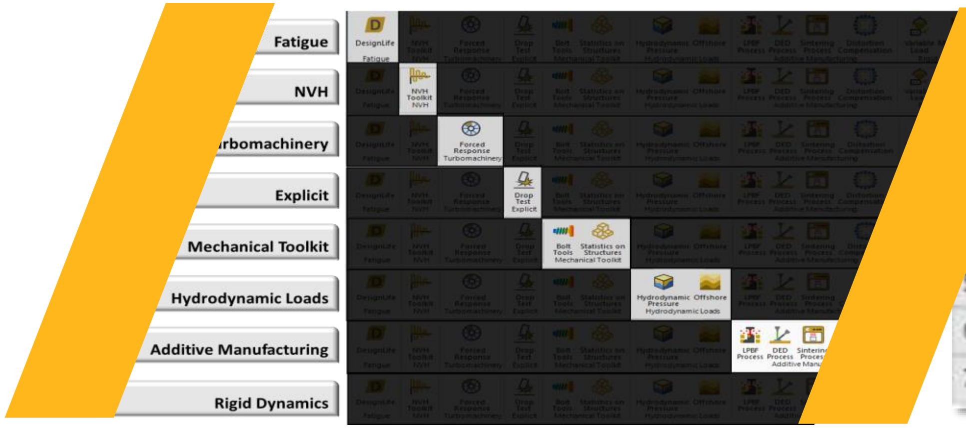
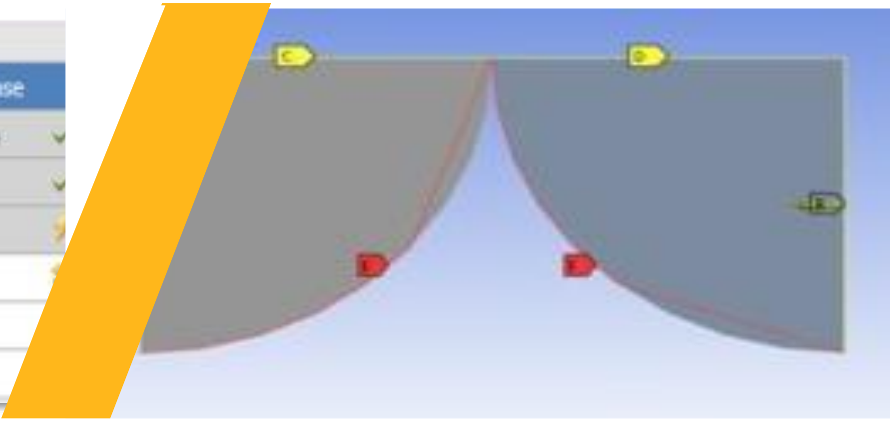
ANSYS Mechanical 2022R2 新功能介绍

新科益系统与咨询（上海）有限公司

Intuitively designed, integrated and dependable solver technology





New intuitively designed, and customizable toolbar for Add-ons

- ✓ Access multipurpose workflows quickly and efficiently
- ✓ On-Off button for load/unload
- ✓ Training content under Ansys Help and Ansys ALH

Robust workflow improvements of MSUP Harmonic and MSUP Transient

- ✓ Reduce overall disk space requirement by 4X
- ✓ Solver speed performance improvement
- ✓ Data mapping time from structural to acoustics harmonic dropped by 50%

Increased exposure of Coupled Field analysis

- ✓ Enables easier workflows for sensor design, MEMs devices and actuators
- ✓ ability to model piezoelectric and acoustic degrees of freedom
- ✓ Includes a library of commonly used piezoelectric materials used in sensor design

Table of Contents

New Feature:

