What's New
MSC Nastran 2018 & MSC Nastran 2019

Bias Mühendislik
Bahadır GÜRSOY

03/10/2019

Outline

- Element Technology
  I. Pyramid Elements
- Contact Analyses
  I. Contact Model Check
- Numerical Computation and HPC Improvements
  I. Automatic Solver and Parallel Selection Process
- Other Improvements
  I. Multi-Mass Configuration
  II. Shell Stress Constraints for Topology and Topometry Optimization
  III. HDF5 Result Database

Element Technology – Pyramid Elements

Introduction

- Pyramid element (CPYRAM) capability added to MSC NASTRAN
- Extends existing solid element topology CTETRA, CHEXA, CPENTA
- Supported in all physics/applications where regular solid elements are supported

Benefits

Enhances automated meshers transition between CHEXA & CTETRA

- CHEXA at the core of geometry
- CTETRA at the boundary of geometry
- CPYRAM can now be used to transition between these elements

Feature Description

- CPYRAM supported in all linear and nonlinear solution sequences
  - SOL 101, SOL 103, SOL 105, SOL 107, SOL 108, SOL 109, SOL 110, SOL 111, SOL 112, SOL 144, SOL 145 and SOL 146
  - SOL 200
  - SOL 400
- All physics already supported by MSC NASTRAN standard solid elements are also supported by CPYRAM element
  - Linear statics
  - Linear dynamics (modal frequency and transient)
  - Buckling analysis
  - DesigFatigue analysis
  - Rotordynamics
  - Vibro-Acoustics
  - Aeroelasticity and Flutter
  - n Sensitivity and Optimization
  - Contact including automated contact generation
  - Material and geometric nonlinearity
  - Linear perturbation analysis
  - Random analysis
  - Coupled analysis
  - Heat transfer
  - Thermo-mechanical coupled analysis
- All types of loading like structural, distributed, gravity, pressure, thermal loads are supported
- Pyramid element results output to OP2 and HDF5 files

User Interface

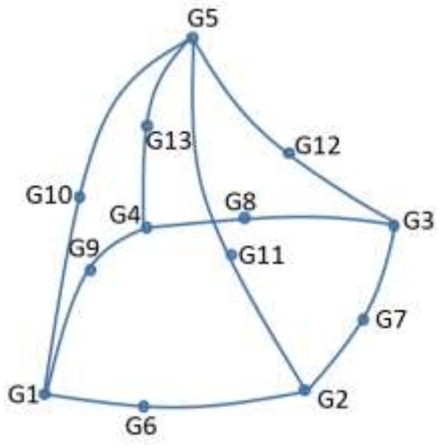
- **CPYRAM Five-Sided Solid Element Connection**

  Defines connections of the five-sided solid element with five **or** thirteen grid points

  Format:

1	2	3	4	5	6	7	8	9	10
CPYRAM	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13		

Describer	Meaning	Type	Default
EID	Element identification number	0 < Integer < 100,000,000	Required
PID	Property identification number of a PSOLID entry	Integer > 0	Required
G_i	Identification numbers of connected grid points	Integer ≥ 0 or blank	Required

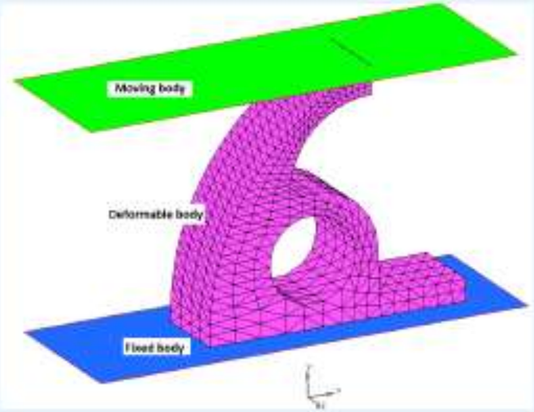


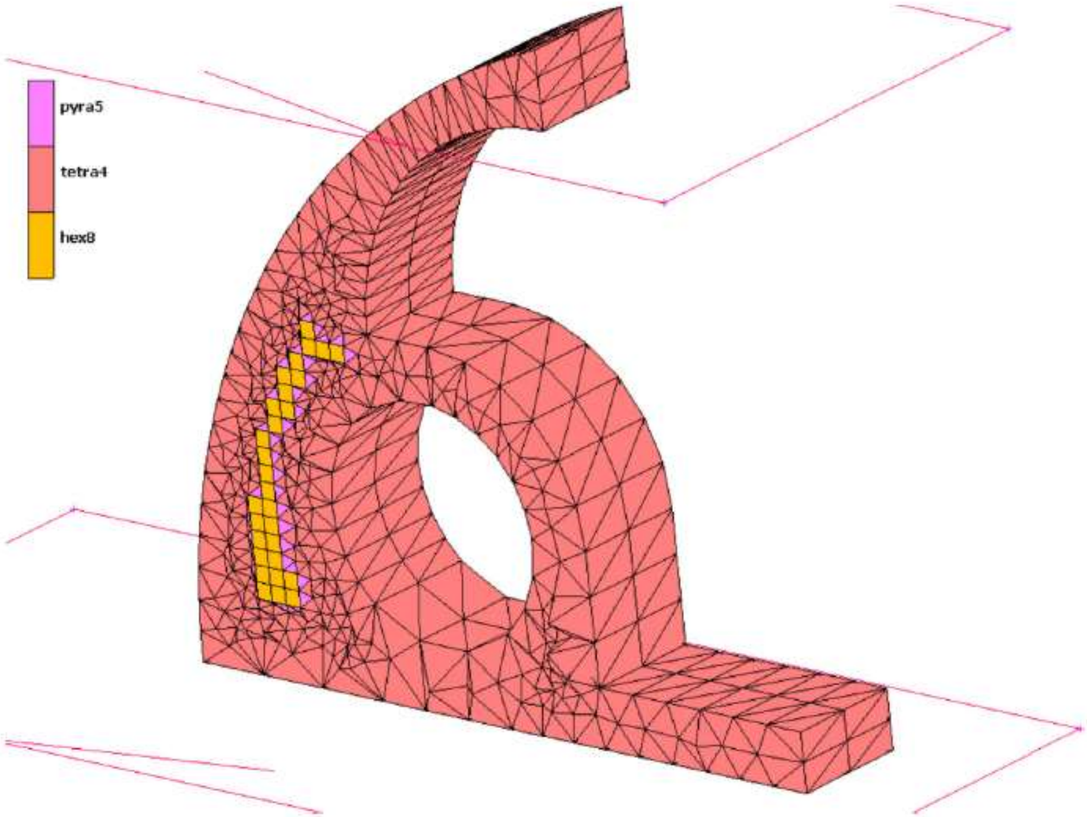
- G1, G2, G3, G4 should define base of quadrilateral, G5 apex
- 5-noded Pyramid, G1-G5 must be defined
- 13-noded Pyramid, all G1-G13 must be defined
- Element coordinate system is used as basic CSYS

Pyramid Elements: Example

Closure of rubber seal - hybrid mesh

Mesh consists of CTETRA, CHEXA & CPYRAM elements
CPYRAM used as transition element

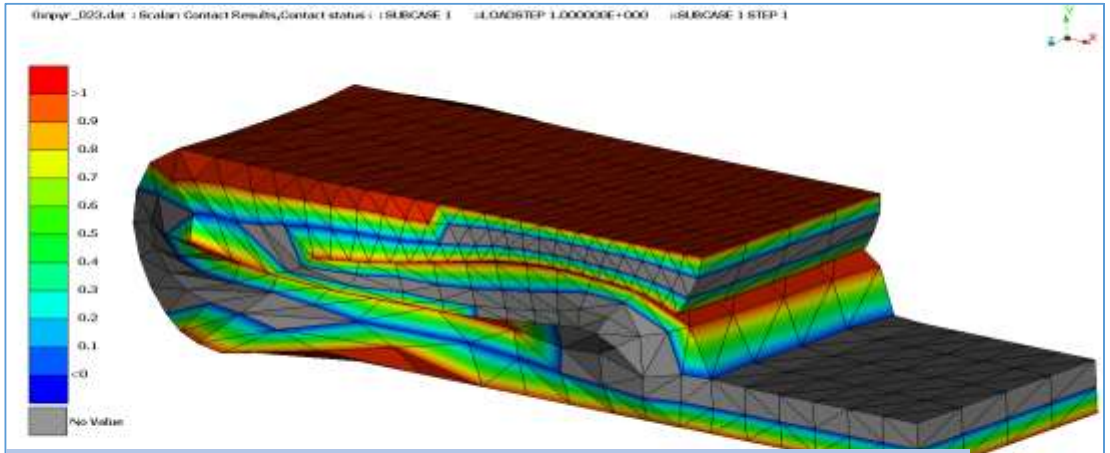
Title	Chapter 50: Closure of a Rubber Seal using Segment-to-Segment Contact with Friction
Features	Segment-to-Segment Contact, Large Sliding and Friction
FE Mesh	
Material Properties	Material for deformable body Neo-Hookean material model defined through the Mooney property menu with $C_{10} = 100$
Analysis Characteristics	Nonlinear static analysis
Boundary conditions	Moving rigid body is moved downward by 200 mm in -Y direction
Applied loads	All the loads have been applied to moving rigid body by activating position controlled method in contact body creation option.



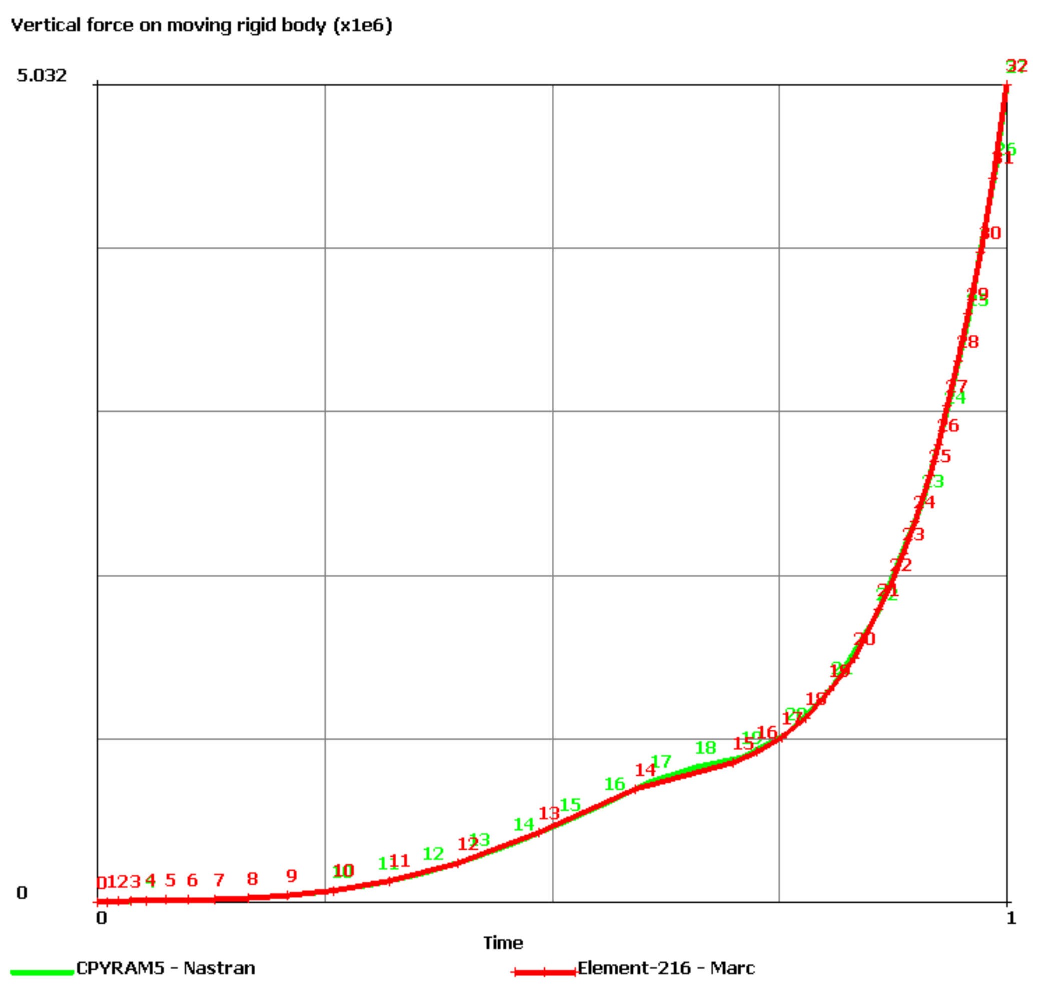
Pyramid Elements: Example

Closure of rubber seal - hybrid mesh

- Deformable rubber seal is placed between two rigid plates, one of which is moved downwards
- The example problem is run in MSC Nastran using 5-noded transitional Pyramid Elements
- Results are compared with MSC Marc



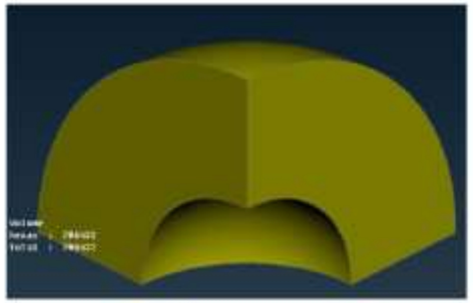
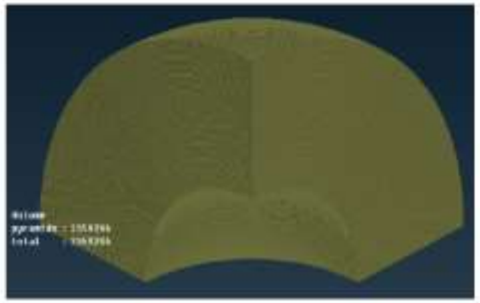
Closure of Rubber Seal simulated in MSC Nastran



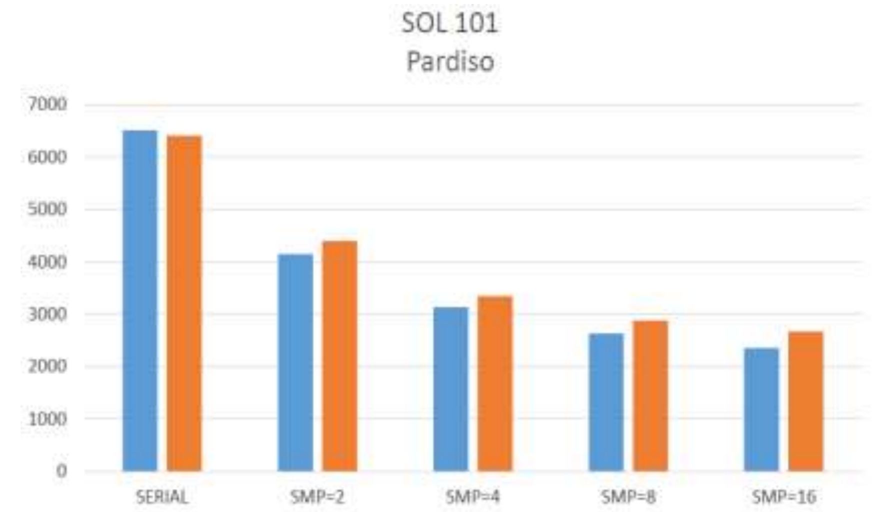
Results are consistent with inclusion of CPYRAM elements

