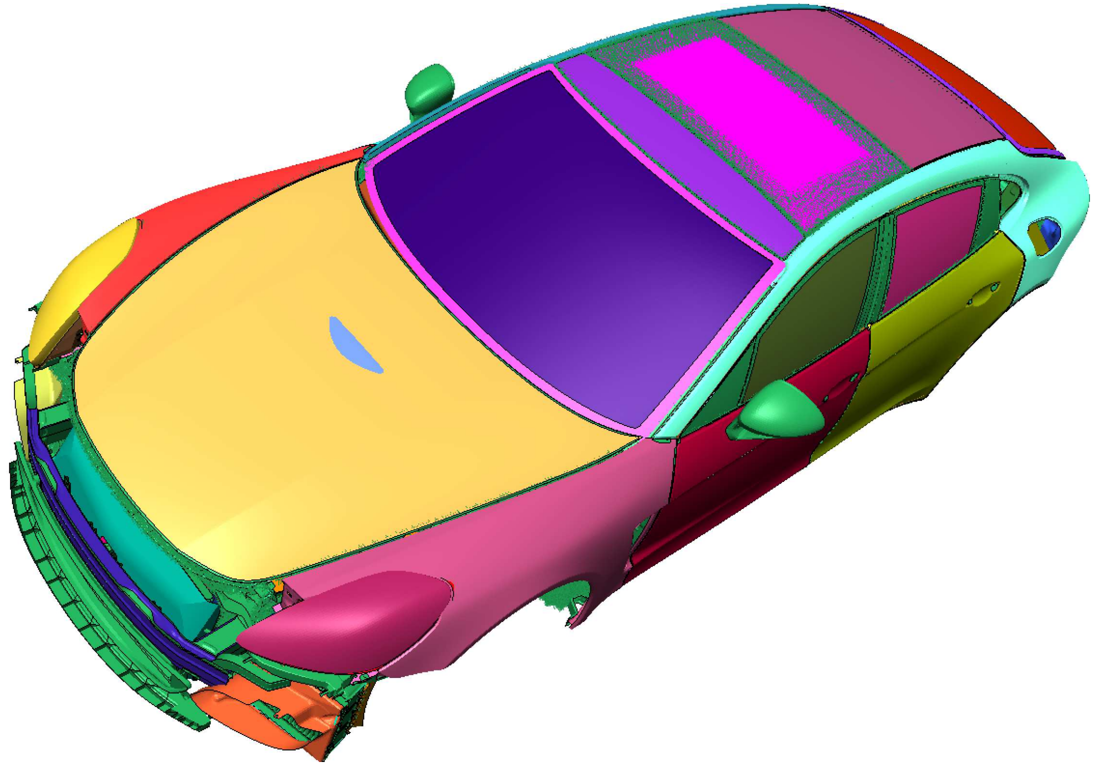
Three measures for **improved MLDR performance**:

- **improved model decomposition** (i.e. extra handling of MPCs)
- **additional static reduction** of Guyan parts via:
  DEFAULT SET SUBGRED = { NONE / EIGEN / MIXED }
- **ADDMODES** calculation now done **within MLDR tree**
  (instead of ADDMODES in original component)

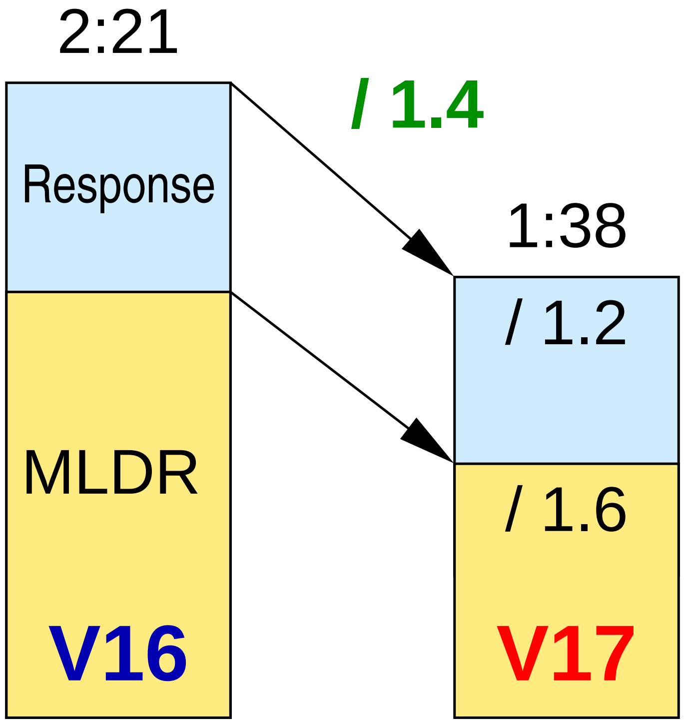
⇒ Especially effective for higher frequency limits and denser (finer) model connectivities.

NVH Analysis of a Car Body - 1

(Vibration Analysis & Modal Frequency Response)



1∗E5-2697
(18 cores)

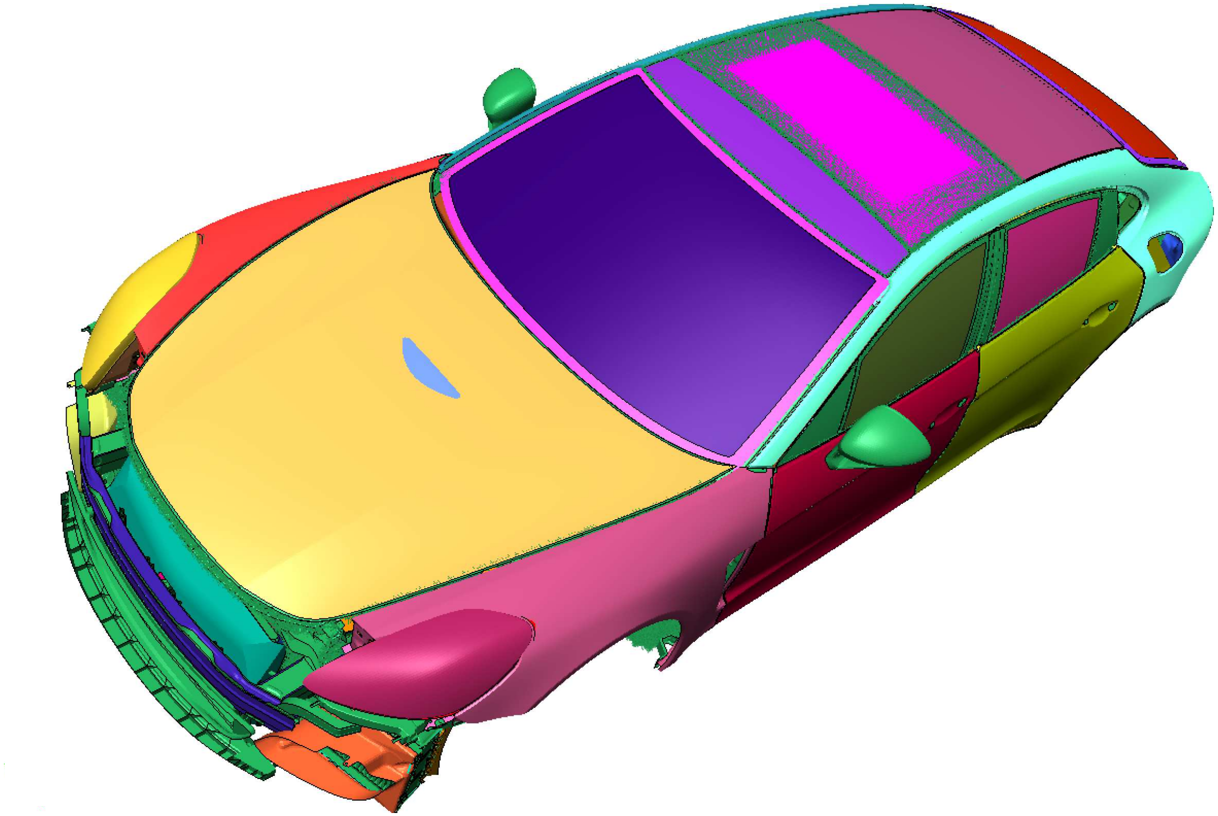


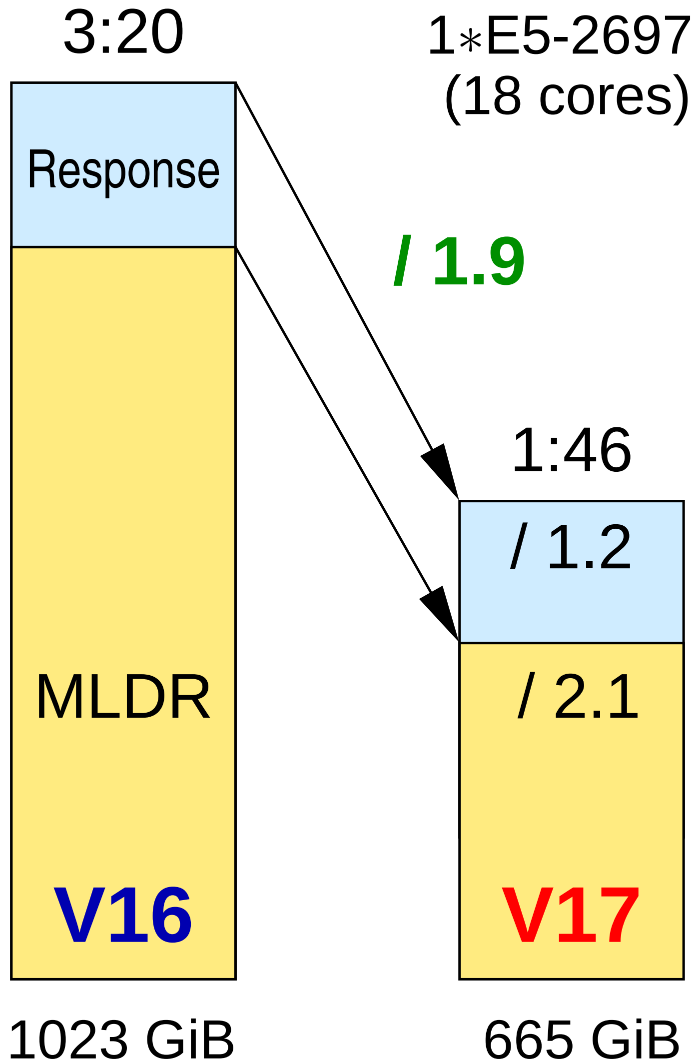
DOFs	15 600 000	
Nodes	3 500 000	
Elements	3 000 000	
Modes	5 800	(Freqlim=800.0)