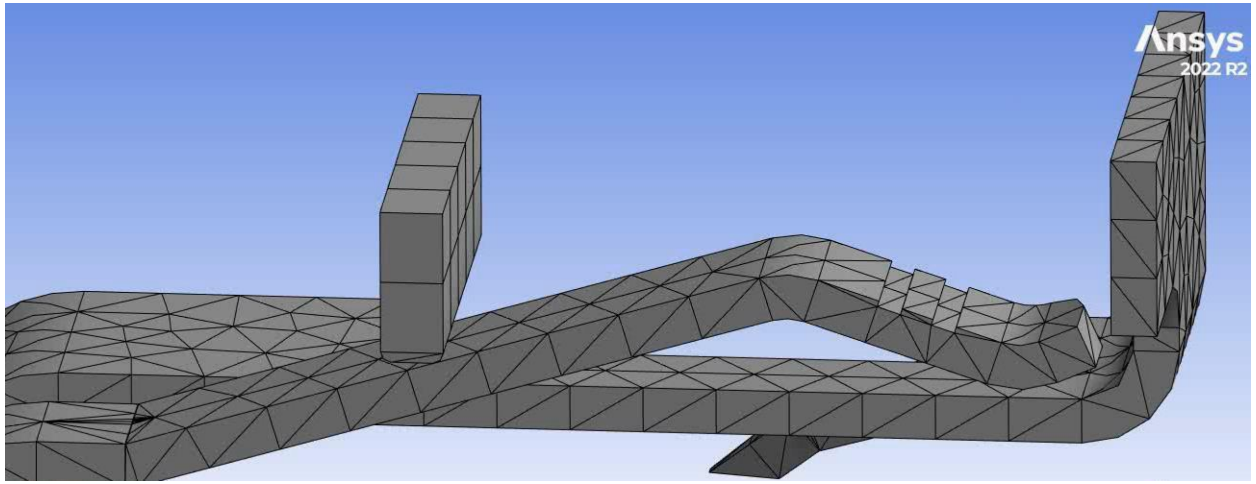
A new unified contact detection option

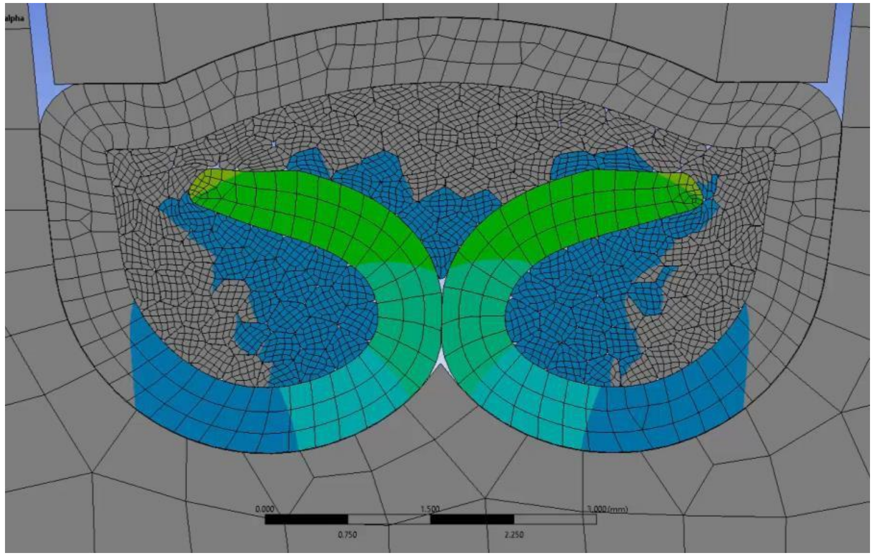
Value Provided:

- Improves solution robustness for non-smooth contact applications

A Unified Contact Detection Method

- A new unified contact detection option (KEYOPT(4)=5) is exposed in 2022R2 (available in Mechanical) which combines three individual contact detection methods together: Gauss point (KEYOPT(4)=0), nodal point (KEYOPT(4)=2), surface projection (KEYOPT(4)=3).
- It adds more contact constraint points at contact interface and results in much less penetration and less mesh sensitivity.
- It robustly solves non-smooth contact problems with different contact scenarios which were generally solved by explicit solvers in the past.