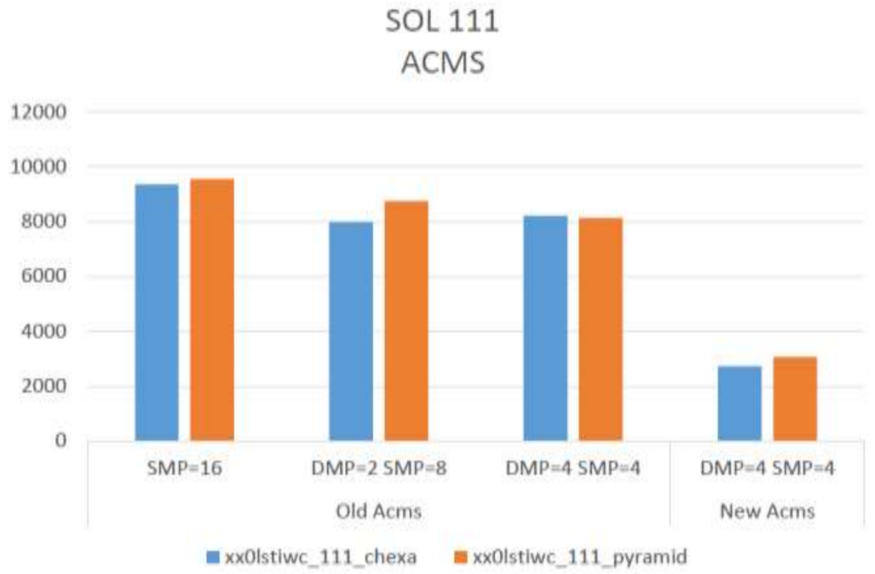
Pyramid Elements: Performance Testing

- All CHEXA element model turned into all CPYRAM model
- 1 CHEXA element converted to 3 CPYRAM elements
- Tested in different SOL sequences & solvers: SOL 101, 103, 108, 111 & 400 and solvers ACMS, PARDISO, CASI, and MSCLDL

xx0lstiwc_chexa Baseline : DOF 4867590	xx0lstiwc_pyramid Pyramid Element : DOF 4867590
	
GRID Entities = 811265 Parts = 1	GRID Entities = 811265 Parts = 1
Solid Elements = 786432 100.0% CHEXA8 Elements = 786432 100.0%	Solid Elements = 2359296 100.0% CPYRAM Elements = 2359296 100.0%
Elements = 786432	Elements = 2359296

Consistent performance when introducing CPYRAM elements





Guidelines and Limitations

- First order linear Pyramid element may show excessively stiff behaviour for bending dominated test problems
  - Drawback of shear locking will not be an issue, as Pyramid elements are mainly used as transition elements in most real world applications
- It should also be noted that second order Pyramid elements behave well for bending problems without any shear locking issue
- Damping (GE / GEij on the Material cards) is supported for Pyramid element
- Digimat support via MATDIGI material entry is not available for the Pyramid element
- Patran will support Pyramid Element in v2020.0
- Preprocessor Ansa (Beta CAE) will support the MSC Nastran Pyramid Element in their next major release as well

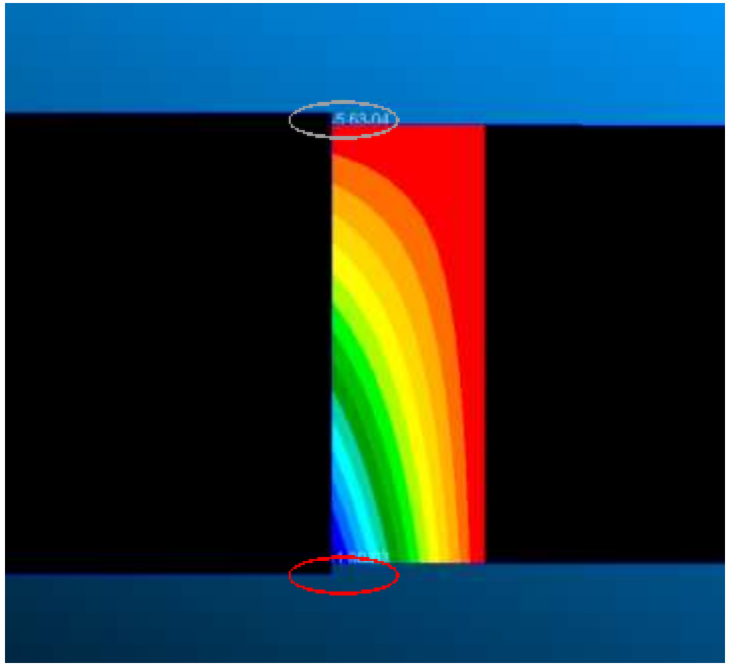
Contact Analyses – Contact Model Check

Contact Model Check Phase I

- **Contact grid status check**
  - Check grid-wise distance between contact pairs
- **Geometry adjustment display of initial stress free contact**
  - Generate Displacement Output Table for post-processing
- **Analytical SPLINE output OF Contact SURFACE**
  - Add a new output table /OBCNURB/ for Analytical Smoothed Surface
- **Bias = 0.0 on BCONPRG/BCTABLE**
  - Enable exact user input 0.0 for BIAS on BCONPRG/BCTABLE

Contact Grid Status Check (BCONCHK)

- Provides the status of contact grids within a given tolerance
  - Help users check the contact status
  - Print the output to the f06 file and / or a post-processing file
  - Stop or run through after contact model check



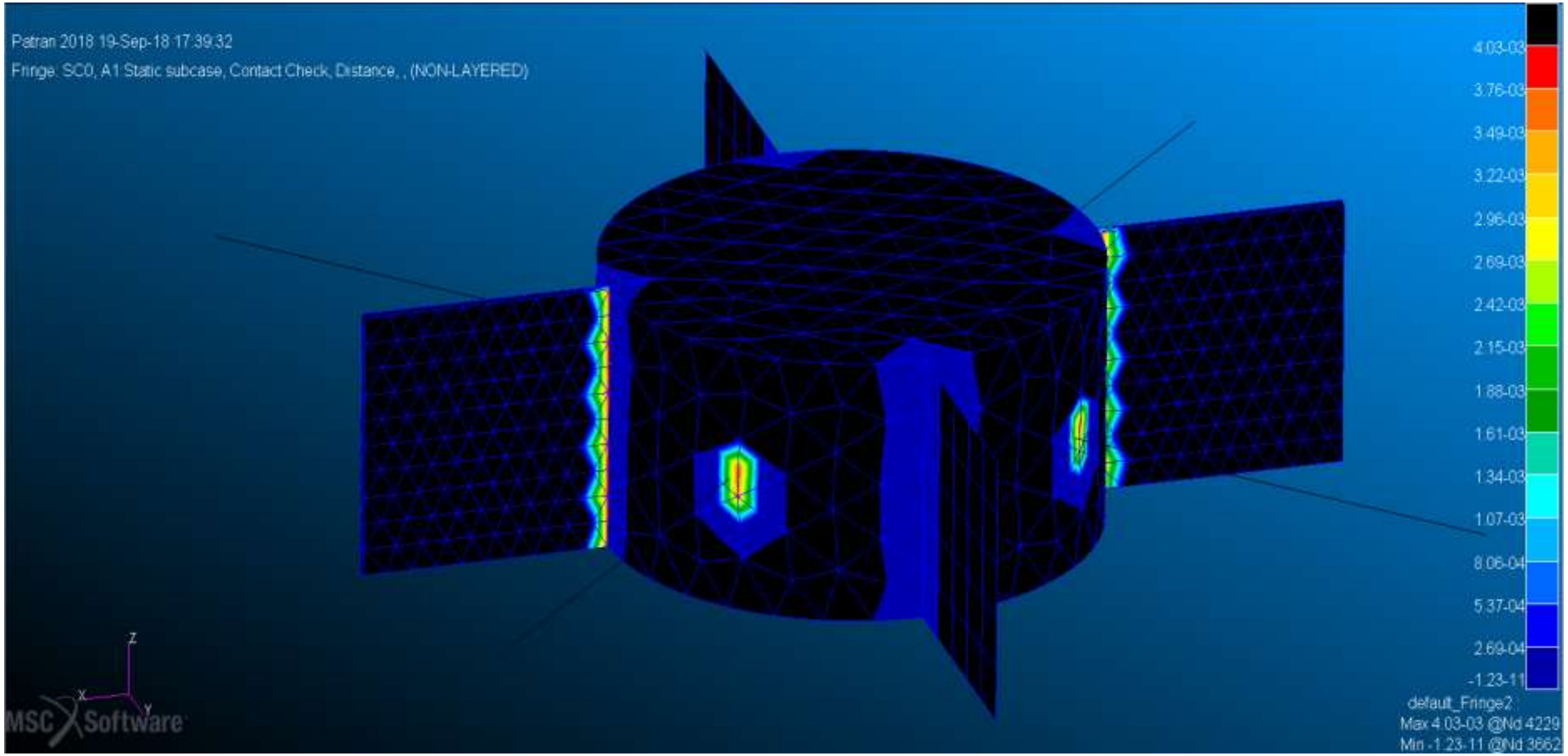
C O N T A C T   G R I D   S T A T U S   C H E C K

TIME = 0.00000E+00 SUBCASE

SLAVE NODE	MASTER ELEM	STATUS	T1	T2	T3	CONTACT DISTANCE
			-	IN BASIC SYSTEM	-	
77	1	PENETRATE	0.0000E+00	0.0000E+00	1.8016E-02	-1.8016E-02
78	1	PENETRATE	0.0000E+00	0.0000E+00	1.8016E-02	-1.8016E-02
79	2	PENETRATE	0.0000E+00	0.0000E+00	1.8016E-02	-1.8016E-02
80	3	PENETRATE	0.0000E+00	0.0000E+00	1.8016E-02	-1.8016E-02
93	1	PENETRATE	0.0000E+00	0.0000E+00	1.2198E-02	-1.2198E-02
94	1	PENETRATE	0.0000E+00	0.0000E+00	1.2198E-02	-1.2198E-02
95	2	PENETRATE	0.0000E+00	0.0000E+00	1.2198E-02	-1.2198E-02
96	3	PENETRATE	0.0000E+00	0.0000E+00	1.2198E-02	-1.2198E-02
109	10	PENETRATE	0.0000E+00	0.0000E+00	6.3807E-03	-6.3807E-03
110	10	PENETRATE	0.0000E+00	0.0000E+00	6.3807E-03	-6.3807E-03
111	11	PENETRATE	0.0000E+00	0.0000E+00	6.3807E-03	-6.3807E-03
112	12	PENETRATE	0.0000E+00	0.0000E+00	6.3807E-03	-6.3807E-03
125	19	PENETRATE	0.0000E+00	0.0000E+00	5.6290E-04	-5.6290E-04
126	19	PENETRATE	0.0000E+00	0.0000E+00	5.6290E-04	-5.6290E-04
127	20	PENETRATE	0.0000E+00	0.0000E+00	5.6290E-04	-5.6290E-04
1280	21	PENETRATE	0.0000E+00	0.0000E+00	5.6290E-04	-5.6290E-04

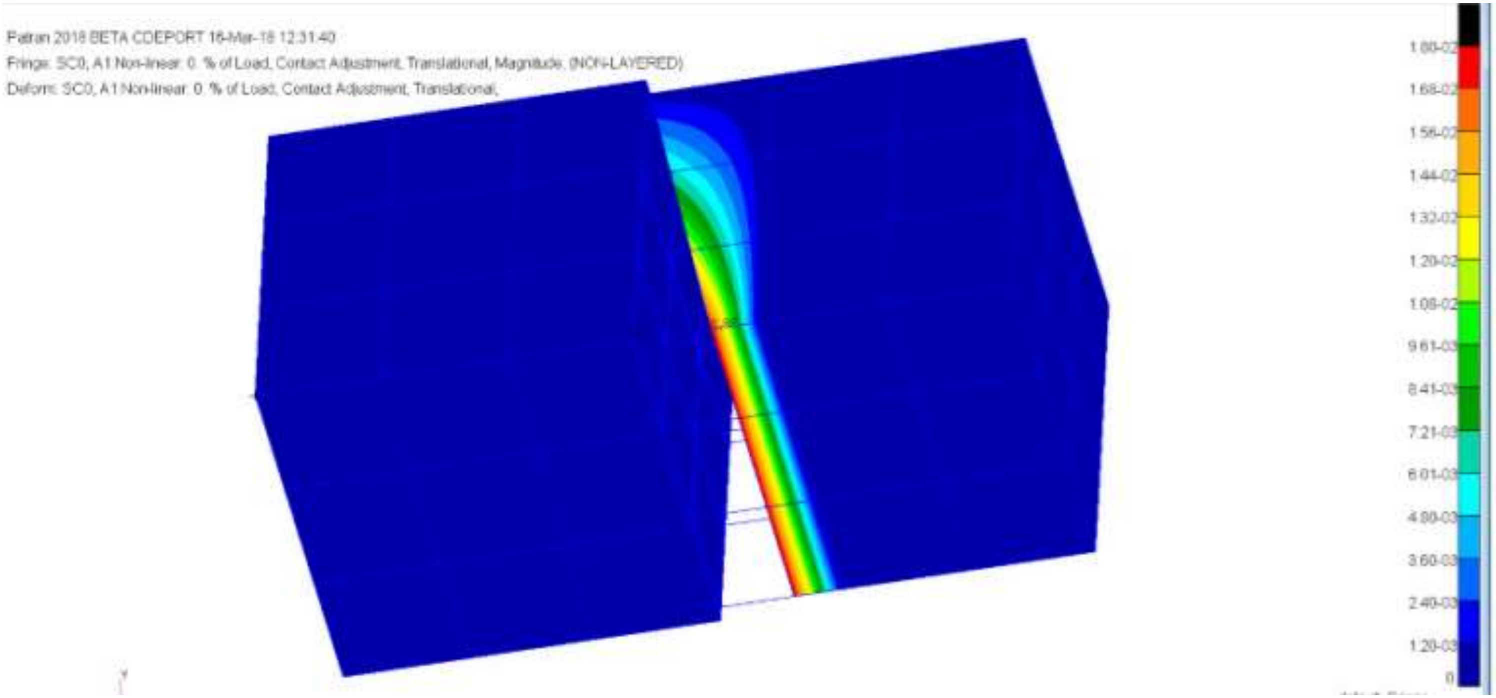
Contact Distance Check

- Contact Check is supported in segment-to-segment method as well



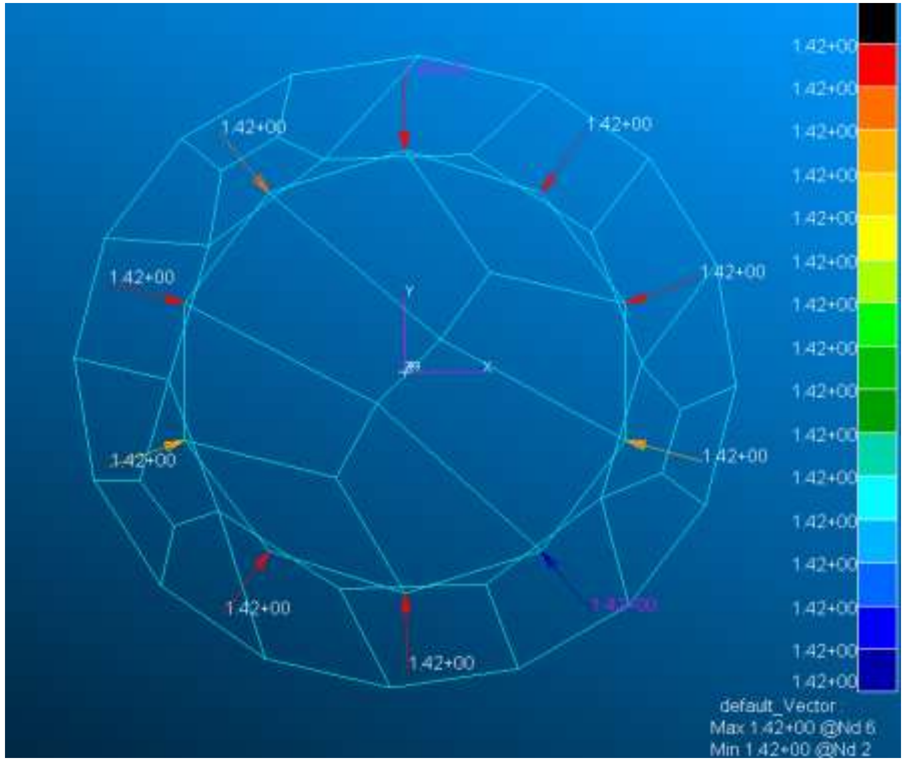
Geometry Adjustment Display of Initial Stress Free Contact