603 GiB 520 GiB

NVH Analysis of a Car Body - 2

(Vibration Analysis & Modal Frequency Response)





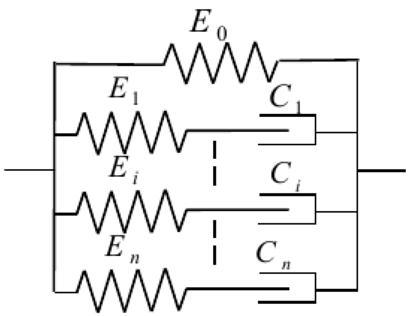
DOFs	20 300 000	
Nodes	4 100 000	
Elements	4 400 000	
Modes	7 300	(Freqlim=800.0)

Elements with Visco-Elastic Material

Technical Context

❑ Generalized Maxwell Model

- Use of a generalized Maxwell model for the Young's modulus



$$E(f) = E_0 + \sum_{i=1}^{n} \frac{i \cdot 2\pi E_i \cdot f}{\left[\left(i \cdot 2\pi \cdot f \right) + E_i / C_i \right]}$$

Prony series

- Introduction of this law in the FE formulations

⇨Creation of **compressible viscoelastic** (frequency dependent) **elements**

- Drawbacks:
  Increased number of dofs (ex: 66 dofs with 7 branches in a 3D hexahedron)

Elements with Visco-Elastic Material
HEXVE/R8 based on Prony Series

- For materials with frequency dependent data in DIRECT TIMEhistory or DIRECT FREQUENCY analysis (e.g. rubber, booster propellant, . . .).
- Two linear hexahedron elements, based on **Prony Series**:
  - **HEXVE8** – Element with explicit user discretisation:
    One element with separate nodes for each term of the Prony series (nodes need an extra MPC JOIN, i.e. one MPC per node per term of Prony series).
  - **HEXVR8** - Element with internal condensation:
    One element with an additional (internal) node for each term of the Prony series (coincident structural nodes, i.e. no MPC Join needed).
- Standard or EAS (Enhanced Assumed Strain) formulation available with DEFAULT SET LHEXSHAPE

Elements with Visco-Elastic Material
Example

i	Basic	1	2	3	4	5
E_i [MPa]	1400	100	175	100	75	150
τ_i [s]	-	$5{\cdot}10^{-3}$	$1{\cdot}10^{-3}$	$7.5{\cdot}10^{-4}$	$3{\cdot}10^{-4}$	$5{\cdot}10^{-5}$
ϱ_i [t/mm^3]	$8.9{\cdot}10^{-10}$	-	-	-	-	-

The frequency-dependent material data are shown in the following figures:

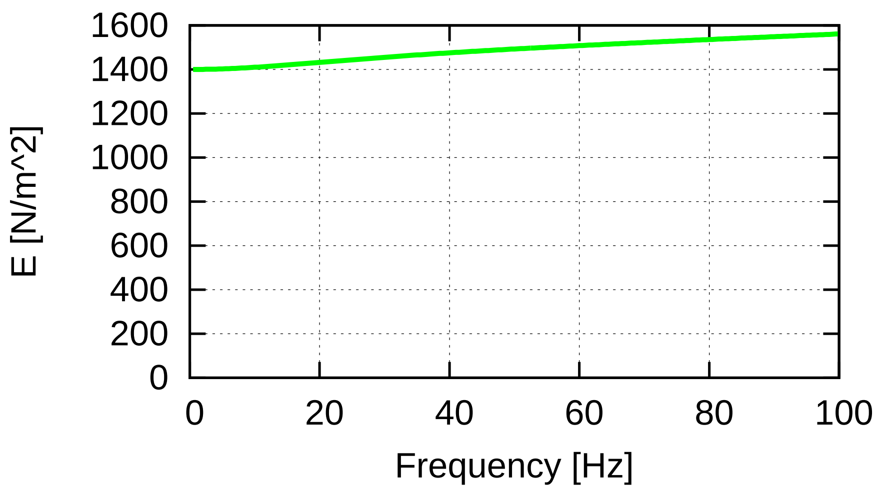


Figure 1: Young's modulus $E(f)$

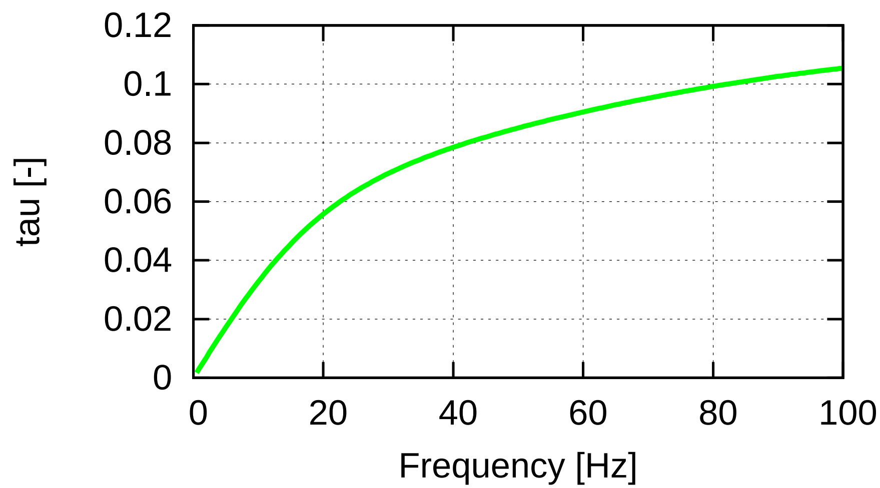
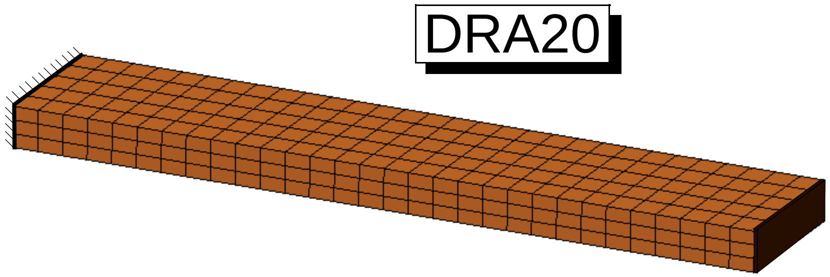


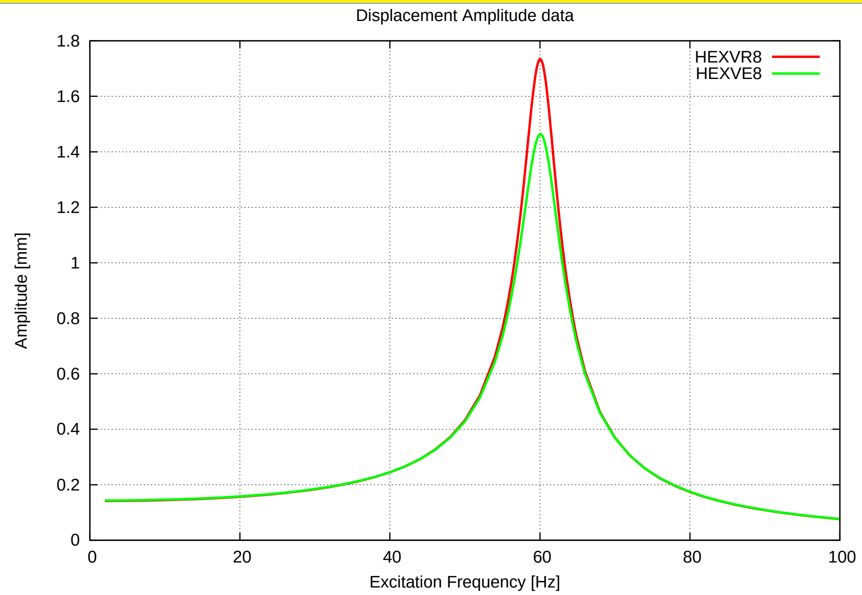
Figure 2: Time constant $\tau(f)$

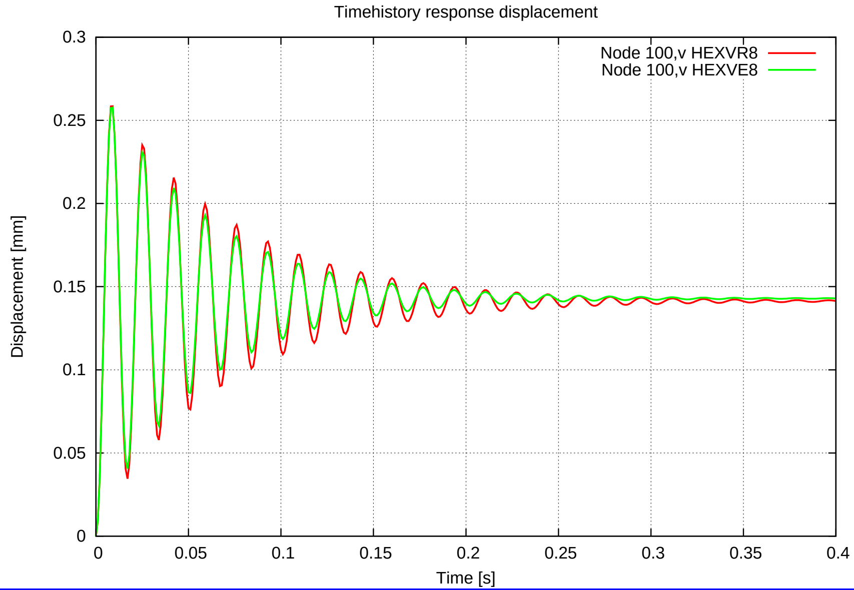
Elements with Visco-Elastic Material Analyses