CAD-IT CONSULTANTS

New Feature:

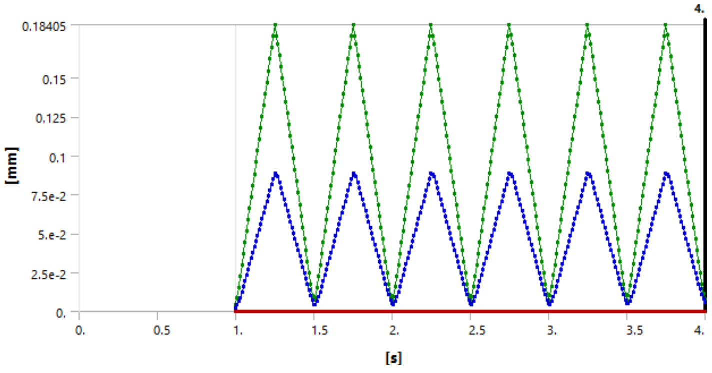
Accurate wear modeling and Fretting Fatigue life prediction

Value Provided:

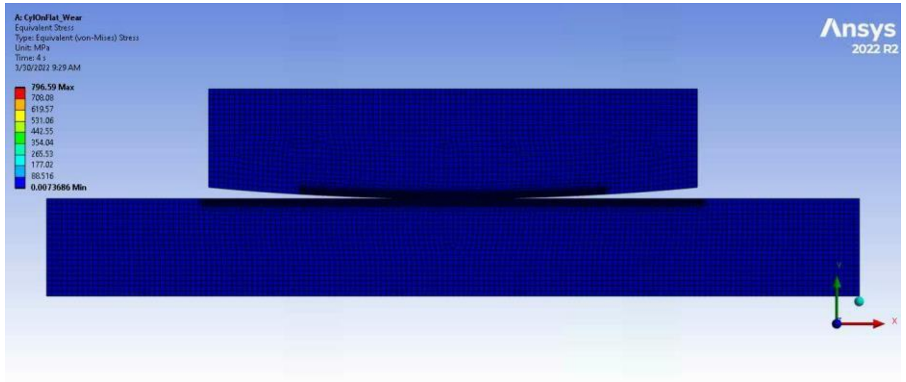
- Automatically scales the wear process so that a few simulation cycles can represent several hundred real cycles, thus enabling accurate prediction of fretting fatigue life

Automatic Wear Scaling-Fretting Fatigue

- Wear & Fretting fatigue are slow processes- simulation for the whole process is prohibitively expensive
- New feature to automatically scale the wear to simulate large number of cycles
- Accurate Wear modeling + nCode=Accurate fretting life prediction



Applied sliding amplitude (mm)	With wear			Experimental life (cycles)	Without wear		
	Physical cycles	Calculated life (repeats)	Calculated life (cycles)		Physical cycles	Calculated life (repeats)	Calculated life (cycles)
0.184	1,385	30.08	41,661	43,983	6	2,398	14,388
0.147	1,485	27.22	10,422	37,717	6	2,278	13,688
0.073	1,888	20.71	39,100	28,800	6	2,972	17,832



Life prediction with new autoscaling + nCode match experiments –both qualitatively (wear increases fretting life as see in experiments) and quantitatively

Automatic scaling enables-
6 simulation cycles = 1385 real cycles

New Feature:

New Toolbar for Ansys Mechanical Add-ons

Value Provided:

- Access multipurpose workflows quickly and efficiently
- On-Off button for load/unload
- Training content under Ansys Help and Ansys ALH