- Geometry of the grids are adjusted to achieve Initial Stress Free Contact
- Help users check the reasonability of the model setup
- Supports permanent glued contact, general glued contact, and touching contact

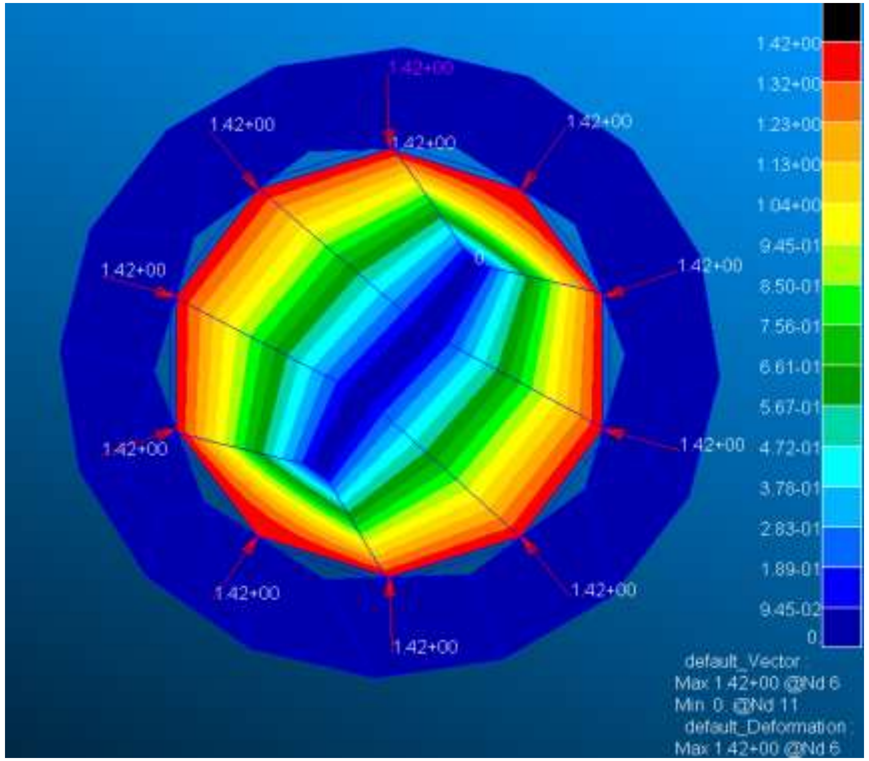


Initial Stress Free in SEGTOSEG Contact

- Seg-to-seg initial stress free contact can adjust geometry to clear gap/penetration



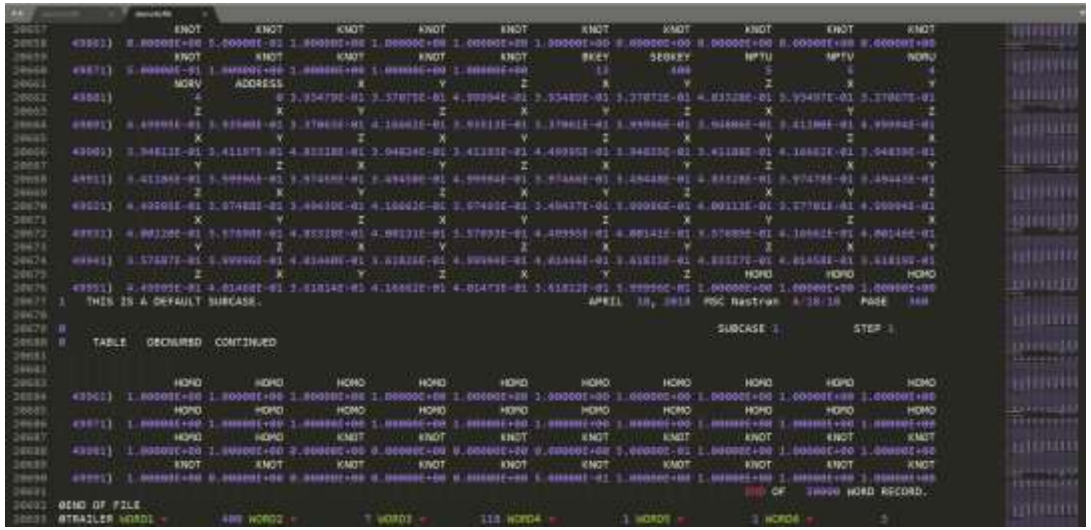
Contact Check Vector without Initial Stress Free



Initial Stress Free Adjustment Vector & Fringe

Output Analytical Spline Smooth Surface

- After FE discretization, continuity of model surface is lost.
- Analytical smooth surface may be generated for C1 continuity
- The smoothened surface is output to HDF5 and OP2 database.
  - It follows the input format of BCNURBS
- Supports all contact types for SOL 400



BIAS = 0.0 In BCONPRG and BCTABLE

- Allow user to apply the real ZERO as the input for BIAS in BCONPRG and BCTABLE and BCTABLE
  - Older versions had a limitation that required user to enter a near zero value

Table 8-5 Geometric Contact Parameters of Touching Bodies in SOLs 101, 103, 105, 107-112, 200 and 400

Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
AUGDIST	Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real ≥ 0.0, see Remark 6. for default)
BIAS	Contact tolerance bias factor. If this field is left blank or is equal to 0.0, the default is the BIAS of the BCPARA entry. A nonblank or non-zero entry will override the BIAS entered on the BCPARA entry. To obtain a near zero value, enter 1.0E-16. (0.0 ≤ Real ≤ 1.0)

Values used during contact : contact bias factor (BIAS)
Default Value = 9.0000000E-01

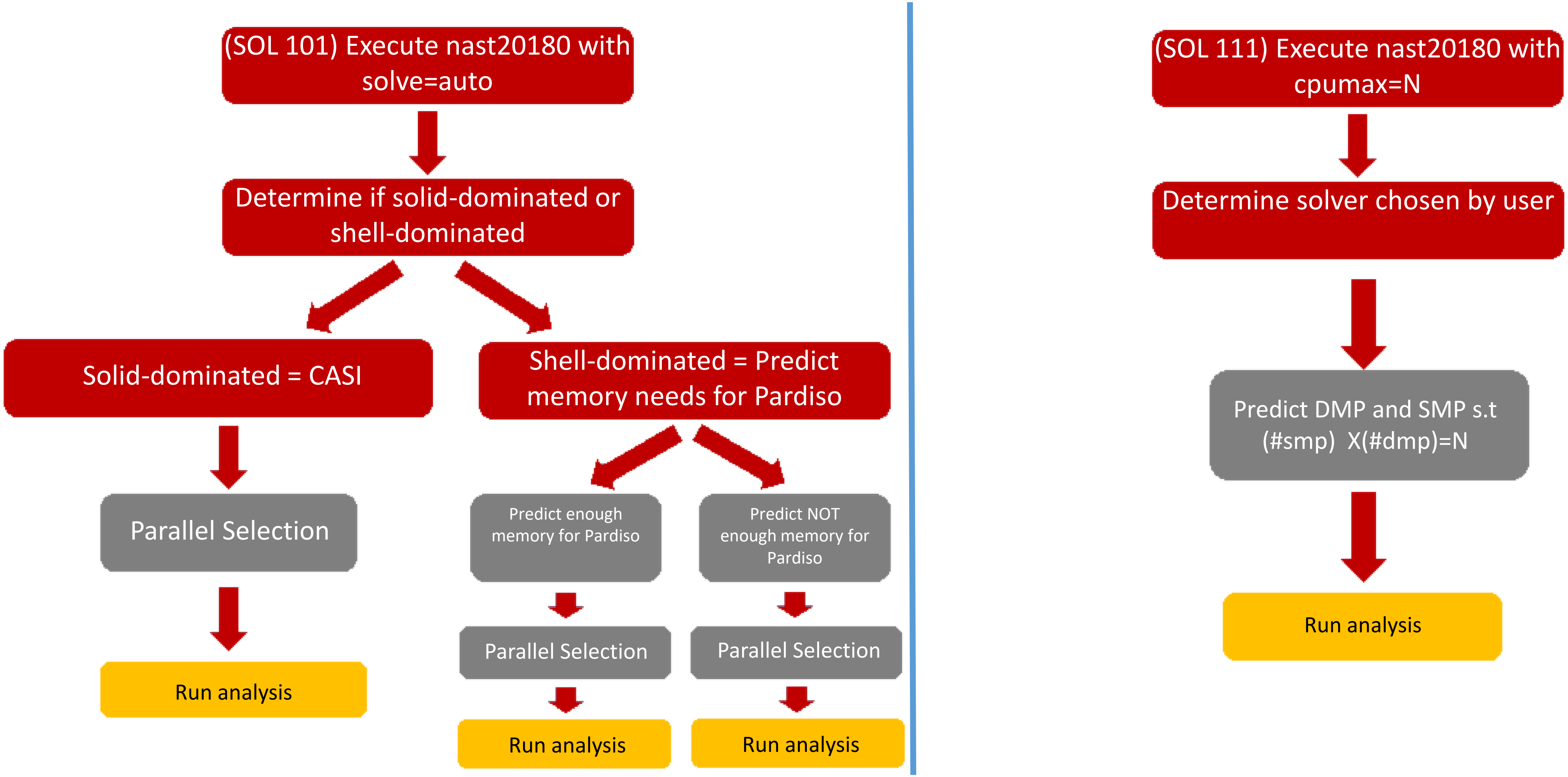
Body ID	1 Defor	2 Defor	3 Defor	4 Defor
1 Defor	N/A	9.0000000E-01	9.0000000E-01	9.0000000E-01
2 Defor	N/A	N/A	9.0000000E-01	N/A
3 Defor	N/A	N/A	N/A	N/A
4 Defor	N/A	N/A	9.0000000E-01	N/A

Values used during contact : contact bias factor (BIAS)
Default Value = 9.0000000E-01

Body ID	1 Defor	2 Defor	3 Defor	4 Defor
1 Defor	N/A	0.0000000E+00	0.0000000E+00	0.0000000E+00
2 Defor	N/A	N/A	0.0000000E+00	N/A
3 Defor	N/A	N/A	N/A	N/A
4 Defor	N/A	N/A	0.0000000E+00	N/A

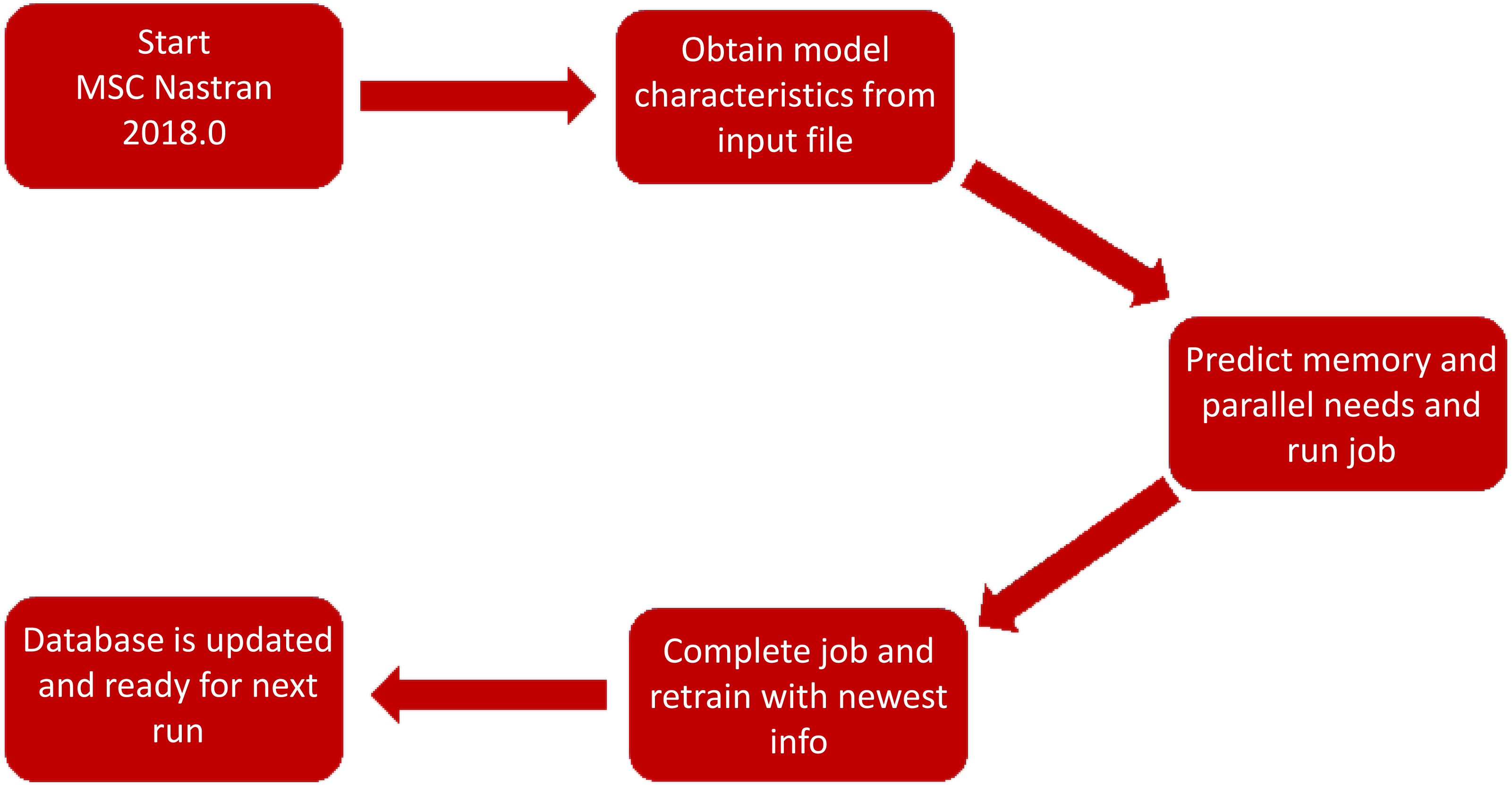
Numerical Computation and HPC Improvements - Automatic Solver and Parallel Selection Process

Automatic Solver and Parallel Selection Process



Automatic Solver and Parallel Selection Process

Machine Learning Capabilities



Automatic Solver and Parallel Selection Process

BIAS MÜHENDİSLİK

Machine Learning Capabilities

Run #1 with solve=auto

```
=== Solver Options ===
Type   :         Pardiso
Memory :        20300 MB
BPOOL  :        15201 MB

=== Parallel Options ===
DMP     :           1
SMP     :           4
```

```
    MEMORY PARAMETERS FOR INTEL MKL PARDISO PARALLEL DECOMPOSITION FOLLOW
                      AVAIL. CORE MEMORY =3250 MB
          APPROX. REQUI. IN-CORE MEMORY =3531 MB
    APPROX. REQUI. OUT-OF-CORE MEMORY =1804 MB
```