- **DIRECT FREQuency**



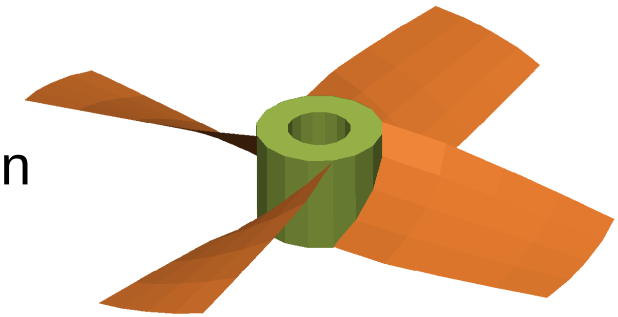
- **DIRECT TIMEhistory**





Dynamics Miscellaneous

- MODAL COMPLEX / MODAL ROTATING:
  - Parallel solver for complex eigenvalues (with automatic selection)
  - Equivalent viscous damping for real eigenfrequencies by new command option RFREQ = **EIGFREQ**
- **Stability of cyclic symmetric rotors** (#sectors≥3) with Multiblade Coordinate (Coleman) Transformation (delivers system with constant matrices in fixed frame of reference; any kind of support possible)



- **Random** response analysis for **FS-Coupled** problems:

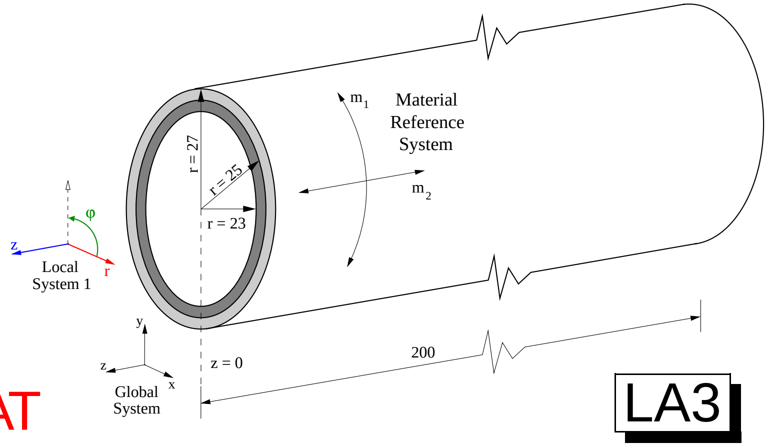
```
MODAL FSCoupled RANDOM  SITUATION=MODRANDRESP
```

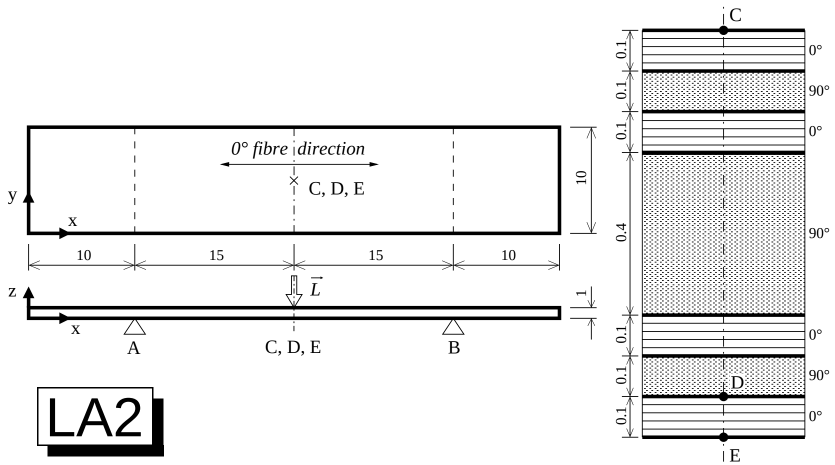
- Stiffness-Shift for buckling analysis

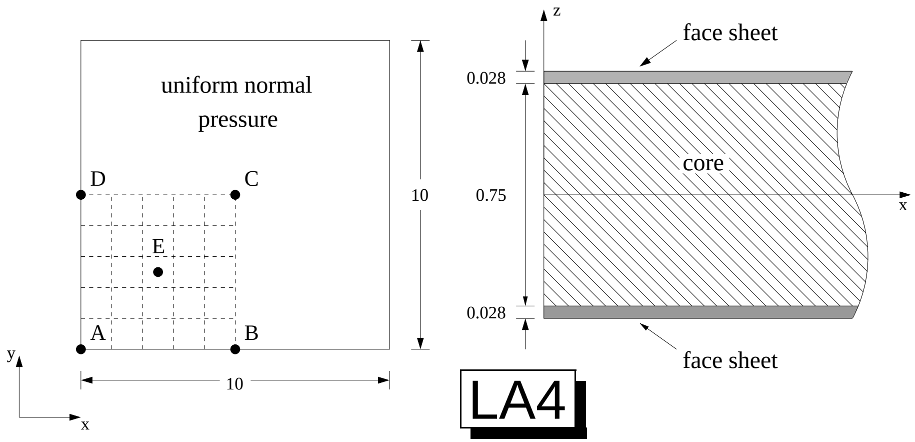
4 Laminates

Revision of Laminates

- Revised implementation of laminate elements
- Adaption of transverse shear behavior
- Element wise re-definition of ply data: $PLYDAT
- New ply failure calculation available: $PLYFAILure







$PLYDAT
Element Ply Property Data

- Element wise re-definition of **ply thicknesses & angles** for given laminate elements (overwriting initial layup from $LAMINATE/$PLY material)
- For manual correction and/or trimming (also used by optimization)
- Available in $SYSTEM-variant ⟹ flexible modeling

```
! Composite ply layup including zero thickness layers for output
  $PLYDAT   ASYM   NPLY = 5   CONT = THICK  ANGLE
    HULL      0.0   0.0     ! 1st ply, bottom
       &  :   0.2   0.0
       &  :   0.6  90.0     ! 3rd ply, middle
       &  :   0.2   0.0
       &  :   0.0   0.0     ! 5th ply, top
```

$PLYFAILPAR
Parameter for Ply Failure Criteria

- Definition of **parameters** (typ. strength-data) **for failure criteria**:
- Available in $RESULTS-variant $\Longrightarrow$ flexible modeling

```
$PLYFAILPAR   NAME=STRESSLIM_CFRP_UD
    LAMMAT_NPLY6  2 3 4 5 : 2200.  930.  50.  160.  50.  160.  :  60.  30.  60.
    LAMMAT_NPLY6  1 6     : 2200.  930.  50.  160.  :  60.

$PLYFAILPAR   NAME=CFRP_UD_TSAI_WU
    LAM2_SYM  ALL  : 2200.  930.  50.  160.  :  60.  :  -0.5

$PLYFAILPAR   NAME=PFP_USER
    MAT_CFK  : 2200.  930.  50.  160.  60.  -0.5
```

UCI: PLYFAILure
Evaluation of a Ply Failure Criterion

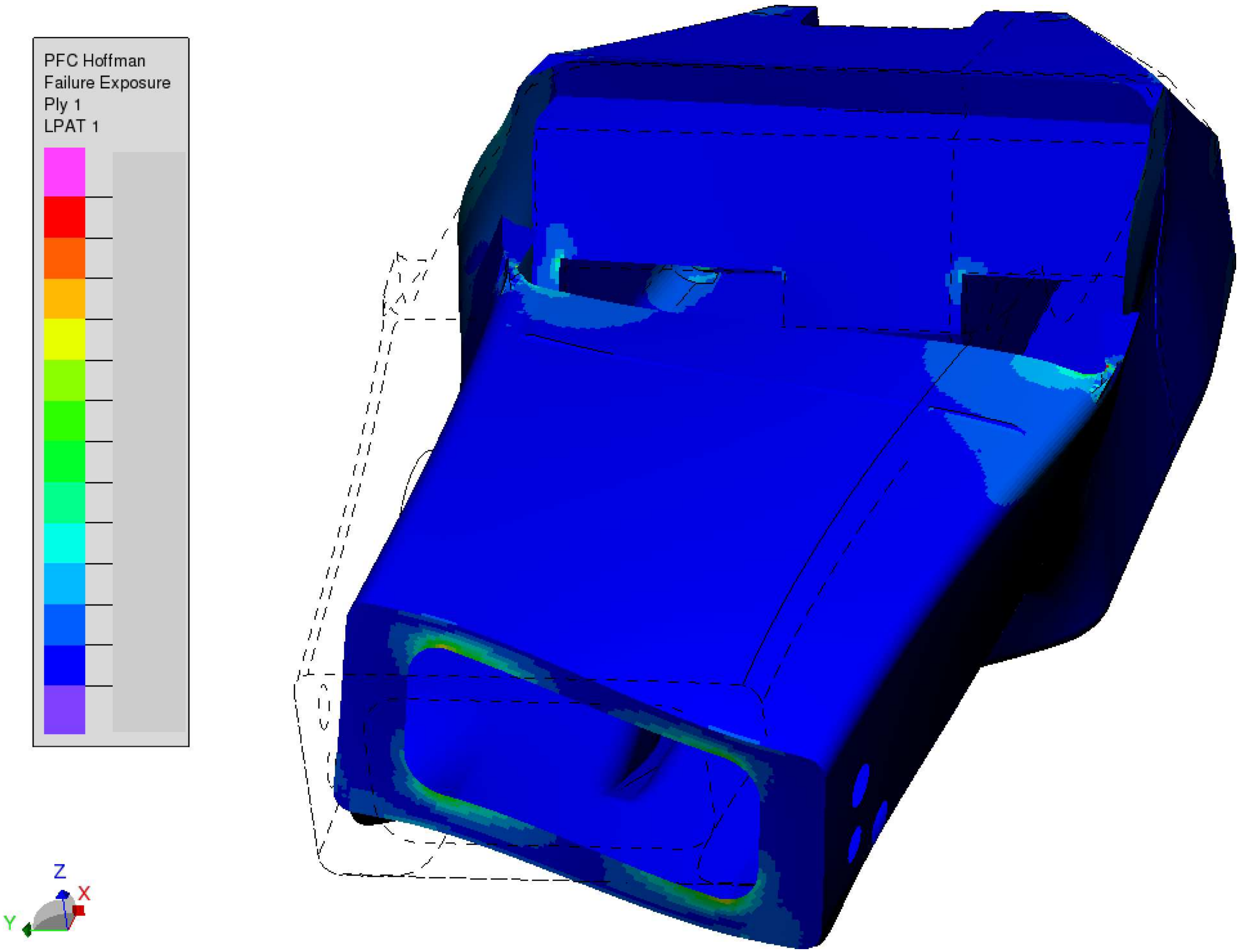
- UCI SELECT Section: $\Longrightarrow$ multiple usages in same TASK possible.
- Calculation of **failure exposure** f : $F_{crit}(R\boldsymbol{\sigma}) \stackrel{!}{=} 1 \quad \Longrightarrow \quad f := 1/R$

$$\textbf{PLYFAILure}\ \ \text{CRITERION} = \left\{ \begin{array}{c} \text{HOFFMAN} \\ \text{TSAI\_WU} \\ \text{USER} \end{array} \right\} \quad \text{PARAM} = pfpname$$

$$[\text{BOOK} = name] \quad [\text{RESULT} = name]$$

- User-defined criteria w.r.t. fibre/inter-fibre exposure via PFCUSRS routine
- Also available in sampling and optimization analyses