CAD-IT CONSULTANTS

Ansys Mechanical Add-ons : Value Messaging

Owned, Developed and Managed by Ansys

Part of installation package of Ansys Mechanical Product/Application

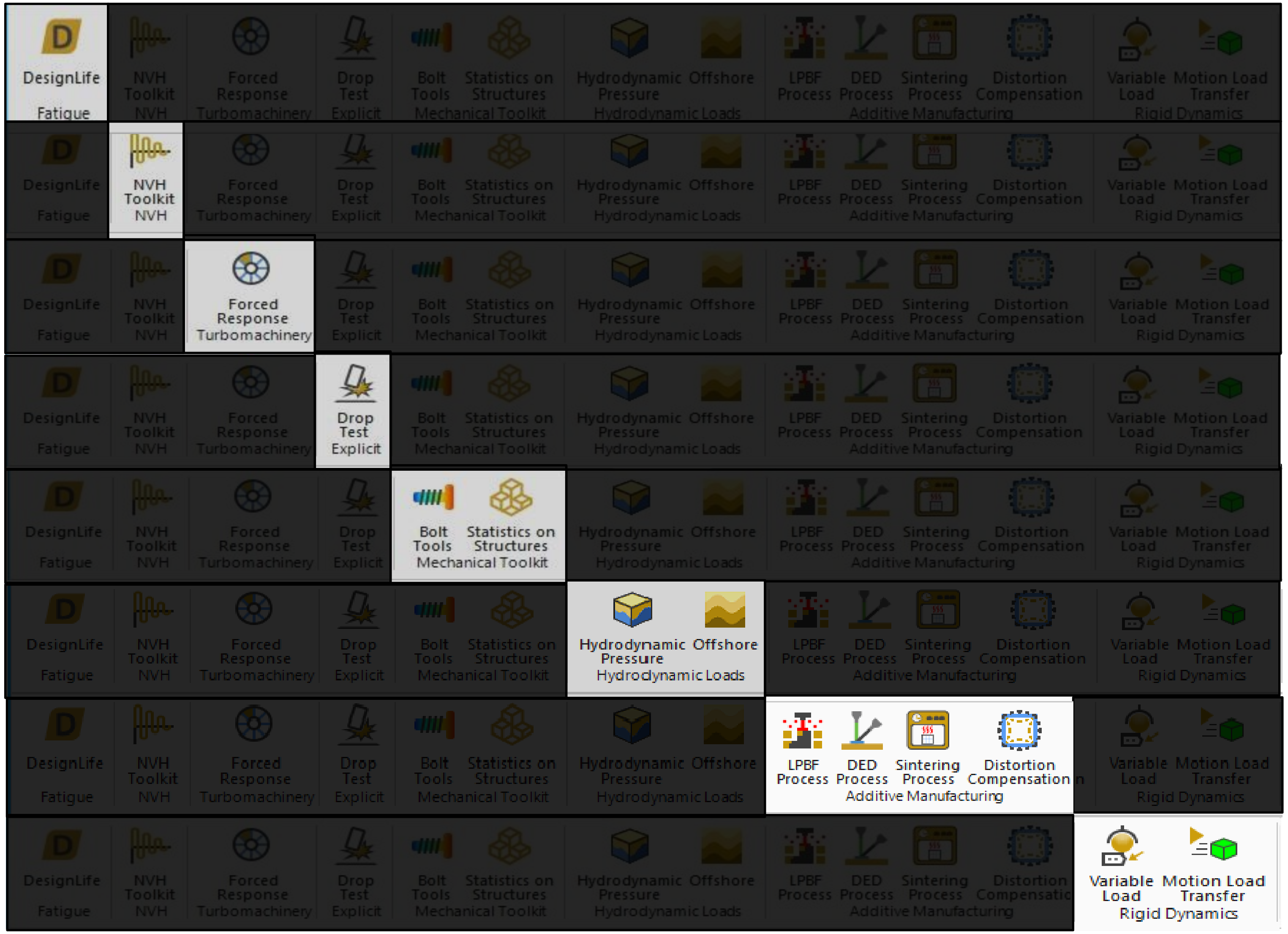
Aligned with targeted Industry or application requirements