Wall Clock Time: **1796 s**

Run #2 with solve=auto

```
=== Solver Options ===
Type   :         Pardiso
Memory :        20972 MB
BPOOL  :        15201 MB

=== Parallel Options ===
DMP     :           1
SMP     :           4
```

```
  MEMORY PARAMETERS FOR INTEL MKL PARDISO PARALLEL DECOMPOSITION FOLLOW
                      AVAIL. CORE MEMORY =3384 MB
          APPROX. REQUI. IN-CORE MEMORY =3519 MB
  APPROX. REQUI. OUT-OF-CORE MEMORY =1803 MB
```

Wall Clock Time: **1750 s**

Run #3 with solve=auto

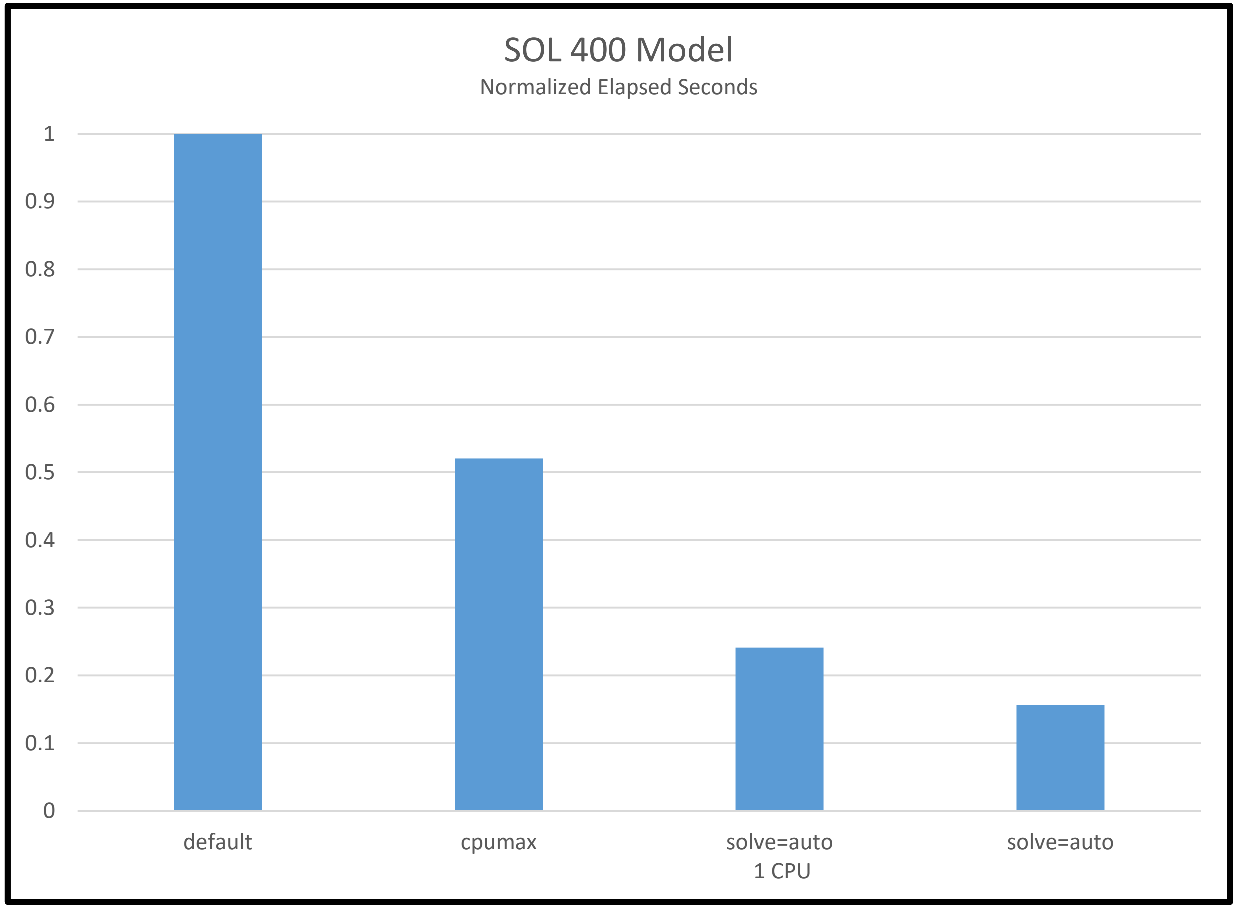
```
=== Solver Options ===
Type   :         Pardiso
Memory :        21744 MB
BPOOL  :        15201 MB

=== Parallel Options ===
DMP     :           1
SMP     :           4
```

```
  MEMORY PARAMETERS FOR INTEL MKL PARDISO PARALLEL DECOMPOSITION FOLLOW
                      AVAIL. CORE MEMORY =3539 MB
          APPROX. REQUI. IN-CORE MEMORY =3519 MB
  APPROX. REQUI. OUT-OF-CORE MEMORY =1803 MB
```

Wall Clock Time: **<u>1241 s</u>**

Pardiso runs in-core after training

Automatic Solver Selection Example



- Default – Serial run with default solver
- CPUMAX set to 20, which resulted in “smp=4”
- SOLVE=AUTO, w/ CPUMAX=1 resulted in the CASI solver being used. (This model is entirely made up of 3-D elements)
- SOLVE=AUTO w/o CPUMAX specified used SMP=8

Other Improvements - Multi-Mass Configuration

Multiple Mass Configurations

Typical aerospace vehicle analysis requires consideration of multiple mass cases involving different structural mass, payloads, fuel conditions and environmental conditions

- Enabled for constructing and analyzing additional mass cases for the standard linear, nonlinear and optimization solution sequences.
- Mass cases are subcase selectable similar to loads and boundary conditions.
- Mass Increments defined via additional BULK Data sections and can be combined with base mass

Solution Sequences Supporting MMC

- SOL 101: Static analysis with inertia relief and gravity load for user specified mass cases.
- SOL 103, 107, 110: Modes calculated for the user specified mass cases.
- SOL 108, 111: Frequency Response calculated for user specified mass cases.
- SOL 109, 112: Transient Response calculated for user specified mass cases.
- SOL144-146: Aeroelastic analysis for user specified mass cases.
- SOL 400: Supported for linear, nonlinear and perturbation analysis.
- Part SE support in all the above solution sequences.
- SOL 200: Optimization will be supported with invariant mass increments.

Other Improvements – Shell Stress Constraints for Topology and Topometry Optimization

Stress Constraints for Shell Topology Optimization

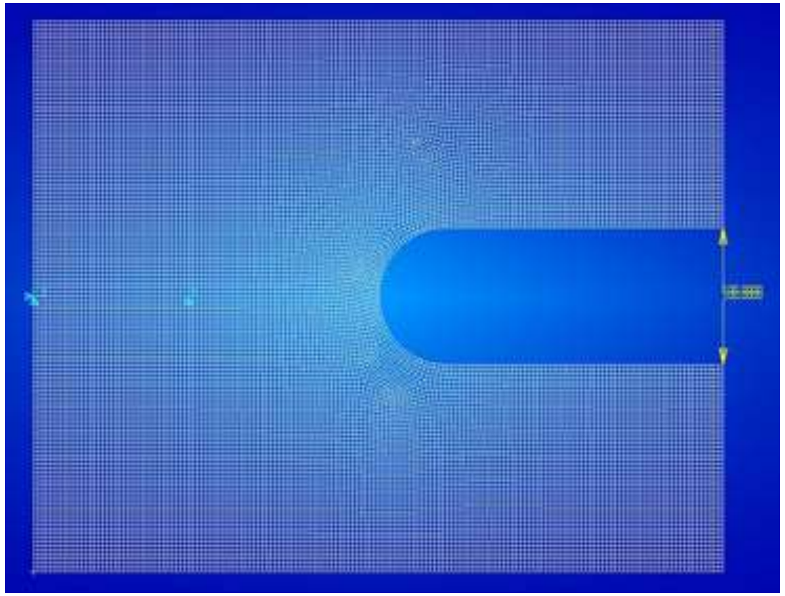
Benefits

- Allow users to have Von Mises stress (at element center) constraints in a topology design optimization task
- Eliminate the guessing of the right fractional mass constraint in classical compliance minimization design tasks

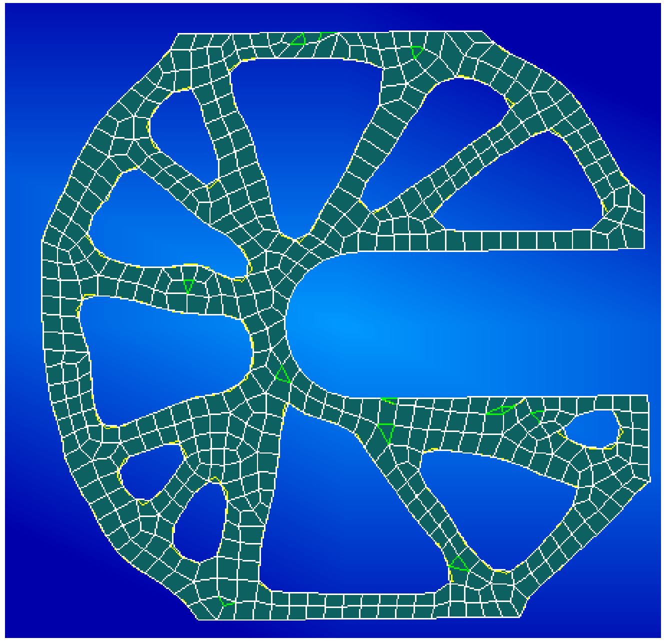
Input

1	2	3	4	5	6	7	8	9	10
TOPVAR	ID	LABEL	PTYPE	XINIT	XLB	DELXV	POWER	PID	
	"SYM"	CID	MS1	MS2	MS3	CS	NCS		
	"CAST"	CID	DD	DIE	ALIGN				
	"EXT"	CID	ED	ALIGN					
	"TDMIN"	TVMIN	TVMAX						
	"STRESS"	STLIM							

Stress Constraints for Shell Topology Optimization



Clip example: The dimensions of the clip are 100 horizontally and 80 vertically.



Minimize FRMASS

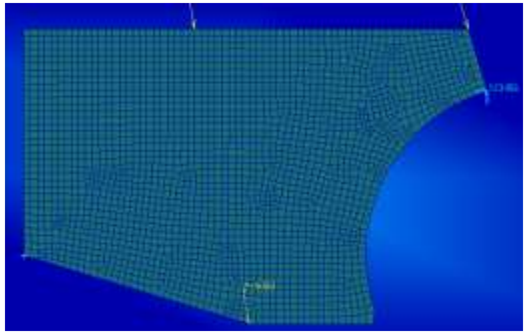
s.t. stress <=100.0MPA

Stress Constraints for Shell Topometry Optimization

BiAS MÜHENDİSLİK

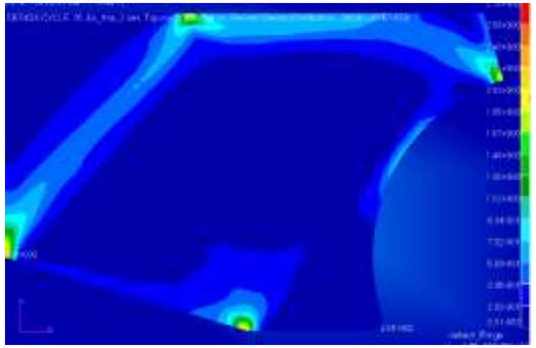
Benefits and limitations

- Allow users to have Von Mises stress (at element center) constraints in a topometry design optimization task
- Support element shell thickness only



Input

TOMVAR	ID	PTYPE	PID	PNAME/ FID	XINIT	XLB	XUB	DELXV	
	"DLINK"	TID	C0	C1					
	"DDVAL"	DSVID							
	"STRESS"	STLIM							



Other Improvements – HDF5 Results Database

HDF5 Result Database (NH5RDB)

- **Data type support**
  - Fatigue vibration data block OEFTGV
  - Solution set output SDISP, SVELO, ACCE and UHT vector
  - Monitor point data in SOL200 and AESTAT
  - Design optimization topometry and topology data TOMVAR and TOPVAR
  - Design optimization response R1TABRG, HISADD and RESP12
  - Wet faces data in acoustic analysis
  - Metadata entries
- **Matrix data**
  - Modal matrix KHH, MHH and BHH
  - General matrix data output

HDF5 Result Database

- **MSC Nastran file management**
  - Support ASSIGN statement for NH5RDB file
    - assign HDF5='myfile.h5'
  - Versioning
    - Append version number in NH5RDB file for multiple MSC Nastran runs
- **MDLPRM parameters**
  - H5MDL: write model data in separate file
  - H5MTX: write matrix data in separate file
  - H5GM34: optionally write GEOM3 and GEOM4 data block in NH5RDB
- **Patran support**
  - Monitor point, design response and contact force

What's New
MARC 2018 & MARC 2019

Bias Mühendislik
Yiğit ÇELİK
03/10/2019

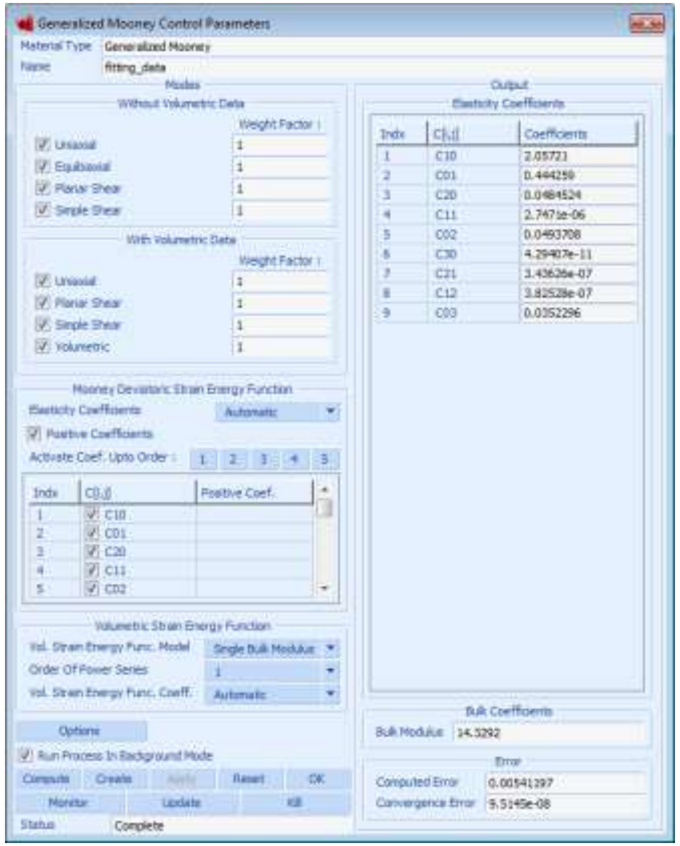
Outline

- Materials
  - I. Expanded Material Data Fitting Capabilities in Mentat
  - II. Support for Additive Plasticity in Hermann Elements
  - III. Prediction of Heat Generated in Viscoelastic Materials Subjected to Repeated Loading
- Contact Analysis
  - I. Automatic Contact Detection in Mentat for Efficient Creation of a Large Number of Deformable Contacts
  - II. Improved Contact Fidelity Between a Rigid Surface and Deformable Body
- Solver – Output Enhancements
  - I. Localized Global Remeshing Capability
  - II. Improved Accuracy and Performance via Localized Options for Convergence Checks
  - III. HDF5 Output
- Additional Enhancements
  - I. More Efficient Process of Defining Multiple Springs and Fasteners with Preload
  - II. Extended Support for Import of Nastran Models with Contact Cards
  - III. General Mentat Usability Improvements
  - IV. Endurica Compatibility with Marc for Prediction of Rubber Fatigue