PLYFAILure Example: Monocoque



Server: 2x12 cores@2.7GHz; 126 GB

147 065	Nodes
148 040	Elements
11	Max.#ply/elem.
881 814	Local DOFs
4	LPATs
5.27	Disk [GB]
1:07	Elapsed [mm:ss]

⟶ Presentation '*Laminate optimization capabilities applied to a racing car monoqocue*' (M. Kinscher, Chr. Wulf) on Friday !

5 Optimization

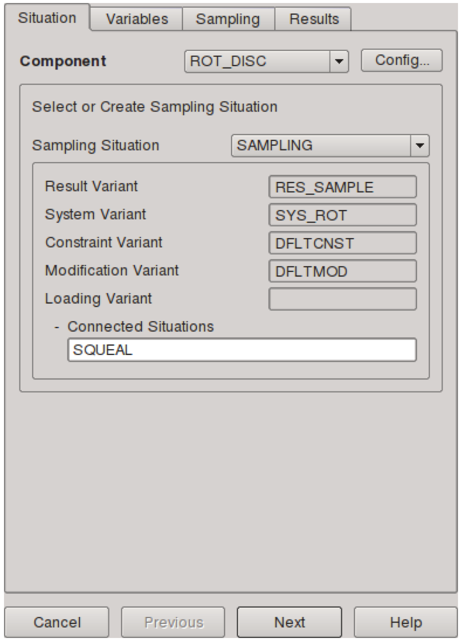
Sampling: Overview

Introduced in PERMAS V15:

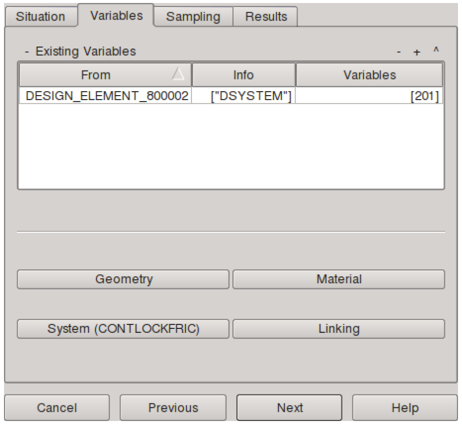
- **Sampling:** Repeated analysis with modified discrete values
- Procedure analogous to optimization or reliability analysis
  - Parameters to be modified as design or basic variables
  - Analysis and sampling steps are grouped in a TASK LOOP
- Explicit definition of discrete values for parameters to be modified

VisPER V6: Sampling Wizard

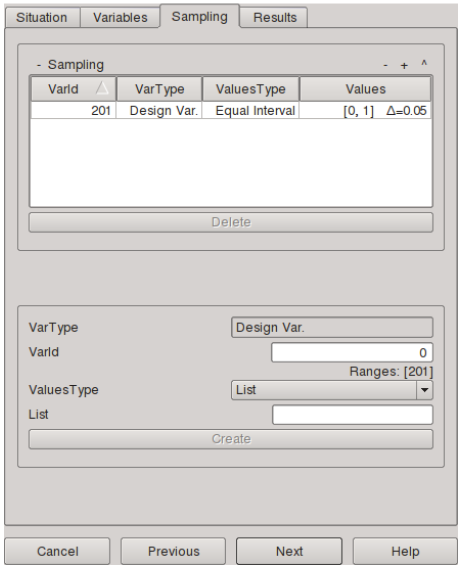
- Set-Up of sampling analyses
- Direct visual feedback
- UCI template for sampling



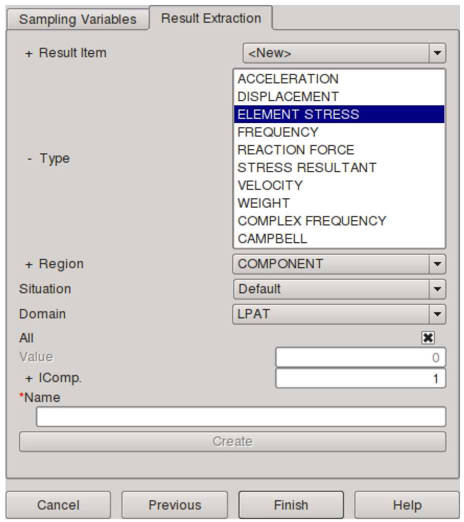
Situations



Variable Data



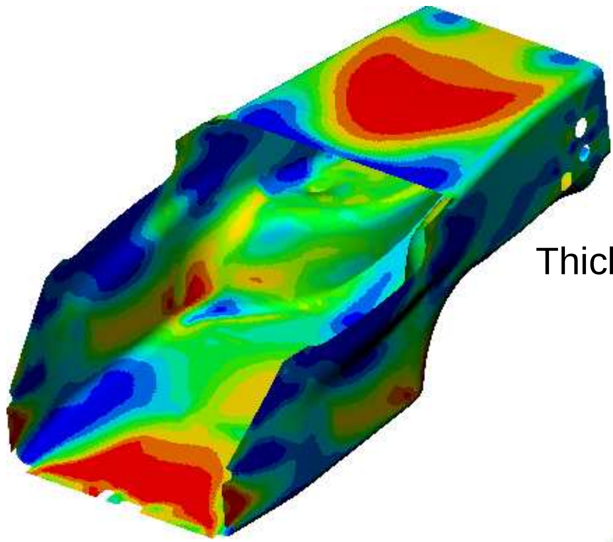
Sampling Points



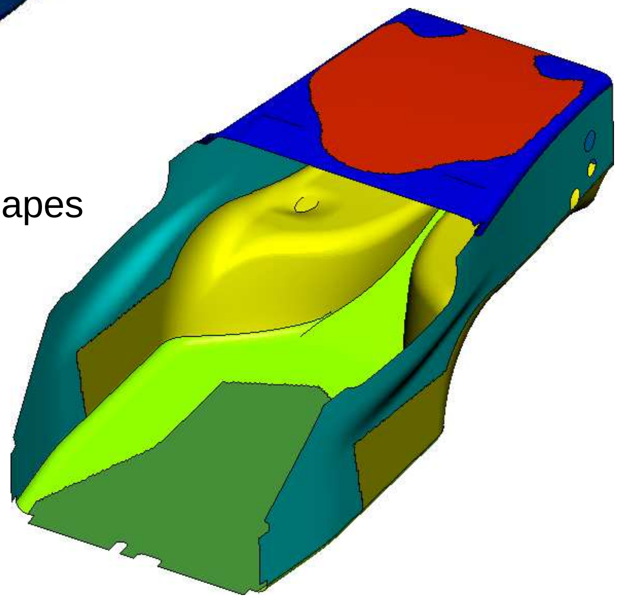
Result Specification

Optimizing Laminates – Free Sizing

- ’**Free Sizing**’ optimization similar to topology optimization, i.e. find new design concepts.
- Thickness of plies with a specific orientation of fiber angles for each element as design variables
  $\longrightarrow$ Generate ply shapes from thickness distributions.
- **$DERESTRAINT TYPE = LAYER**:
  Elementwise formulation of constraints on ply thicknesses, e.g. maximum total thickness over all element layers.
- **$DVTPAR KIND = PLY**:
  Extension of ’Free Sizing’ to laminates.
- UCI **TOOL8** to generate ESET from thickness distributions.



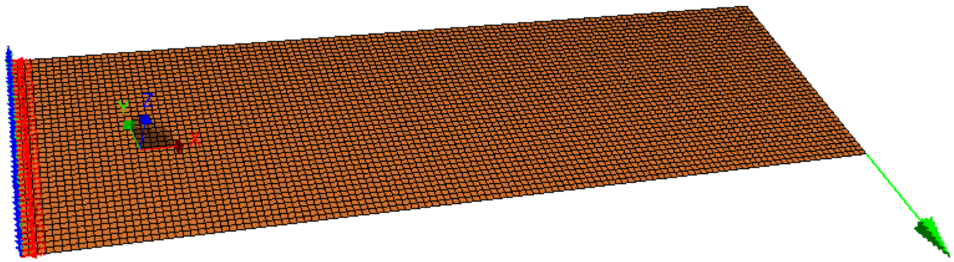
Thickness distribution (per ply)



Ply shapes

Optimizing Laminates – Sizing

- Optimizing of pre-defined laminates, i.e. 'classical' sizing optimization with a moderate number of design variables.



- **$DVMPAR** extension for laminates: ply thickness and angle.