Ansys Mechanical Add-ons : 2022 R2 – Enhancements

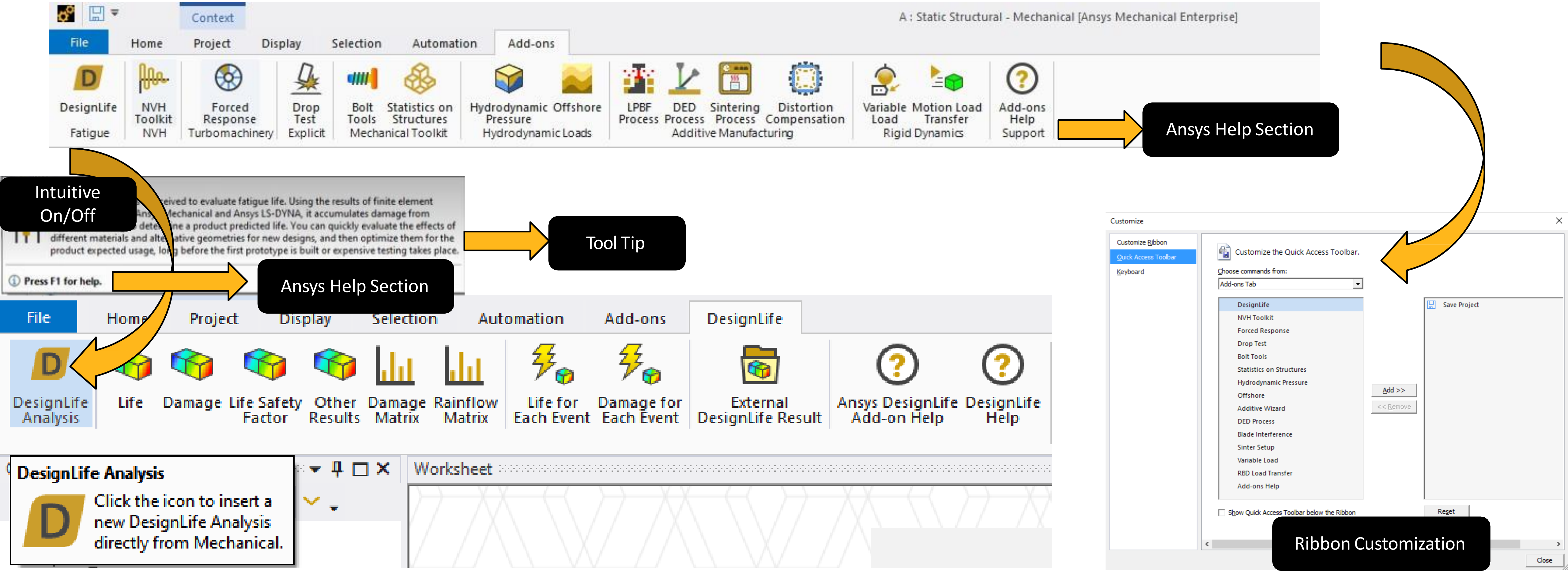
- **Easy access & Intuitive design**
- **Available under Ansys Mechanical Toolbar**
- **Not loaded by default**
- **On-Off button for load/unload**
- **Sufficient training content under Ansys Help and Ansys ALH**
- **Usage/Popularity can be tracked under APIP Program**
- **License controlled**

Ansys Mechanical Add-ons : 2022 R2 – 14 Different Add-ons

- Fatigue
- NVH
- Turbomachinery
- Explicit
- Mechanical Toolkit
- Hydrodynamic Loads
- Additive Manufacturing
- Rigid Dynamics



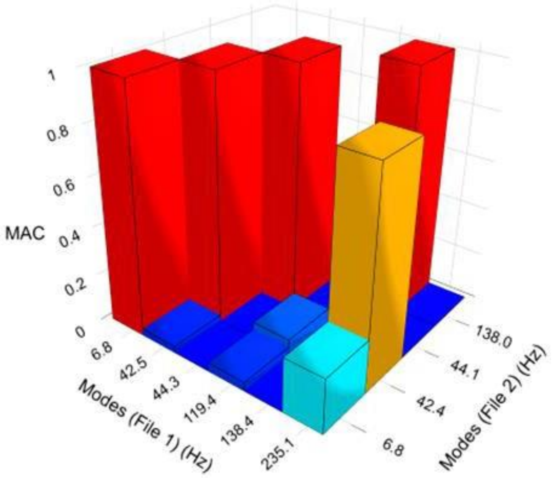
Ansys Mechanical Add-ons : 2022 R2 – Design