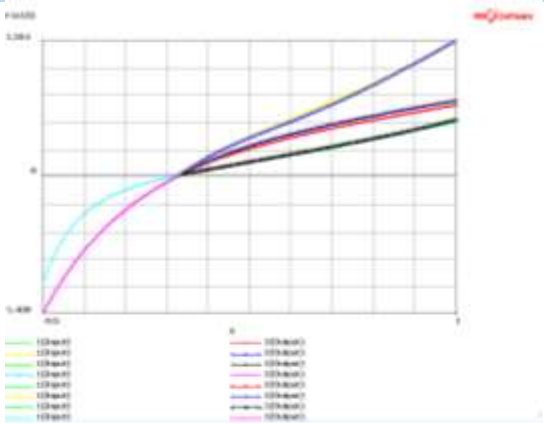
Materials

Improved User Efficiency via Expanded Material Data Fitting Capability in Mentat

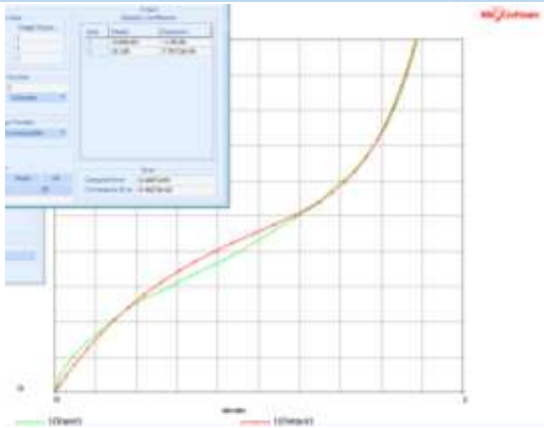
- **Modeling Application:** Improve quality and robustness of material definitions defined by complicated experimental data via an automated process
- **Example Engineering Application:** Simplify use of the Ogden-Roxburgh model by providing an automatic high quality fit to experimental data
- **Implementation:**
  - Accessed via the existing material data fit interface
  - Addition of the following material definitions
    - Series expansion for the volumetric strain energy function,
    - Full support of time independent isotropic hyperelastic models
    - Ogden-Roxburgh damage model



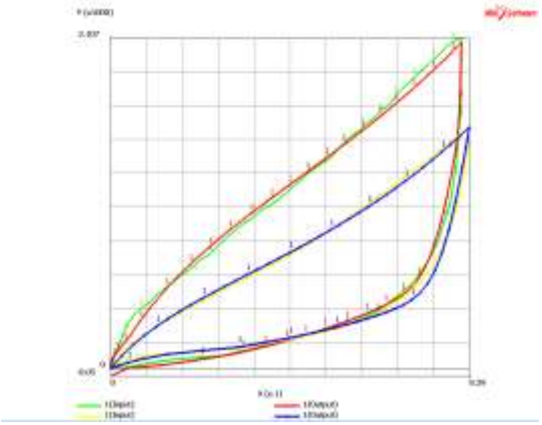
Example: Curve Fits



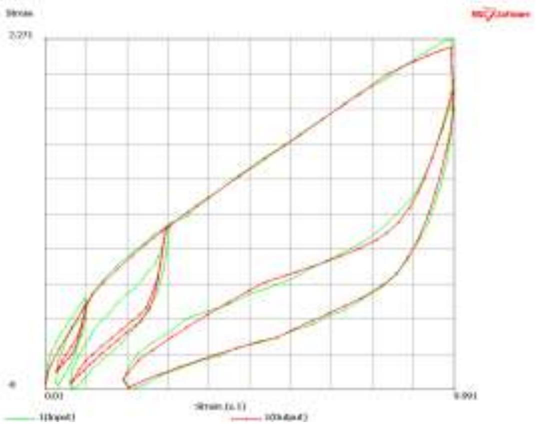
Time Independent Hyper Elastic Materials Data Fitting with Differential Evolution



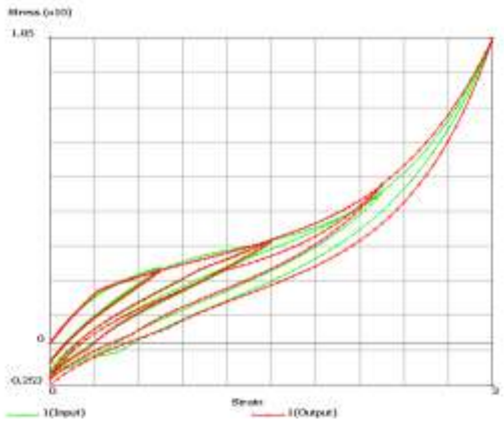
Ogden Data Fit without Volumetric Data



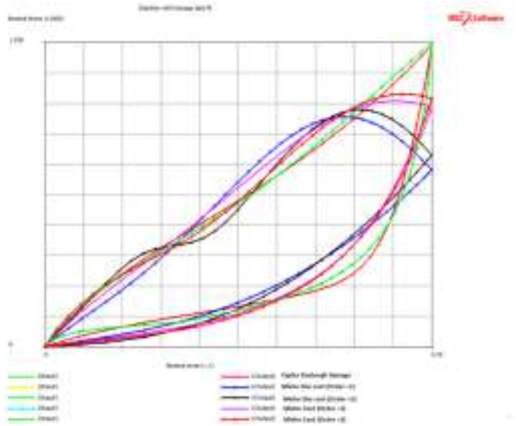
Data Fitting with Elasticity O-R Damage Viscoelasticity and Plasticity



Elasticity with Damage And Viscoelasticity and Plasticity



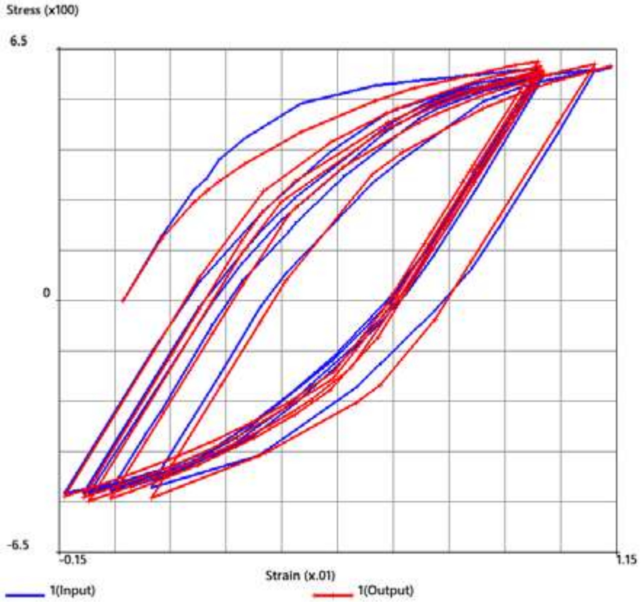
Elasticity with Damage and Plasticity

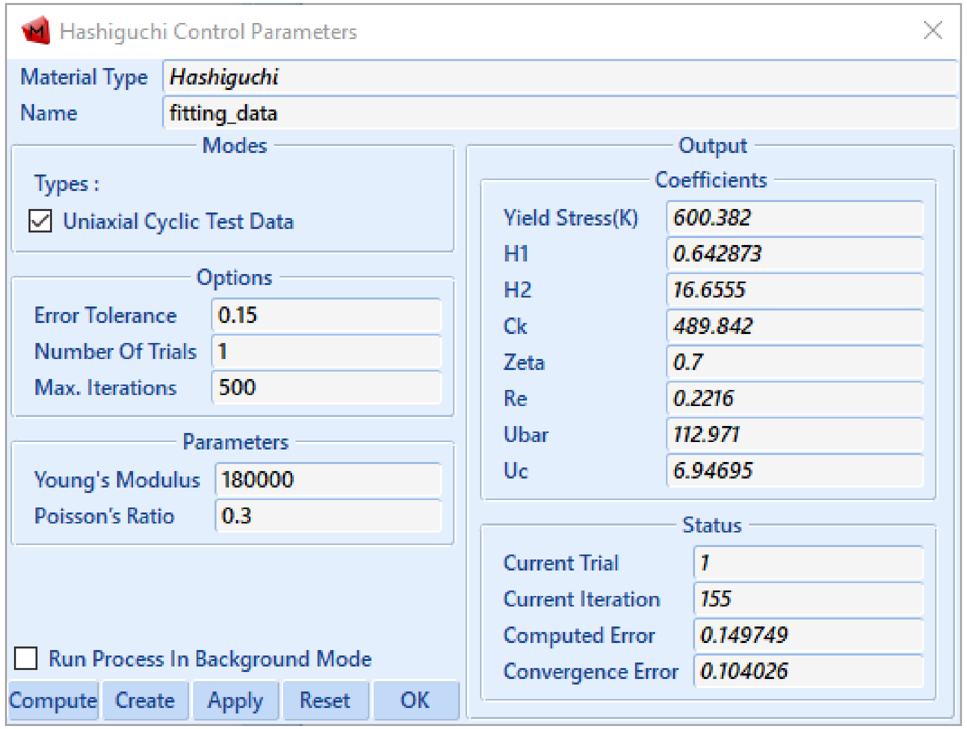


O-R Damage Comparison with Miehe Damage

Additional Material Support in Experimental Data Fit

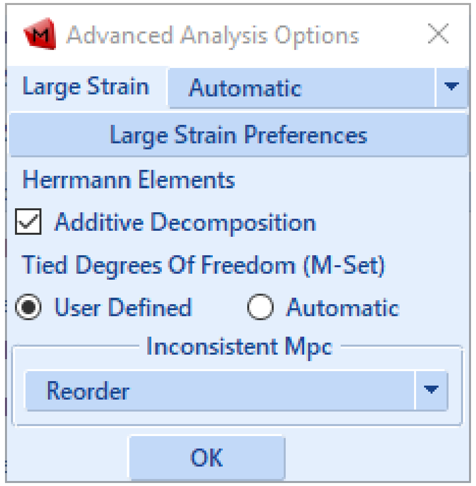
- **Modeling Functionality:**
  - ➢ Extend experimental data fitting to include cyclic plasticity to increase ease of use and reduce potential for error
- **Example Engineering Application:**
  - ➢ Metal subjected to repeated loading and unloading in the plasticity region such as may occur in pipes during extreme events
- **Implementation:**
  - ➢ Select Plasticity in the Experimental Data Fit Window and then se





Support for Additive Plasticity in Herrmann Elements

- **Modeling Functionality:**
  - ➢ Expand Herrmann elements (for incompressible materials) to include an additive plasticity formulation that may improve accuracy and/or reduce runtime over strain smoothing elements
- **Example Engineering Application:**
  - ➢ Manufacturing processes such as forging, upsetting, extension or deep drawing, and/or large deformation of structures that occur during plastic collapse
- **Implementation:**
  - ➢ Assign a Herrmann element
  - ➢ Large strain preferences in the element option
    - "Additive Decomposition" option will be turned on by default (can turn off manually)

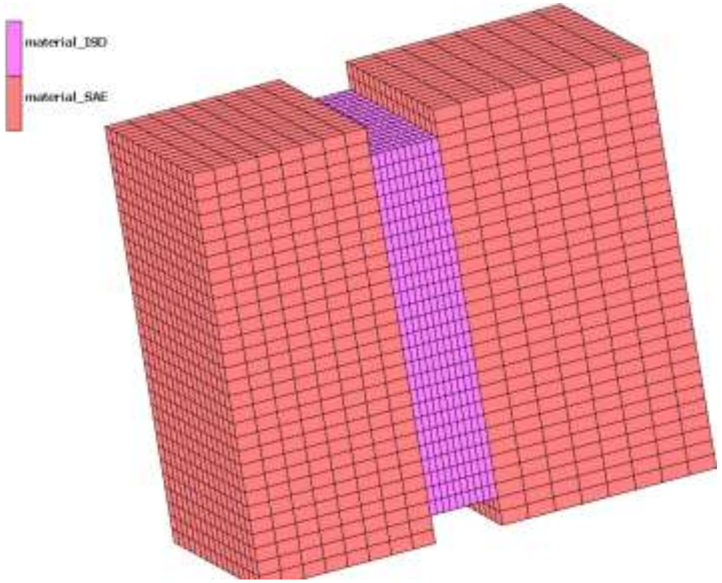


Predict Heat Generated in Viscoelastic Materials Subjected to Repeated Loading

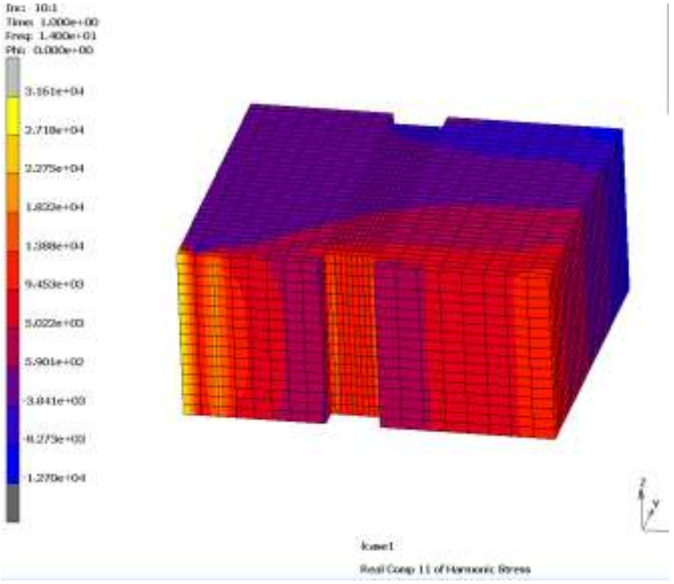
BIAS MÜHENDİSLİK

- **Modeling Application:** Calculate heat induced by repeated loading of rubber and rubberlike material
- **Example Engineering Application:** Improve prediction of rubber component performance in automotive
- **Implementation:**
  - Coupled Sequential analysis
    - Harmonic analysis (linear or nonlinear) with one frequency per load case (multiple frequencies- multiple load cases)
      - No temperature dependence
    - Coupled Sequentially with a Steady state or Transient thermal analysis using the harmonic data to drive heat generation
      - Temperature dependent properties supported
    - Relevant results could be thermal flux or equilibrium temperatures depending on solution chosen

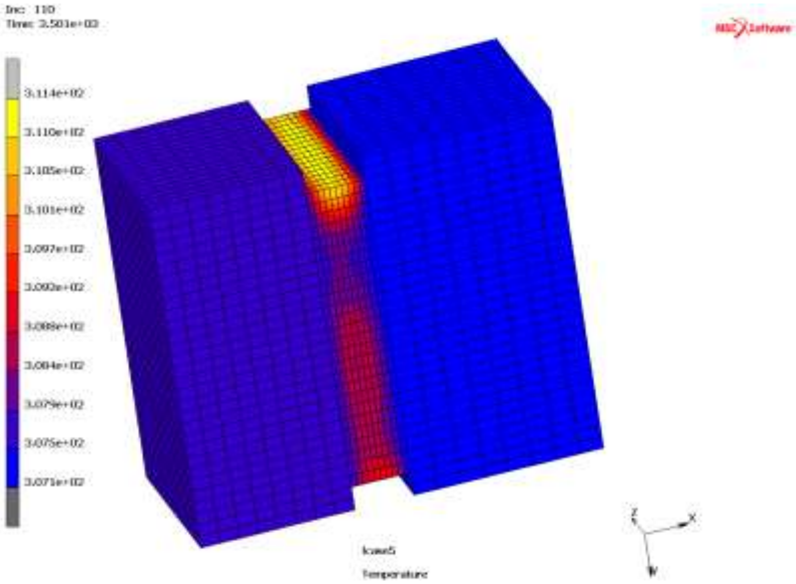
Example: Self Heating of a Viscoelastic Damper