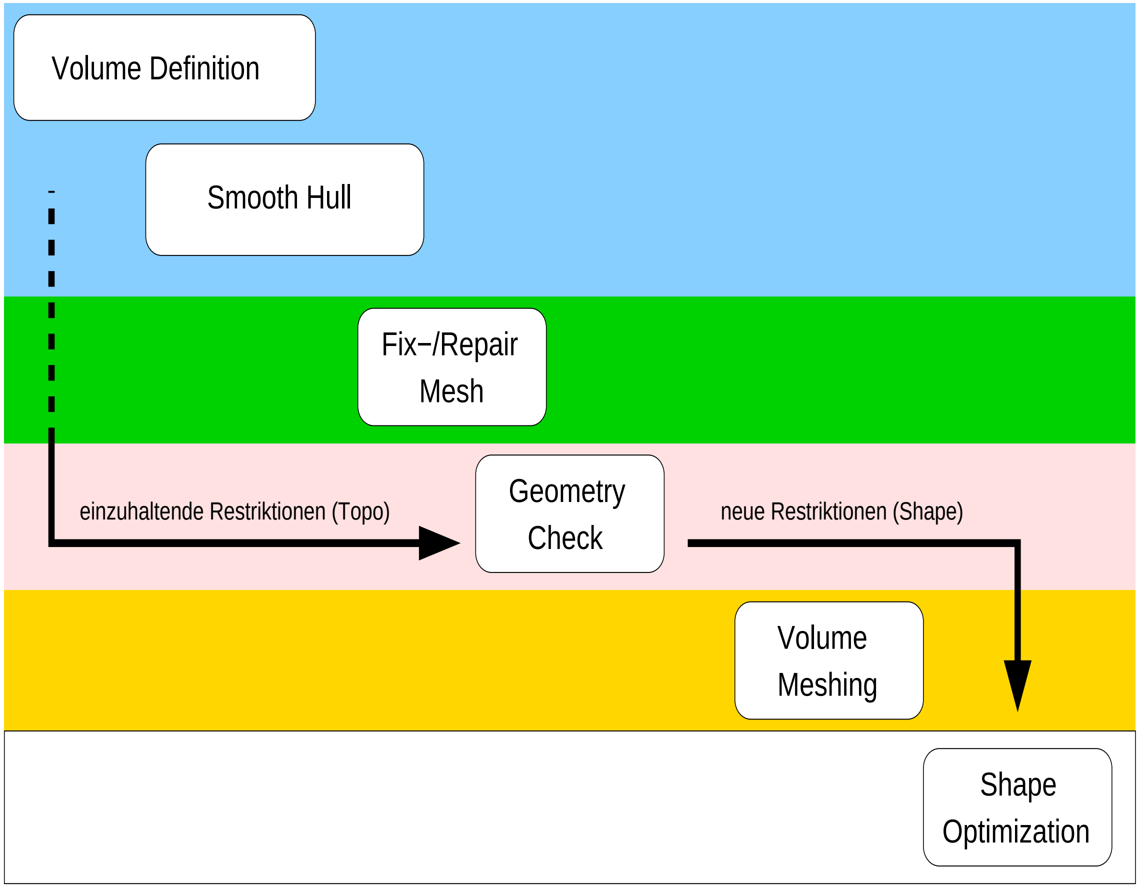
```
$DVMPAR NAME = MAT_198  MODE = SCALE !(default)
     PLY  MAT_201   1          1   : 1.0    : 401
     PLY  MAT_201   2          1   : 1.0    : 402
     PLY  MAT_201   3          1   : 1.0    : 403
     PLY  MAT_201   4          1   : 1.0    : 404
!
$DVMPAR NAME = MAT_198  MODE = SHIFT
     PLY  MAT_201   1          2   : 1.0    : 4011
     PLY  MAT_201   2          2   : 1.0    : 4012
     PLY  MAT_201   3          2   : 1.0    : 4013
     PLY  MAT_201   4          2   : 1.0    : 4014
```

- **$DVMPAR MODE = SHIFT** as new modification rule especially for ply angles.

- $DCONSTRAINT PLYFAILURE as design constraint on ply failure criteria.

⟶ Presentation '*Laminate optimization capabilities applied to a racing car monocoque*' (M. Kinscher, Chr. Wulf) on Friday !

- German Research Project **OptiAMix** (ongoing BMBF project, 2017-2019)
- INTES works on innovative methods for optimization based design for additive manufacturing (AM).
- Important topics:
  - Extension of optimization methods for the needs of AM
  - Capability to check part designs against manufacturing restrictions
  - Provide an integrated workflow from topology to shape optimization



⟶ Presentation '*Efficient Design Creation and Validation*' (R. Fischer) on Friday !

Optimization Miscellaneous

- $RELDIR **SHRINK = BOTH** for design element DFREE, if a mold parting line (PARTLINE) was defined.

- Skip current analysis for TOPO/OPT to allow further parameter adaption (e.g. in case of temporary mesh degeneration).

- New design constraint COORCHANGE (e.g. to limit the superposition of shape basis vectors in parametric shape optimization).

- New UCI switch OPTCMTRACK to control the mode tracking of complex eigenmodes.

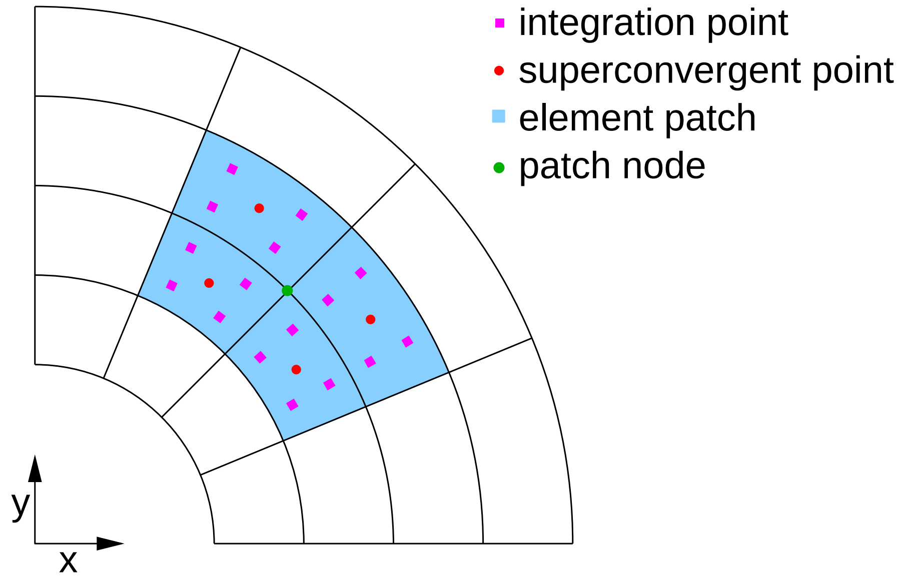
6 Stress Calculation

New Implementation of SPR Stresses

SPR – **Smooth Patch Recovery** method:

- Stress evaluation at superconvergent points (see Zienkiewicz) with nodal value from patch-wise least square fit.

- Stress distribution with higher accuracy: $\sigma^*(\boldsymbol{x}) = N_i(\boldsymbol{x})\,\sigma_i^*$



$\Rightarrow$ Now **applicable for volume, flange, membrane and shell elements** !

- SPR stresses at element nodes, nodal points or as principal SPR stress:

```
GENerate SPR Stress
GENerate Nodal Point STRESS TYPE = SPR
GENerate PRINcipal   STRESS TYPE = SPR
```

New: Absolute Error Indicator (AEI)

Stress based, **absolute** error indicator:

```
GENerate AEI
```

(former strains based, relative ERROR INDICATOR still existing, too)

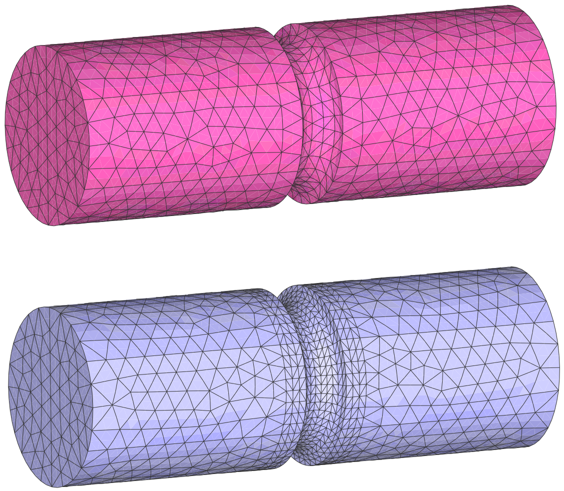
- *Absolute* means that the result is given in absolute stress units as indicator for discretisation errors or singularities respectively.

- Definition:

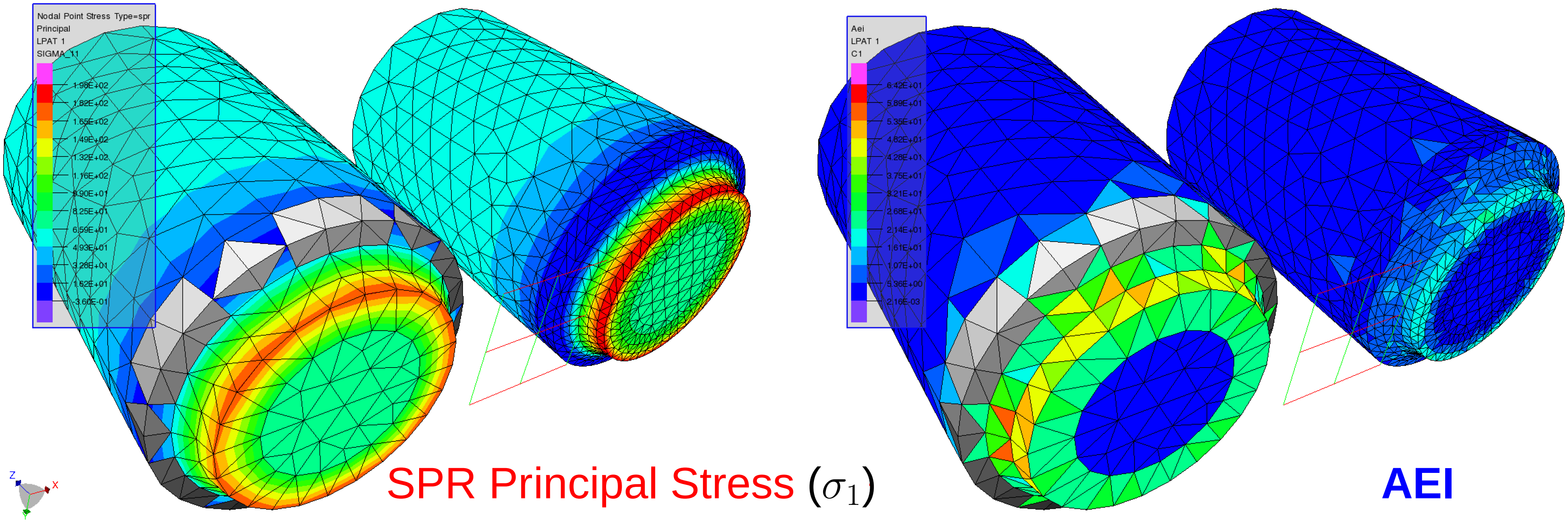
$$\sigma_{\text{AEI}} = \max_{V} \left\| \boldsymbol{\sigma}_{\text{FE}} - \boldsymbol{\sigma}_{\text{SPR}} \right\| ,$$