Damper Model



Harmonic Stresses



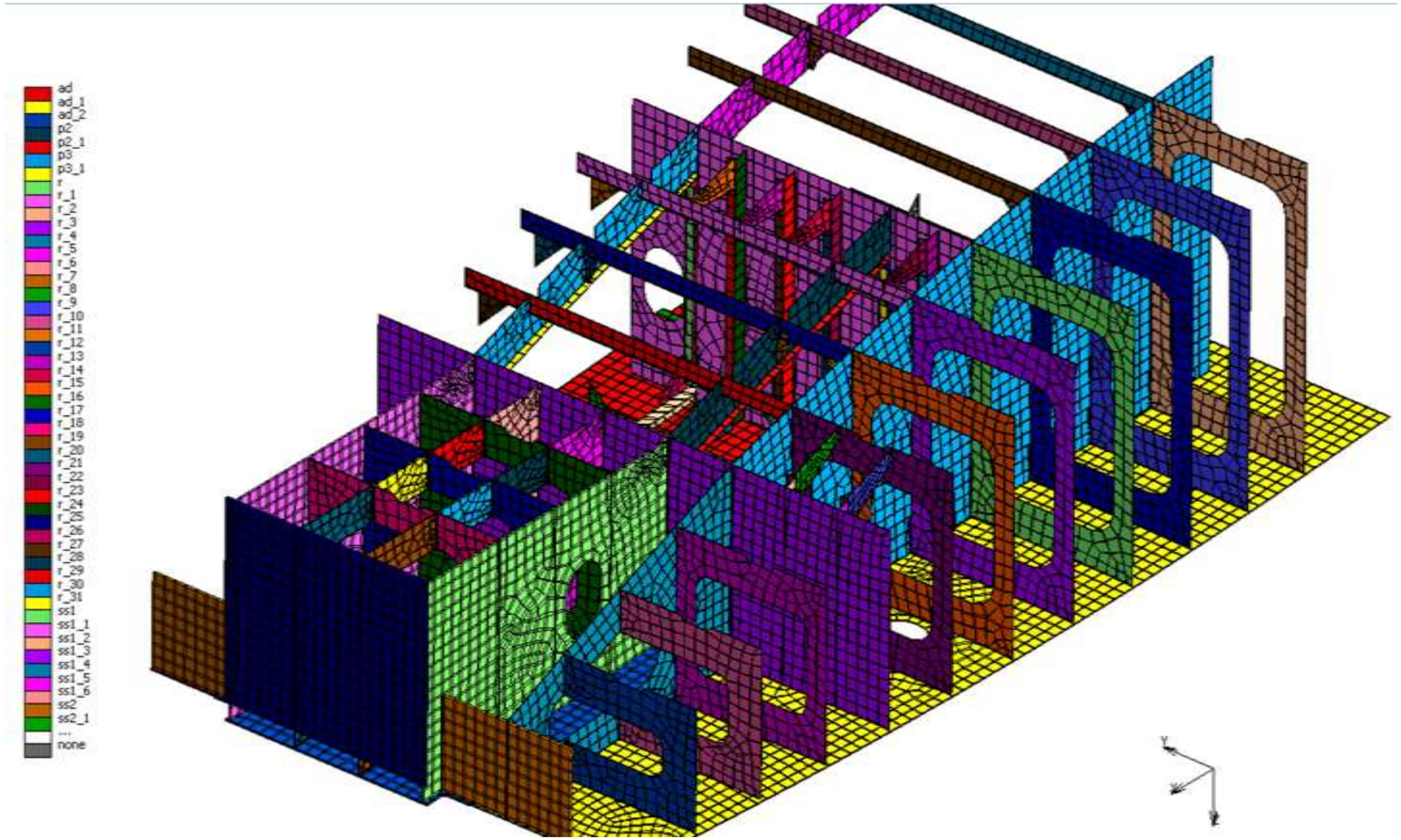
Temperature

Contact Analysis

Automatic Contact Detection in Mentat for Efficient Creation of A Large Number of Deformable Contact

- **Modeling Application:** Improve user efficiency and reduce error by automatically detecting contact pairs and populating the contact table
- **Example Engineering Application:** Large assemblies and complicated models with multiple instances of deformable-deformable contact
- **Implementation:**
  - User creates contact bodies
  - User specifies contact type and maximum contact search distance for the detection algorithm
  - User specifies whether detected contact body pairs are added and/or pre-existing entries are replaced in the contact table.
    - Option to remove pre-existing entries in the contact table if the detected distance is above a limit.
    - Option to exclude self contact
  - User reviews the newly generated or modified contact table entries and can edit as with manual entries

Example: Large Assemblies with Multiple Contacts



Use case : Barge Part
Application : Transportation

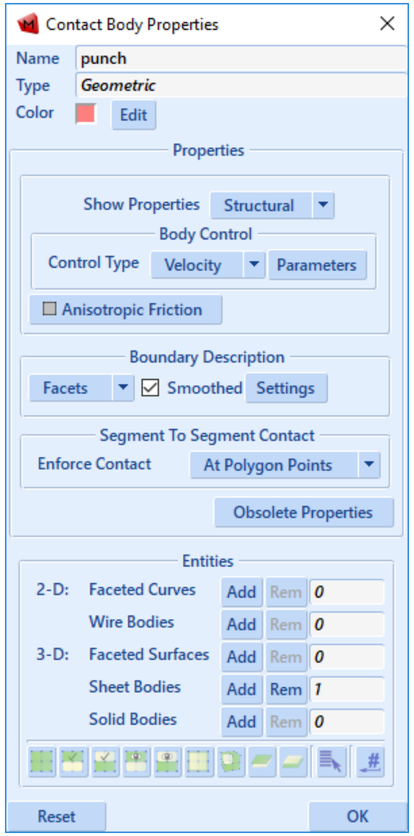
Use Case Details :

Total Contact Bodies : 139
Total Contact Pairs : 275

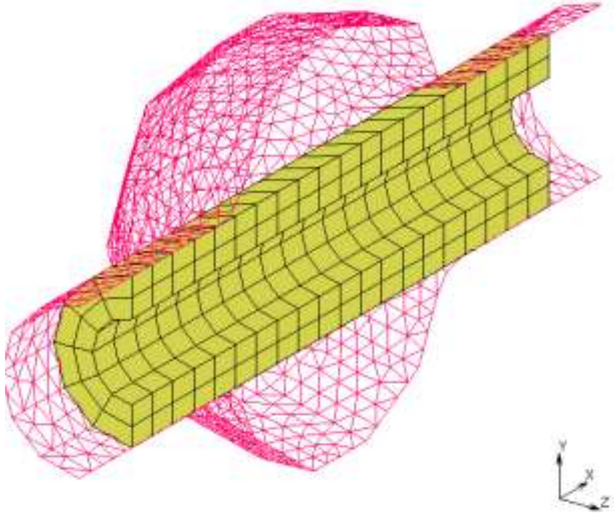
Improved Contact Fidelity Between a Rigid Surface and Deformable Body

- **Modeling Functionality:**
  - Expand contact options between rigid STL surfaces and deformable bodies to include segment to segment for improved accuracy
- **Example Engineering Application:**
  - Die Forming Simulations
- **Implementation:**
  - STL surface can be used with or without the faceted surface option
  - Additional contact points are defined by the solver on the rigid body to limit penetration
  - Augmentation and associated parameters previously limited to deformable to deformable contact are now supported in geometric to deformable contact
  - Also support delayed slide off between deformable and faceted surfaces

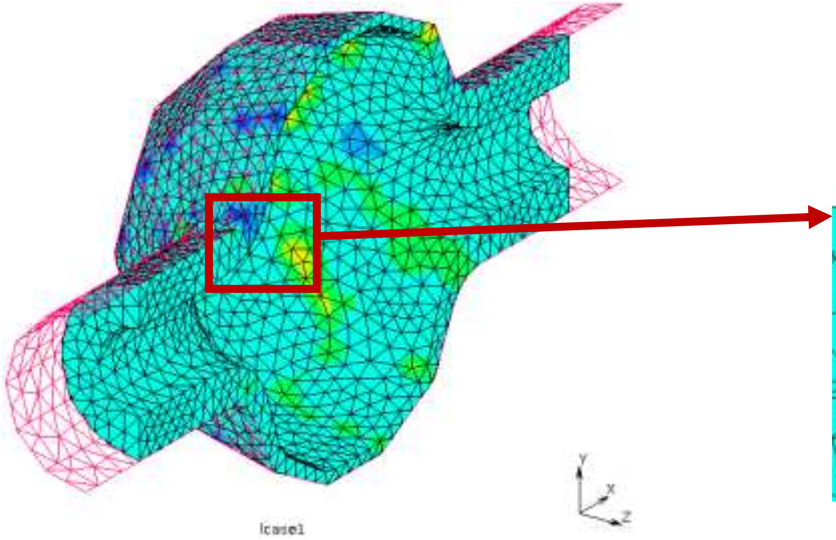
Hydroformed Parts



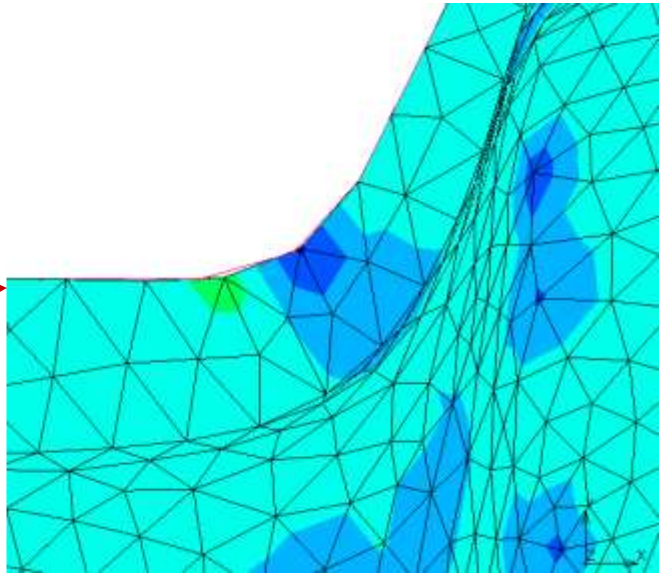
Example: Hydroforming



Model



Results



Close Up Results

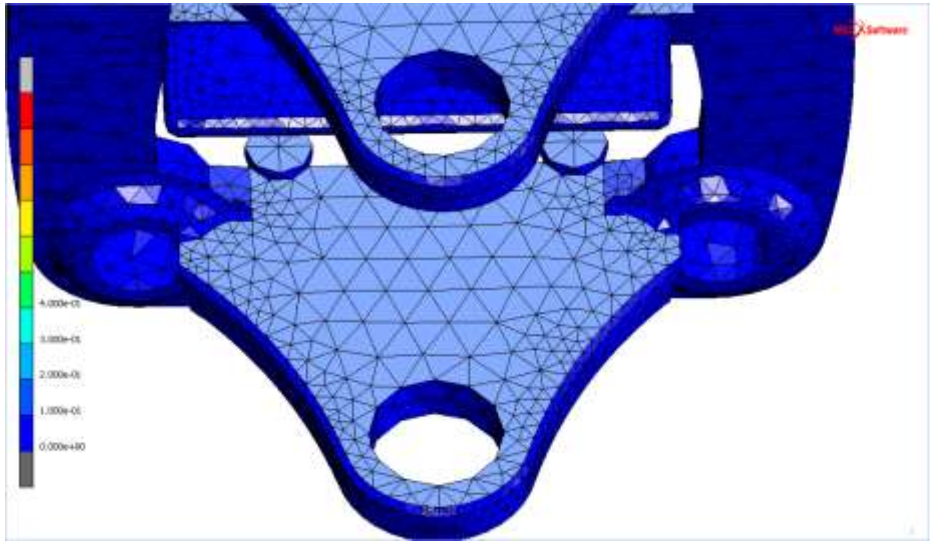
Solver – Output Enhancements

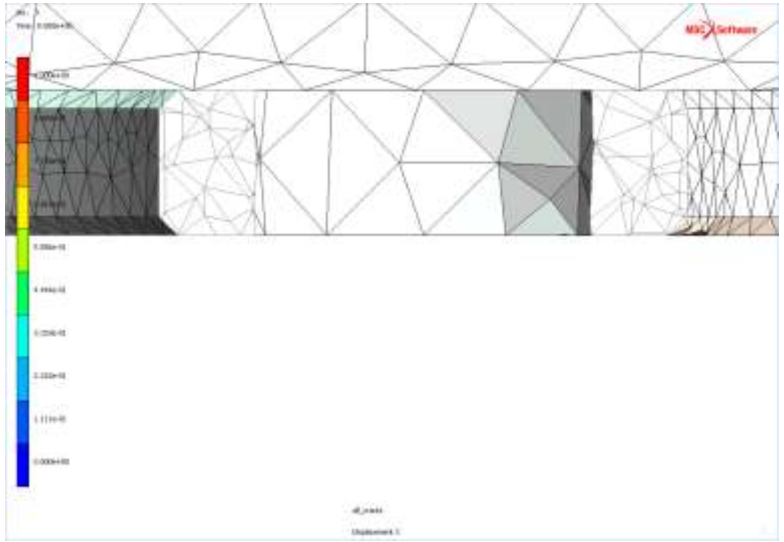
Localized Global Remeshing Capability

- **Modeling Functionality:**
  - Add option to limit global remeshing to a specified region in a contact body to reduce runtime and maintain consistent mesh in remote areas compared to remeshing the entire contact body.
- **Example Engineering Application:**
  - Crack propagation
- **Implementation:**
  - New option in Global Remeshing Properties Menu for "Remesh Region". Initial region can be specified by
    - Set of Elements
    - Crack Vicinity
    - Prescribed Geometric Shape (box, sphere, etc.)
  - Compatible with existing advanced density controls
    - Region, Distance, Element Quantity, etc.

Example: Localized remeshing around a crack tips

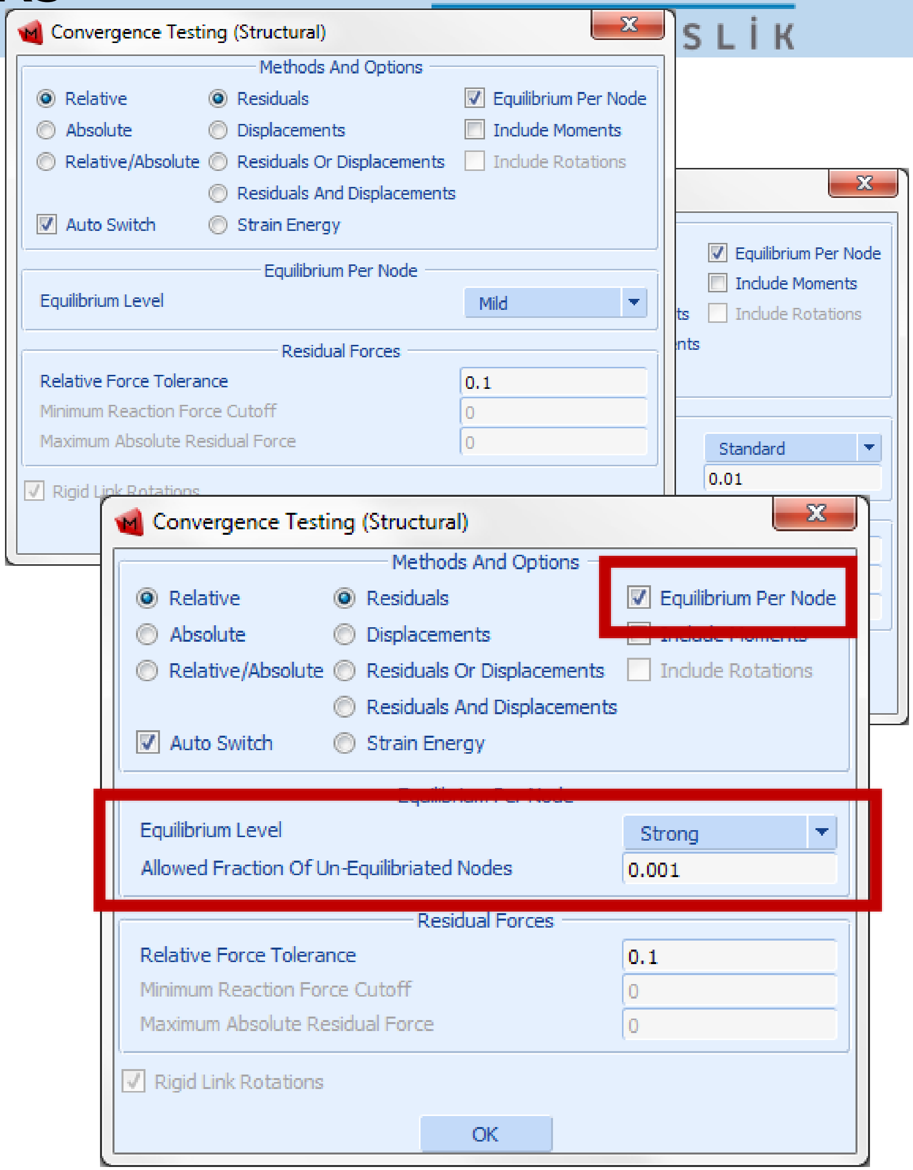
Sr. No.	Parameters	Old Remeshing Method	New Remeshing Method	% reduction
1	Total Memory (in Mbyte)	556	317	43
2	Total Time	256	68	73.5



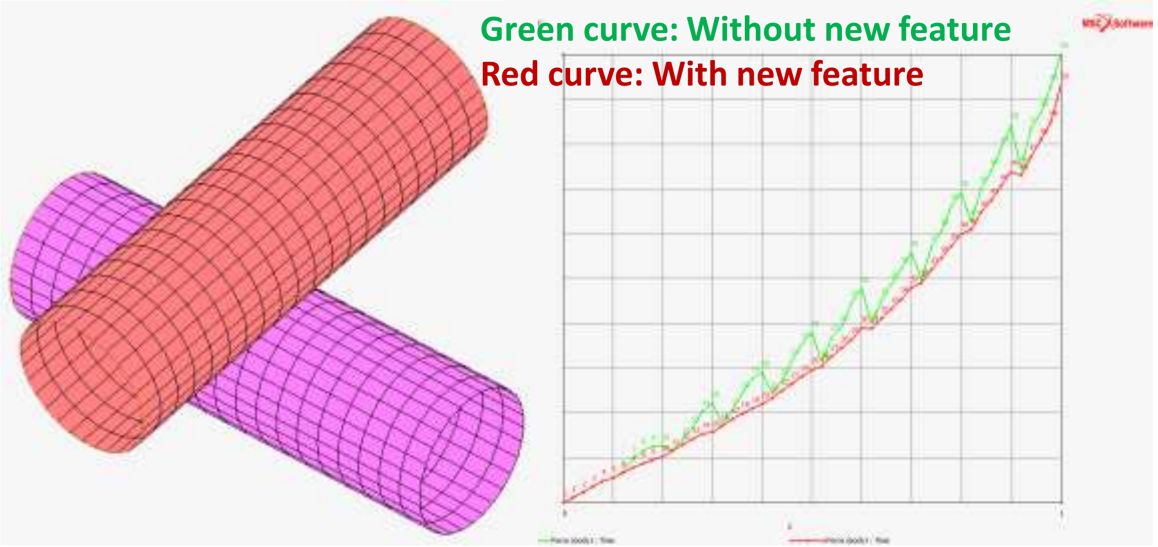


Improved Accuracy and Performance via Localized Options for Convergence Checks

- **Modeling Application:** Perform convergence checks using force residuals and maximums locally at individual nodes
- **Example Engineering Application:** Improve the efficiency and accuracy at a local level for a wide range of simulations
- **Implementation:**
  - User selects “Equilibrium per node” option
  - User prescribes desired level of accuracy
    - For “Mild”, a single check is performed and one extra iteration is forced if not satisfied
    - For “Standard” level the default is 0.005 fraction of unequilibrated nodes allowed
    - For “Strong” level the default is 0.001 fraction of un-equilibrated nodes allowed
    - For “Standard” and “Strong” levels the user can manually input an allowed fraction of un-equilibrated nodes
  - User may request post codes related to the local convergence process:
    - Residual Force
    - Residual Moment
    - Nodal Force Convergence Ratio
    - Nodal Force Convergence Status

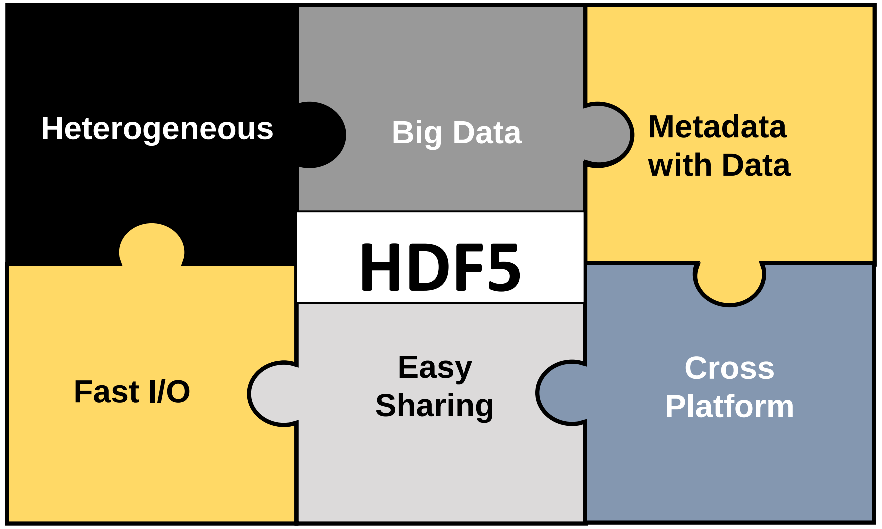


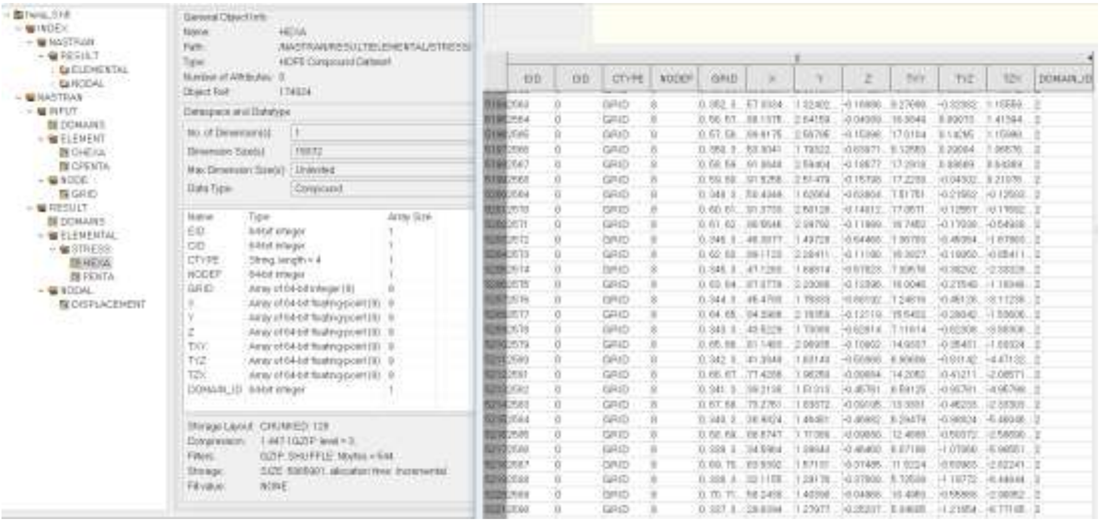
Example: Improved Results Accuracy



HDF5 Output

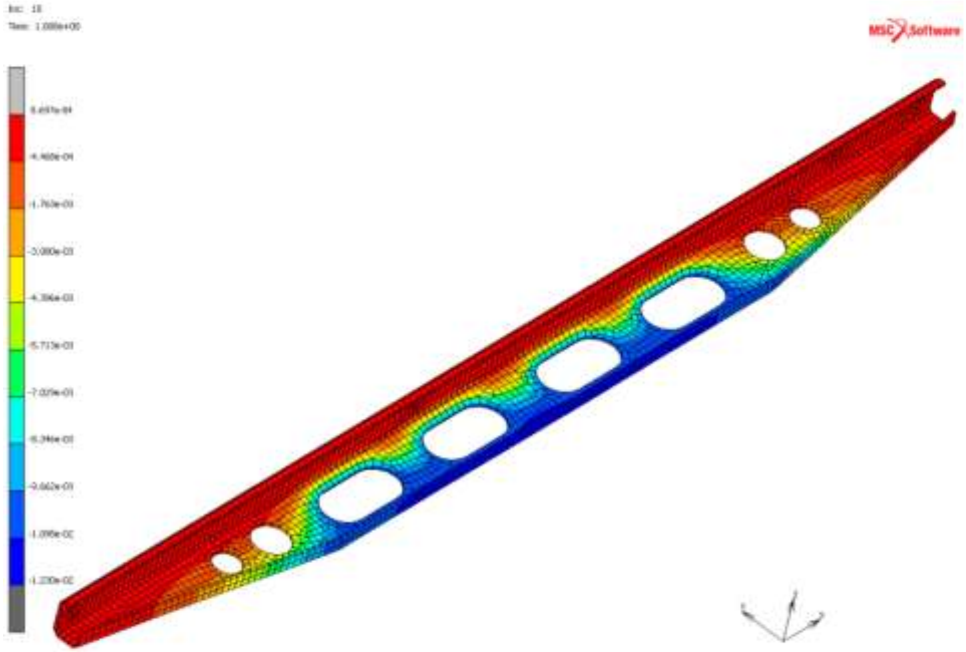
- **Modeling Functionality:**
  - ➢ Alternate open format results file generated by Marc
- **Example Engineering Application:**
  - ➢ Improve integration into downstream engineering processes by accessing output data directly without a post-processer
- **Implementation:**
  - ➢ HDF5 output file
  - ➢ Schema compatible with MSC Nastran HDF5
  - ➢ List of Supported Elements in this release
    - Linear and Quadratic 2D plane stress, plane strain and axisymmetric.
    - Linear and Quadratic 3D continuum.
  - ➢ Output Supported in This Release
    - Nodal: Displacements, External Forces, Reaction Forces
    - Element: Stress, Strain