(Frobenius norm of differences between classic FE and higher accuracy SPR stress calculation)

LS19: SPR Stress and AEI for Volumes

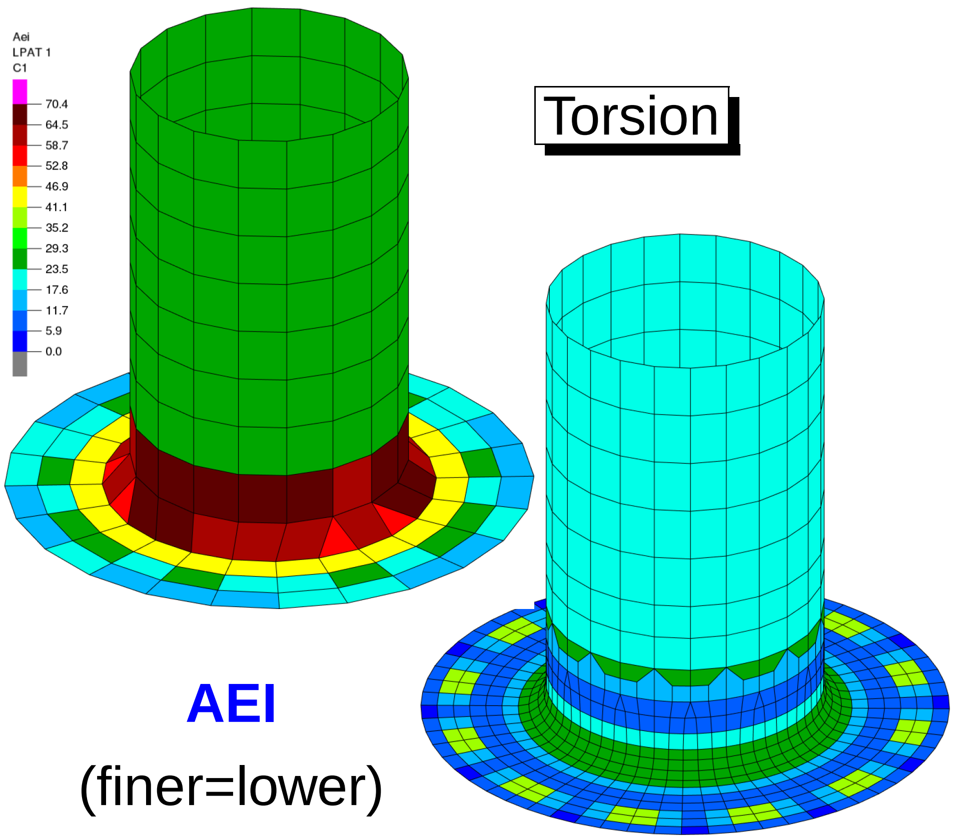


- Nick break test under tension load
- Two discretizations (coarse/fine)
- TETs with 2 hull faces get no SPR/AEI (SPR needs $\geq$1 interior edge node)



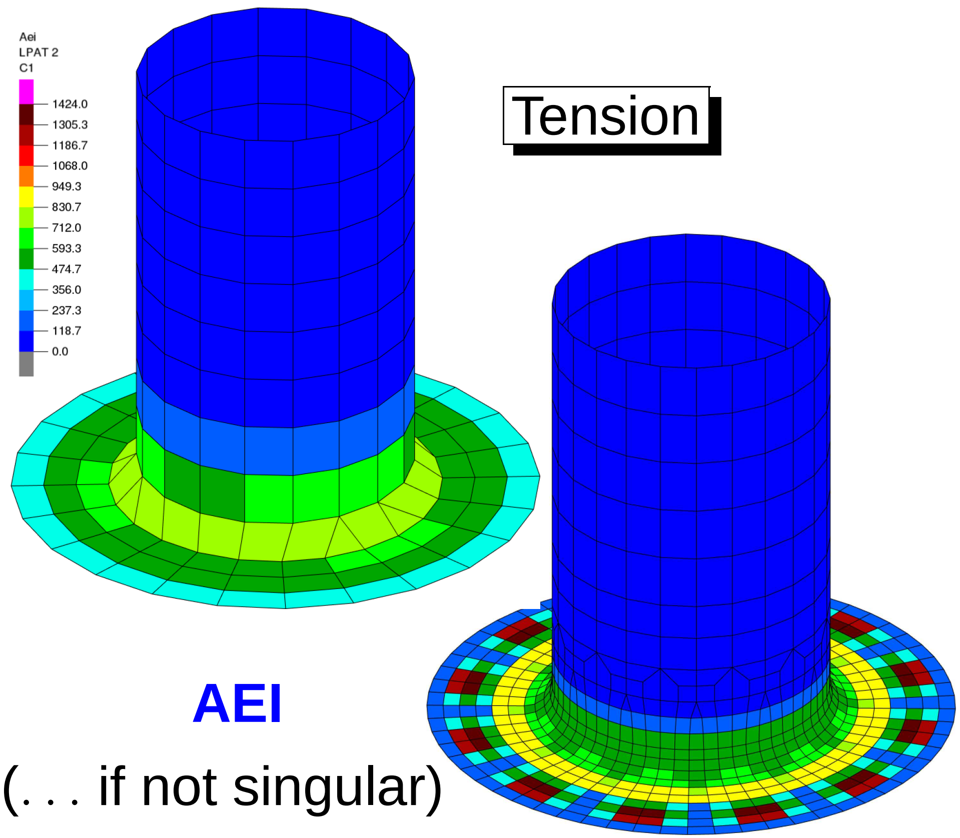
LS20: SPR Stress and AEI for Shells

- Flange tube under torque and tension
- Two discretizations (coarse/fine)



AEI

(finer=lower)



AEI

(. . . if not singular)

Stress Calculations Miscellaneous

- Reliable Stress Gradients, e.g. for Lifetime Analysis:

```
GENerate STRESS GRADient SURFID = 1001
GENerate PRINcipal STRESS TYPE = SPRGRAD
```

  - Gradient direction defined by surface normal
  - In V17 with SPR at (element) nodes
    (in V16 only as result at element center)

  $\Rightarrow$ Much better than building simple finite differences !

- Signed nodal point stress mises (σ_v with sign of σ_{max}):

```
GENerate Nodal Point STRESS MISES   VALUES=SIGNED
```

- Principal Stress/Strain without direction:

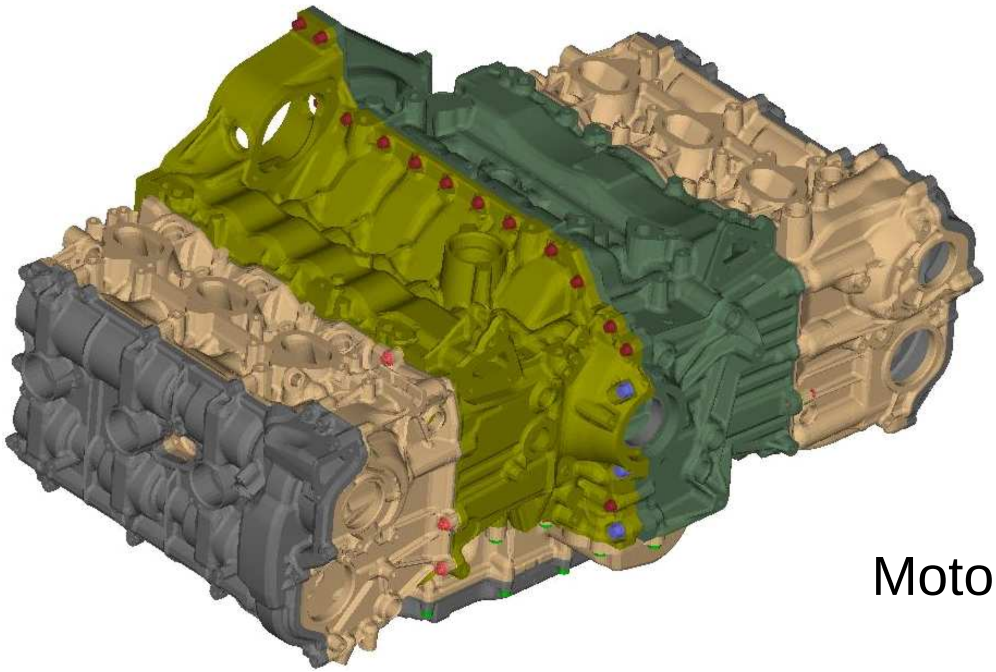
```
GENerate PRINcipal STRESS   VALUES=SCALAR
```

7 Performance

Parallel Stress/Strain Calculation

- Linear element stress & strain calculation now parallel.
- Some nodal point stress calculations also parallelized (e.g. principal stress).

⇒ Improved Speed-Up for large models and/or models with many right-hand-sides (e.g. with $NLLOAD).