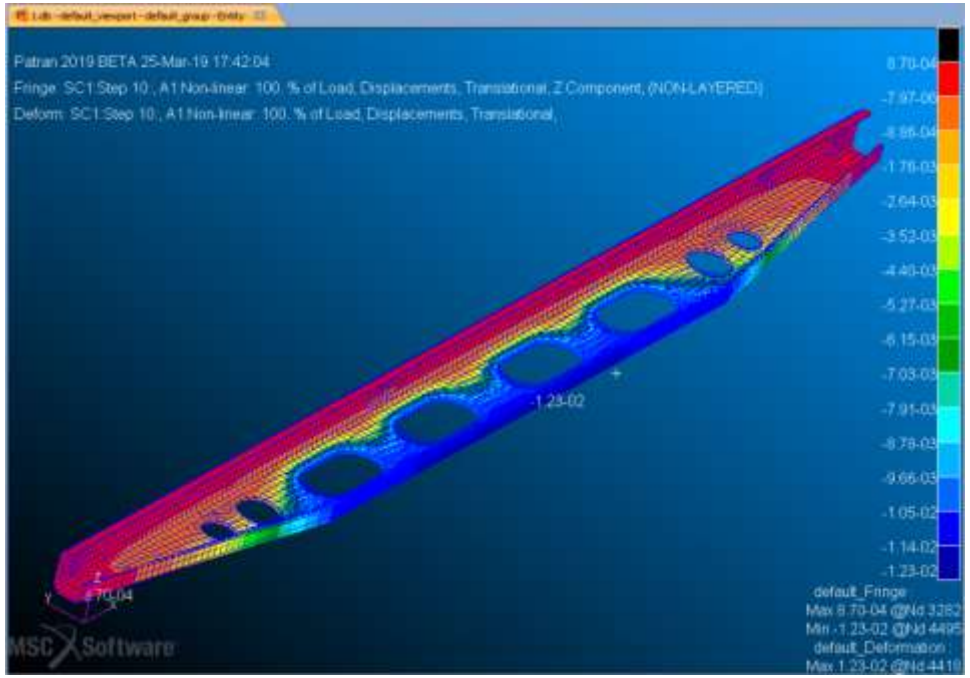


Example: Open H5 Results in Patran

BIAS MÜHENDİSLİK



.t16 results in Mentat



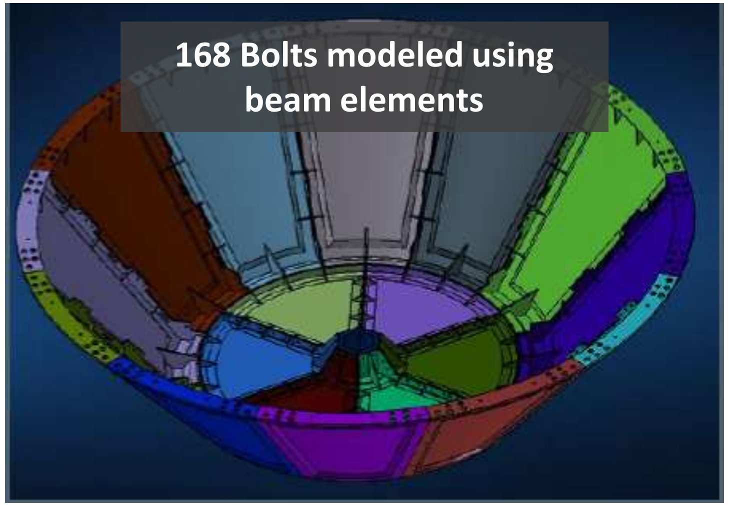
.H5 results in Patran

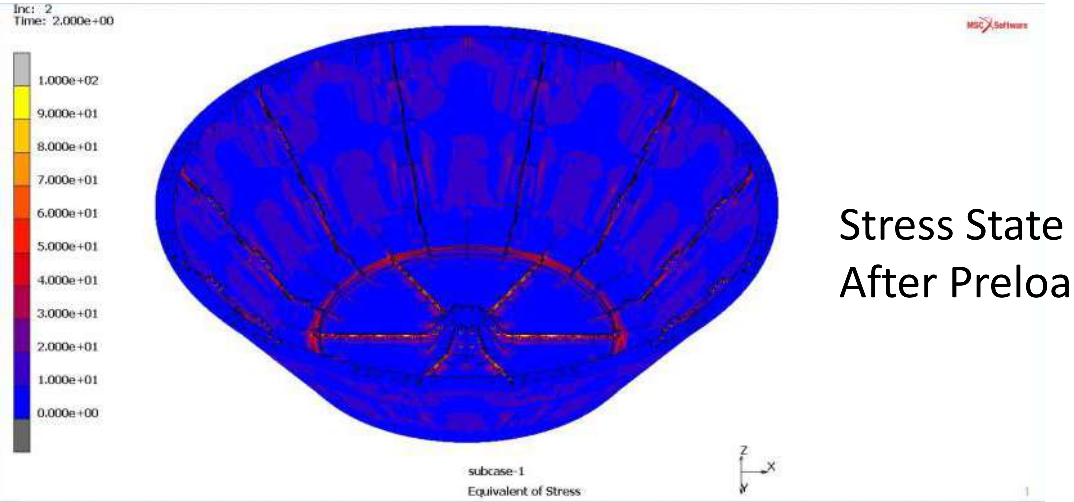
Additional Enhancements

More Efficient Process of Defining Multiple Springs and Fasteners with Preload

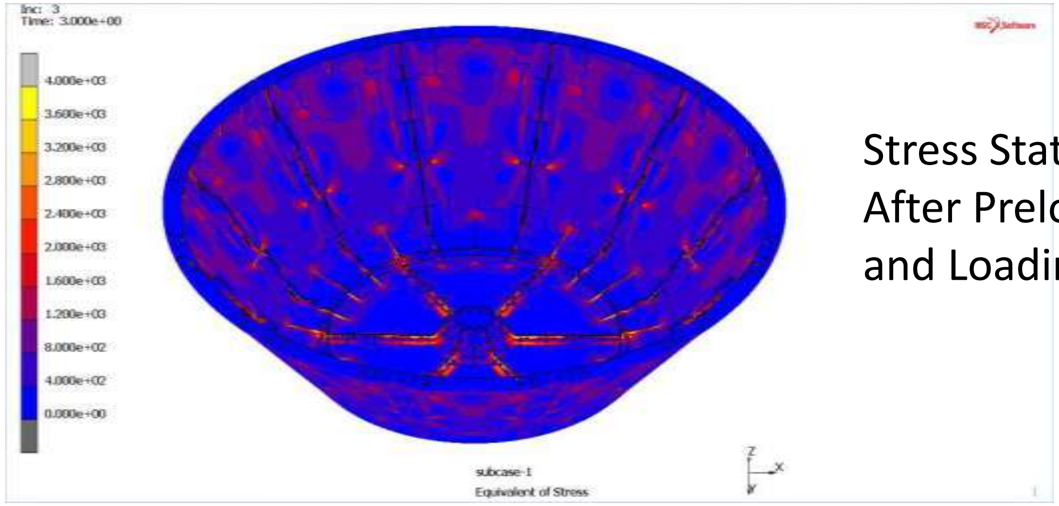
- **Modeling Application:** Efficient and user friendly method for defining common behavior to multiple springs and preloaded fasteners
- **Example Engineering Application:** Assembly with a large number of fasteners with preload or springs
- **Implementation:**
  - Support for large rotation of the Cross-Section option
  - Streamline process for the application of the Cross-Section option to Bolts (new data base entry):
    - Select elements of the bolt
    - Mentat will automatically create and position the control node
    - Apply a boundary condition for preload by selecting the control node or, by a special filter, elements of the bolt
    - The axial direction of the bolt is automatically determined and updated by Marc
    - During post processing, the bolt force is plotted in the current direction
    - Special global variables for control node displacement as well as axial and shear force
    - Multiple bolts can quickly be generated using a single action
  - Springs are now handled as elements:
    - Easily apply the same stiffness and damper properties to multiple springs
    - Ability to activate and deactivate springs per load case

Example: Satellite Dome Assembly





Stress State
After Preload

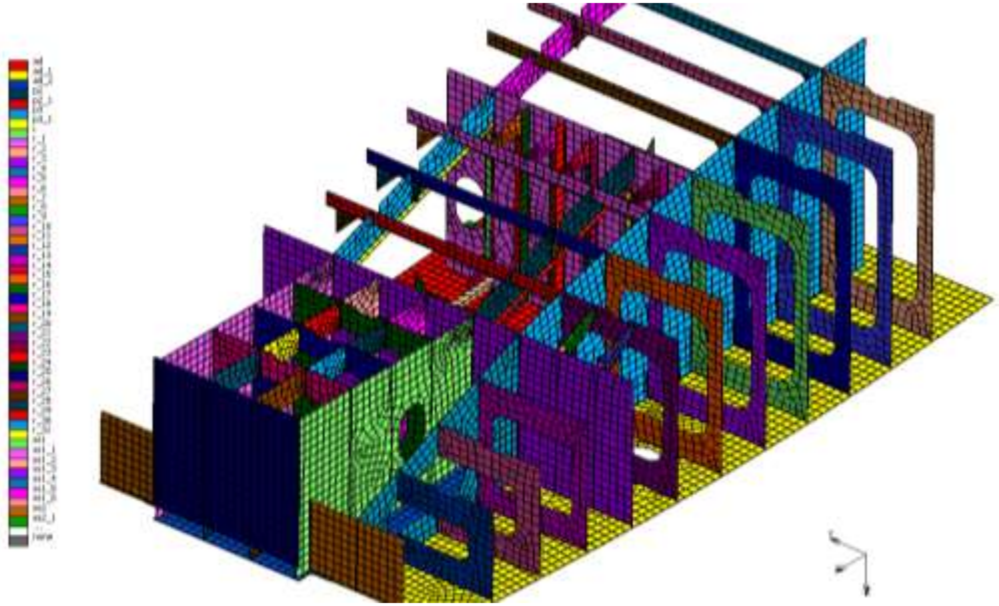


Stress State
After Preload
and Loading

Extended Support for Import of Nastran Models with Contact Cards

BIAS MÜHENDİSLİK

- **Modeling Application:** Share common model data across different solvers
- **Example Engineering Application:** Transition from a Nastran Linear run to a Marc Nonlinear Run
- **Implementation:**
    - Extended Support for Import of Nastran Models with Contact Cards



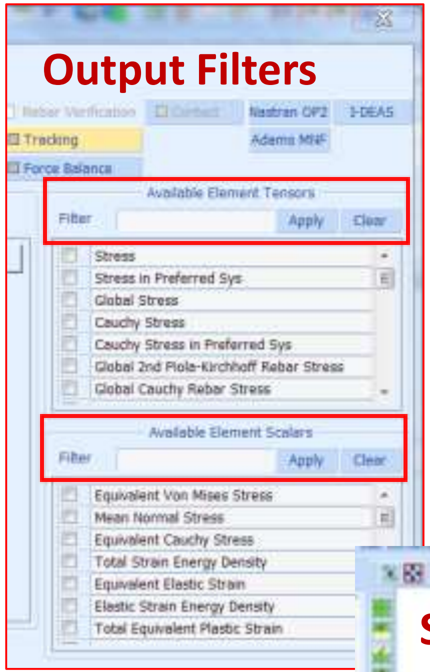
Nastran Cards Tested:
BCPARA, BCONPRG, BCONPRP, BCONNECT, and BCTABLE1

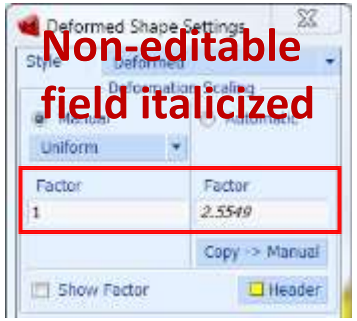
- Correct Translations of
    - Contact Body
    - Contact Table
    - Touching and Glued Settings
    - Breaking and Separating Glue
    - Distance Tolerance
    - Thermal contact

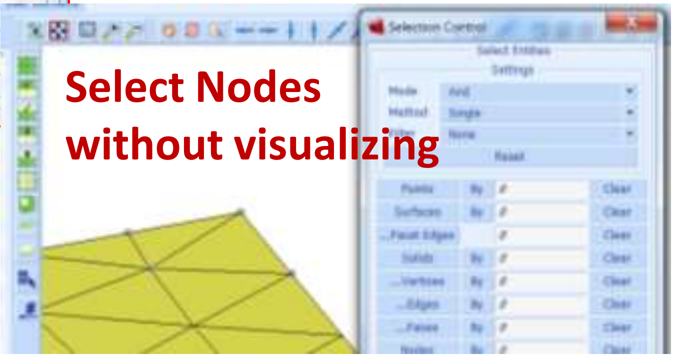
General Mentat Usability Improvements

BIAS MÜHENDİSLİK

- New rendering defaults
  - Default color and contour maps schemas have changed
  - Nodes, points and vertices are not plotted by default
  - Elements, solids and Model Sections are plotted as solids
- Improved visibility of model features/definitions
  - Highlighted/selected nodes, points and vertices have different symbols to better see them
  - Non-editable fields are plotted in italics, to distinguish them from editable fields
  - Filters have been added to easier select post processing quantities
  - Icons have been added to easily set the LookAt point and to switch expanded shell/beam plotting on/off.
- New Interface Options
  - Attach colors and attach symbols can be switched on and off;
  - Even if nodes are not visible, nodes can be selected (e.g. to apply Boundary Conditions)
- Improved User Interaction
  - Commands associated with several icons (e.g. switch plotting nodes on/off) do no longer break run commands



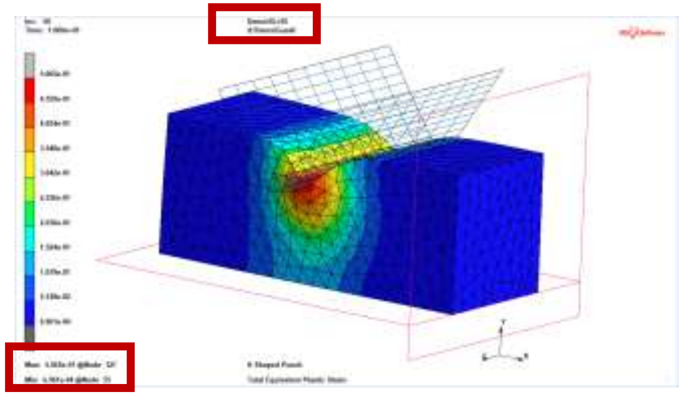




General Mentat Usability Improvements

- Default Multi-Level Undo – preset to 10, can be from 1 to 50

- Materials and Geometric Properties – color assignment

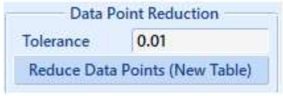


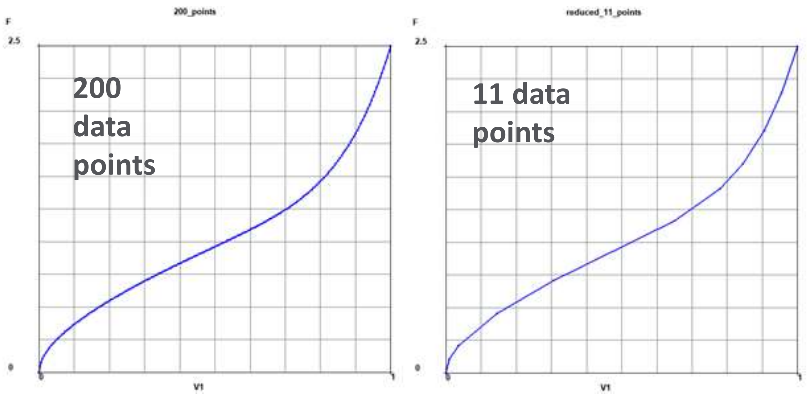
- Default Maximum and Minimum values for contour plots

- New RHS mouse options for
  - “show all”
  - “hide all”
  - “show only”

→ curves, surfaces, elements, model sections, solids, contact bodies and sets

- “Reduce Data Points” option in table menu





Endurica Compatibility with Marc for Prediction of Rubber Fatigue

BIAS MÜHENDİSLİK

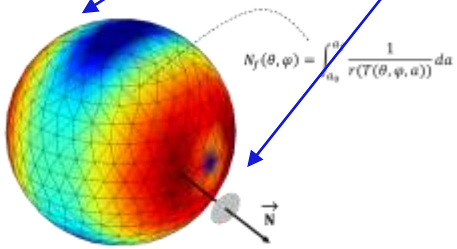
Endurica CL
Fatigue Analysis Software

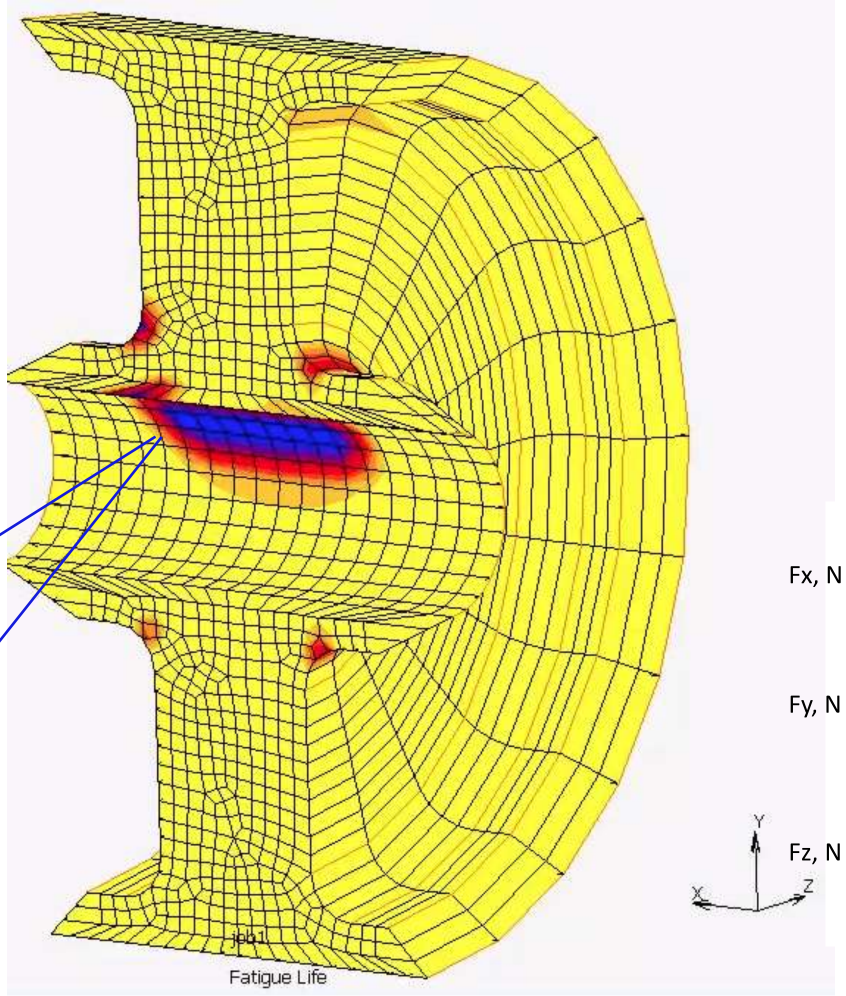
for

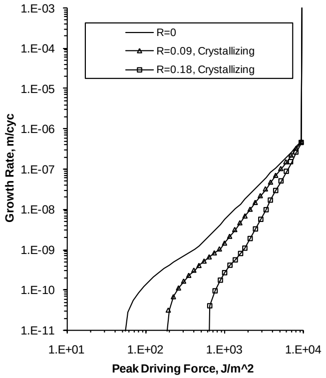
Marc

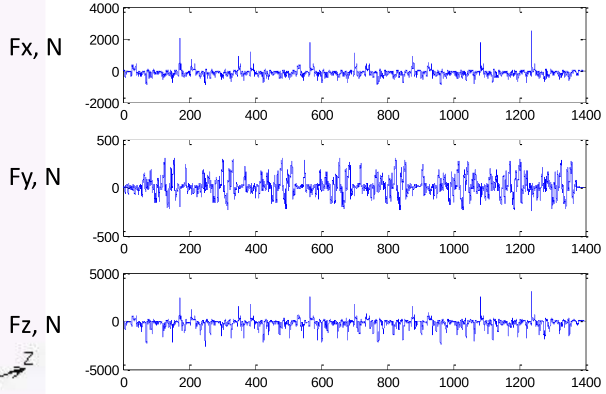
Pohlman

"In the automotive industry, from design to launch is becoming more and more of a time crunch. This is what the Endurica software does in conjunction with our testing. It allows us to cut that timing down, and allows our customer base – heavy truck and off-highway engineering staffs – to do their job and not worry about if they have a durability problem." Steve Pohlman, VP GM Global Elastomers, Tenneco, Rubber & Plastics News, 24 July 2017









Endurica
Get Durability Right

See the workflow demo at: https://youtu.be/HJQtvjRT8VY

Thank You