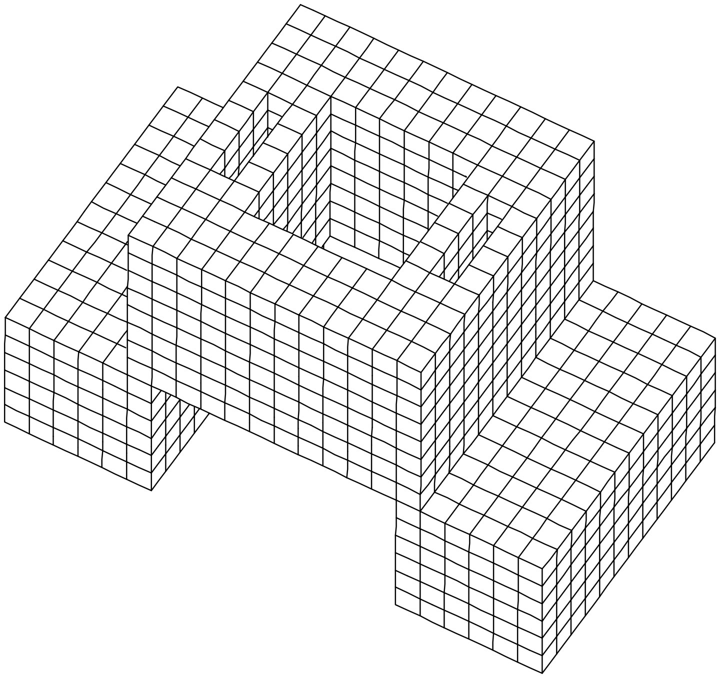


[sec]	V16	V17	SpUp
EPST	1600	468	3.42
SIGVM	156	43	3.63
GRLEXS	1756	**511**	3.44

Motor model (56 MDOFs) on 4*Xeon 8180

Tipp: Also use fast HDF export, e.g. 6555 (BOF) ⟶ **1218** [sec] (HDF).

Parallel Stress/Strain Calculation
Temperature Loads and $NLLOAD



Artificial Block:

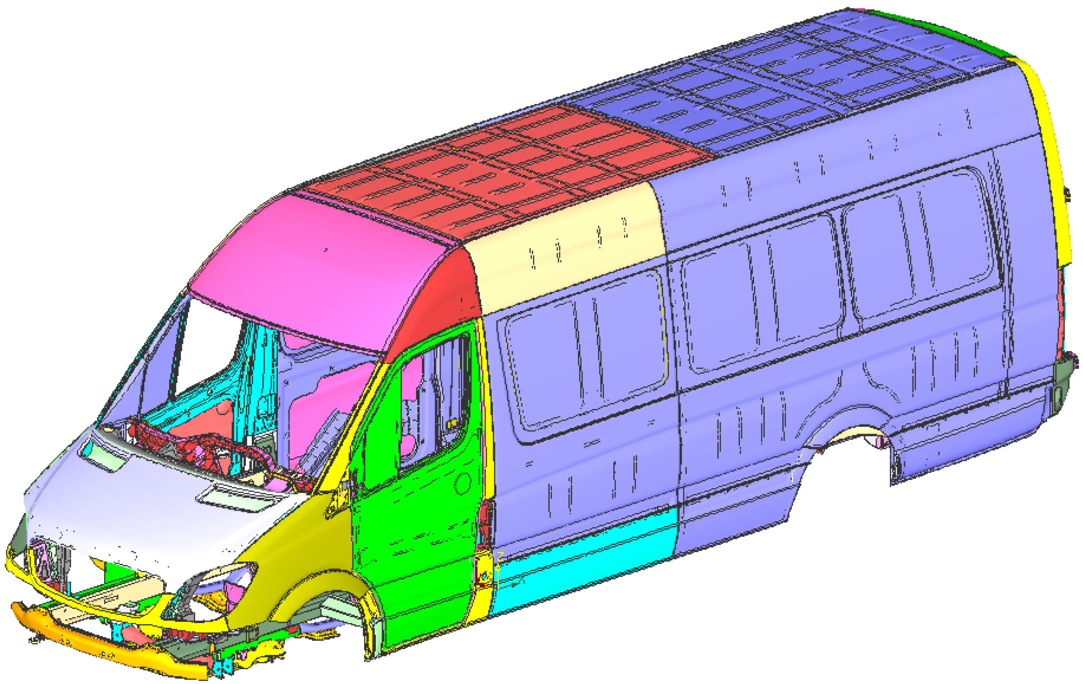
13 600 000 DOFs
4 300 000 Elements (HEXE8)
Temperature load with $\alpha(T)$

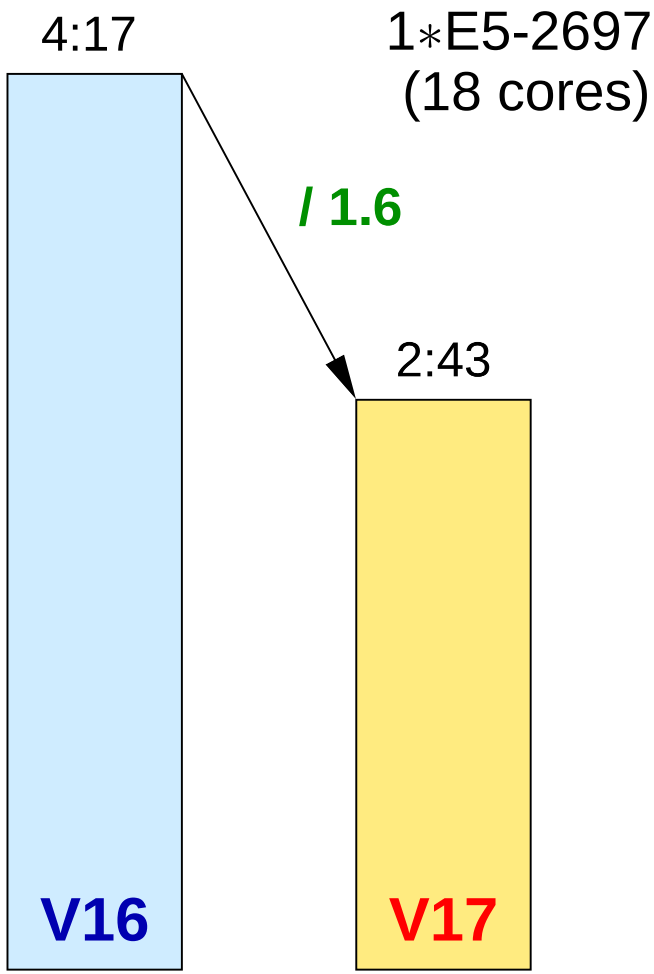
on 4*Xeon 8180

```
task
 exec
  static
   gen el stress
   gen el stress mises
   gen nodal p stress
   gen prin stress
```

10 LPATs no $NLLOAD			10 LPATs 100 Time Steps			10 LPATs 200 Time Steps		
V16	V17	SpUp	V16	V17	SpUp	V16	V17	SpUp
845	92	9.18	1644	110	14.95	**2234**	**132**	**16.92**
30	17	1.76	99	29	3.41	176	40	4.40
30	27	1.11	163	146	1.11	301	286	1.05
17	2	8.50	182	13	14.00	**372**	**25**	**14.88**
(incore)			(almost incore)			(moderate I/O)		

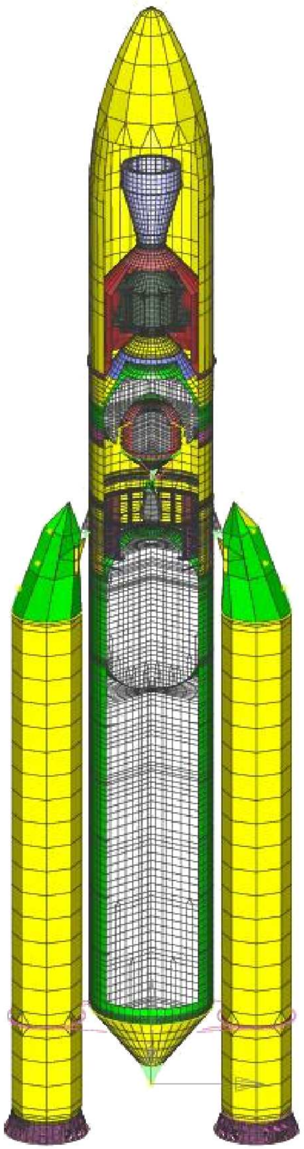
Faster Response Analysis (including MLDR accelerations)





Nodes	4 800 000	
Modes	11 350	(Freqlim=600.0)
Load cases	54	

Generalized Inertia Relief
E.g. Rocket in Flight



New solution scheme without explicit G0 matrix:

$$\boldsymbol{X_s} = \left(\boldsymbol{I} - \boldsymbol{G}_0\boldsymbol{K_a}\right)^{-1}\boldsymbol{G}_0\boldsymbol{F}$$

- For GINERT with SOLV=ITER (and SOLV=DIRECT with K_a=0).
- Huge time & disk savings (16 cores + XPU):

V16:	Several days and $>$15 TiB
V17:	$\longrightarrow$ 14 hours and $<$ 2 TiB

Nodes:	2 900 000	
Unknowns:	10 600 000	(DISP)
	700 000	(PRES)

GPU Accelerators – XPU Module

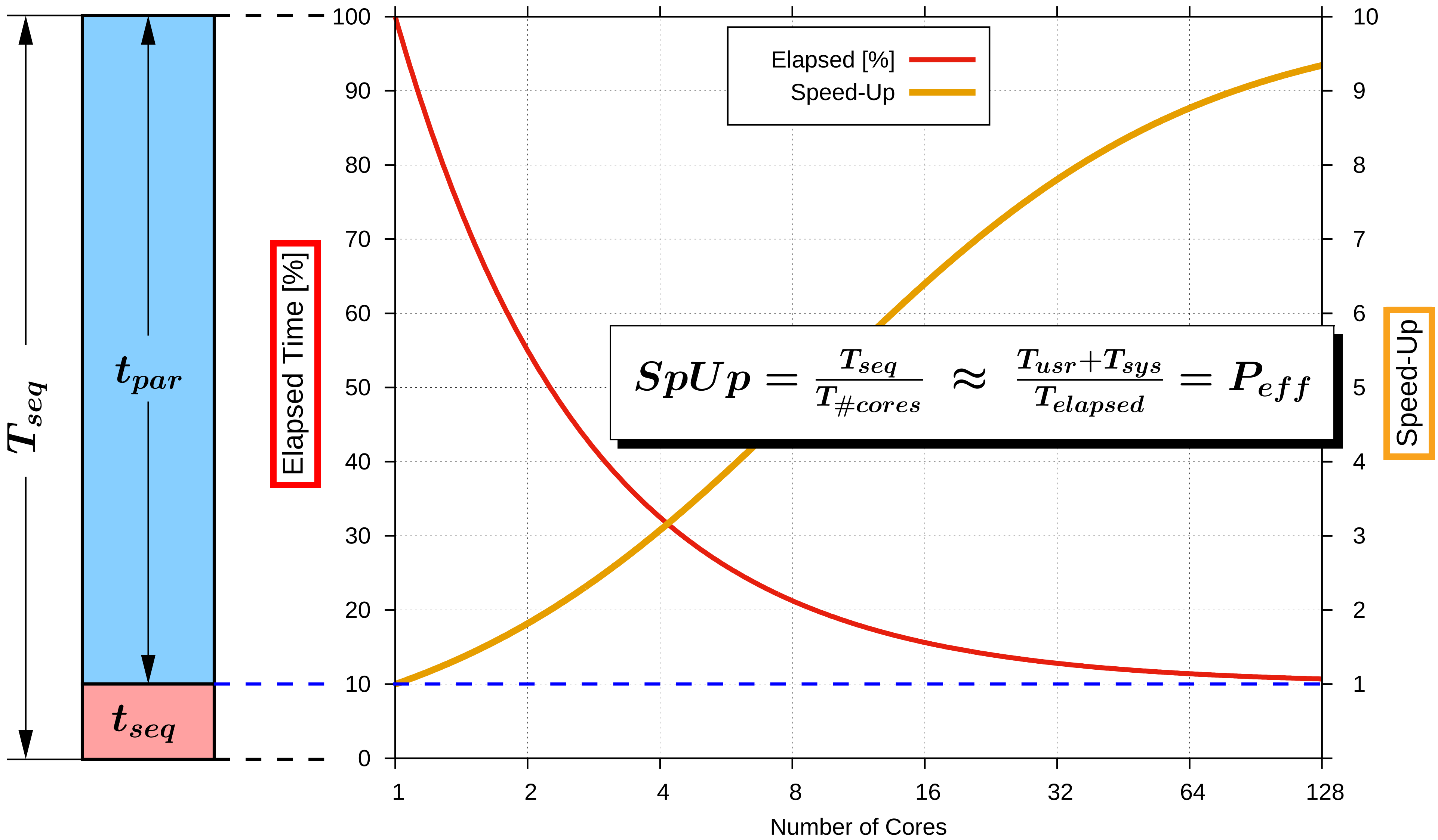
- Support of **Nvidia Tesla K20/40/80** [c/m/x]
  and new: **Nvidia Tesla P100** [c/m/x]

- PERMAS V17 with statically linked CUDA library:
  Client needs Nvidia drivers only (but not CUDA library any more)

- PERMAS V17 available for NVIDIA CUDA-7.0, 8.0 and 9.0
  (Tesla P100 needs CUDA-8.0 for full performance)

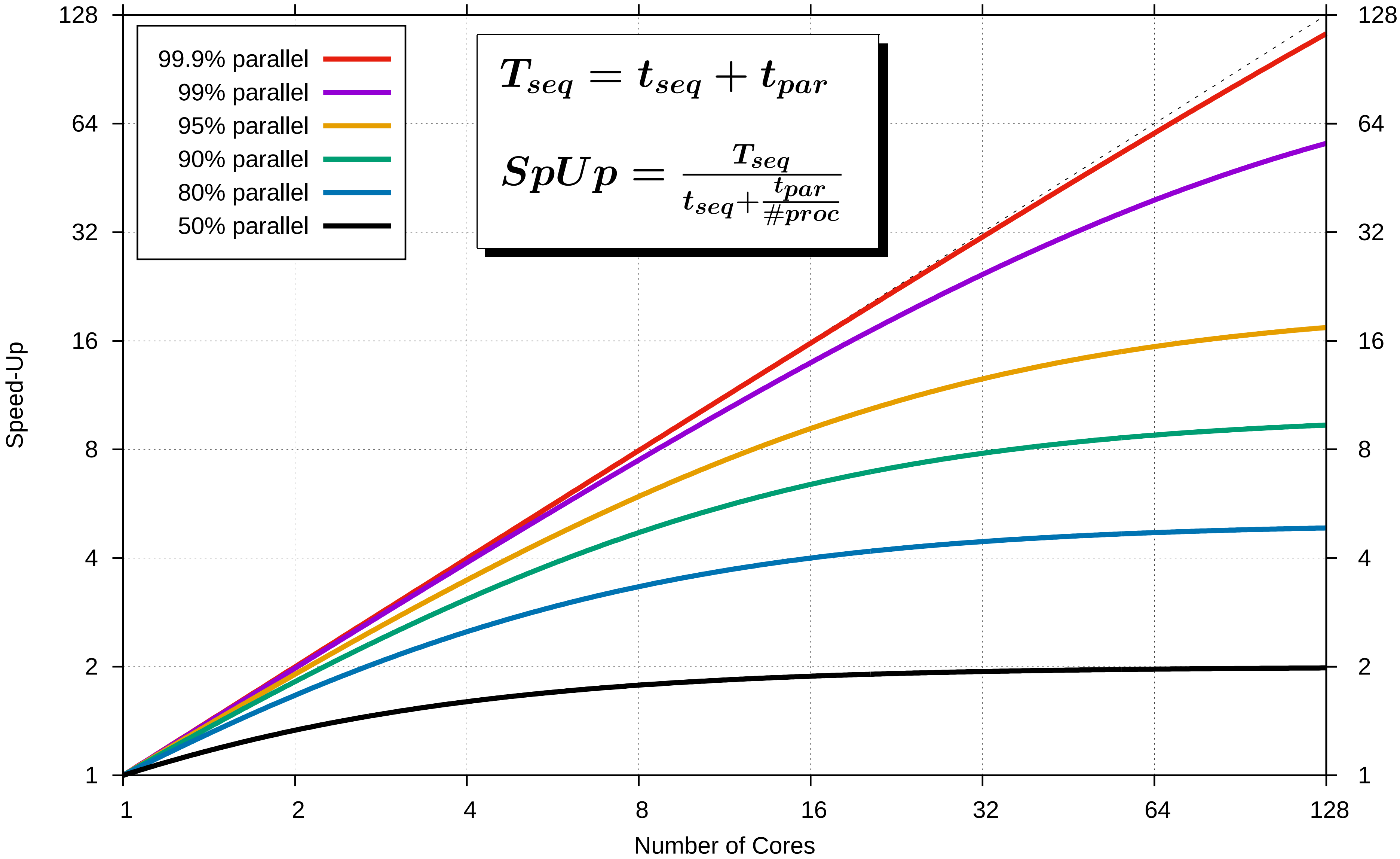
- PERMAS V17 can **monitor the GPU's temperature and** warns if the accelerator is not running with maximum possible **clock rate**.

Parallel Efficiency
(Elapsed Time, Cores & Speed-Up)



Fighting Amdahl's Law

How many cores are useful ?



Contact Analysis with 112 Cores
(Motor Model with Gaskets, 56 MDOFs)

PSUM: Profile statistics for Steps & Processors

calls	Step/Proc	Elapsed	%part,total	User+Sys	%part,total	pEff	I/O[GiB]	GFlp/s
1	SYLECA	7283	(42.1)	685306	(63.5)	94.10	576	1330.4
1	.HPFWS	5764	(79.1,33.3)	562374	(82.1,52.1)	97.56	575	862.2
2	.HYMULT	1517	(20.8, 8.8)	122917	(17.9,11.4)	81.05	1	3110.1
1	SYACBK	3910	(22.6)	288259	(26.7)	73.72	1385	1140.5
1	.HYCHOLA	3834	(98.1,22.2)	287801	(99.8,26.6)	75.06	1385	1162.1
1	SLLECA	1224	(7.1)	65891	(6.1)	53.85	1976	716.8
4	.CASREC	737	(60.2, 4.3)	61927	(94.0, 5.7)	84.05	1	1129.8
9	..CASOLVA	577	(78.3, 3.3)	49857	(80.5, 4.6)	86.43	0	1418.0
6	...HYCHOLA	441	(76.4, 2.6)	38344	(76.9, 3.6)	86.98	0	1852.6
1	CSMSCN	840	(4.9)	29121	(2.7)	34.65	0	-
1	.HFSOPTN	324	(38.6, 1.9)	28594	(98.2, 2.6)	88.22	0	-
1	LPLEA0	832	(4.8)	1719	(0.2)	2.06	0	-
1	.EPLOAD	570	(68.4, 3.3)	568	(33.1, 0.1)	1.00	0	-
1	GRLEXS	616	(3.6)	988	(0.1)	1.60	64	8.4
1	.EPST	517	(83.9, 3.0)	510	(51.6, 0.0)	0.99	38	-
:		:		:		:	:	:
	Sum	17249 [sec]	99.8%	1079932 [sec]	100.0%	62.61	6978	1096.9
	Total	17287 [sec]		1080011 [sec]		62.48	6978	1097.6

V16.10.009 on lightning with 112 Exec & 1 I/O-Thread (4 of 4 slots)

8 Summary

Outlook
Next Release and Future Versions

- PERMAS **V17** and VisPER **V6** release in June 2018 !

- PERMAS V18 and VisPER **V18** in 2020
  (VisPER as integral part of PERMAS system, i.e. 'just a module'):

2018		Develop		2020
PERMAS V17	$\Longrightarrow$	V17.10	$\Longrightarrow$	PERMAS V18
VisPER V6	$\Longrightarrow$	V6.1 $\rightarrow$ V17.10	$\Longrightarrow$	VisPER V18

Outlook
Future Topics

From current perspective the next developments will – amongst others – tackle the following topics:

- Further improvement of simulation workflows
- Extended nonlinear capabilities (e.g. large strains)
- Acceleration of large contact models
- Ongoing leading edge capabilities in dynamics
- Support of additive manufacturing
- etc.

Summary
PERMAS V17 & VisPER V6

- **PERMAS V17** – and **VisPER V6** – offer
  - A **bunch of new features** and analyses
  - **best class performance** and parallelization
  - High interoperability with other simulation tools
  - **Tailor made wizards** allow easy use of expert knowledge.
  - More comfort by more powerful general commands
  - Generalization improves reliability of simulation work-flow.

  $\Rightarrow$ Ongoing innovations protect client's investments.

- For details and for a full list of features:
  - Visit **V17-Upgrade Workshops** at INTES or at client's site.
  - Look at V17 Release Notes (June 2018).
  - Explore new training examples in UM550 Example Manual.