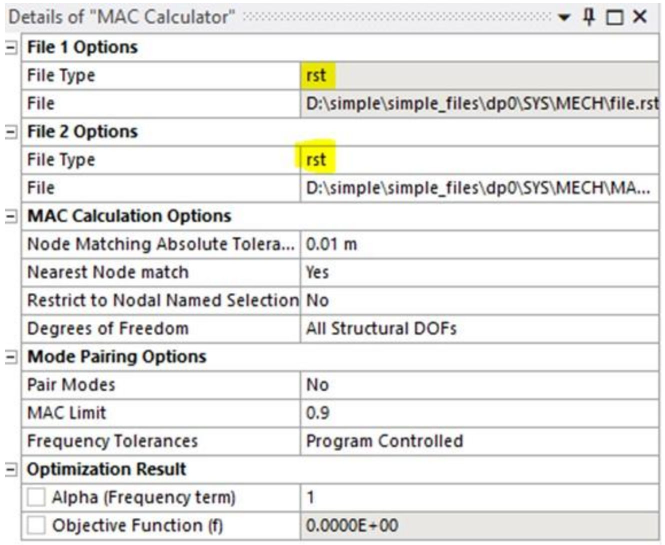


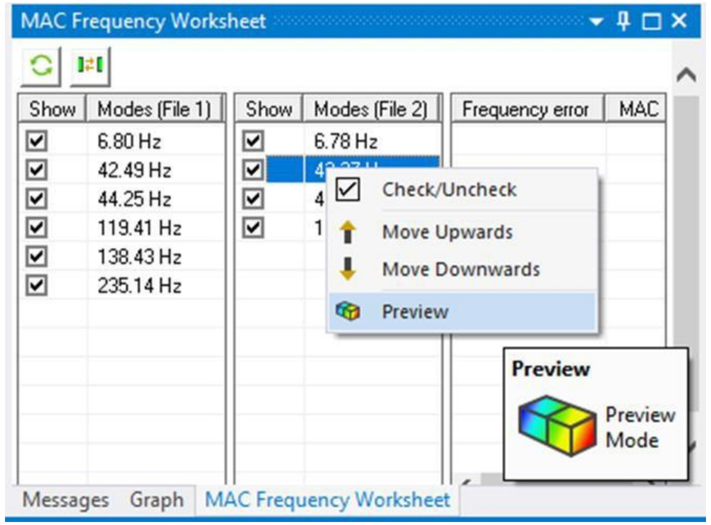
NVH Toolkit Add-on

NVH Toolkit Add-on: MAC UI enhancements

- Addition of the 3D MAC Table.
- Added support for rst-rst file comparison without model rotation. This option is enabled to identify modal changes in computational models if design changes are introduced (e.g. change in the materials, mesh, contact hypotheses…), but not changes in terms of global positioning. The rest of the MAC Calculator functionalities continue to be applicable.
- Mode Preview implemented in the Frequency Worksheet to be able to animate modes before solving the MAC Calculator.







NVH Toolkit Add-on: COMAC Calculation

- Coordinate Modal Assurance Criterion (COMAC) has been added as an additional output of MAC. COMAC is computed for each pair of matched Degrees of Freedom

Matched Nodes Table

Nodes (File 1)	Nodes (File 2)	COMAC (UX)	COMAC (UY)	COMAC (UZ)
169	6	0.0828	0.5126	0.0194
173	7	0.0438	0.5114	0.0117
180	3	0.0461	0.5114	0.0124
184	2	0.0871	0.5126	0.0232
188	1	0.0000E+00	0.0000E+00	0.0000E+00
194	5	0.0000E+00	0.0000E+00	0.0000E+00
195	8	0.0733	0.2718	7.9320E-03
201	4	0.0686	0.2718	7.8081E-03

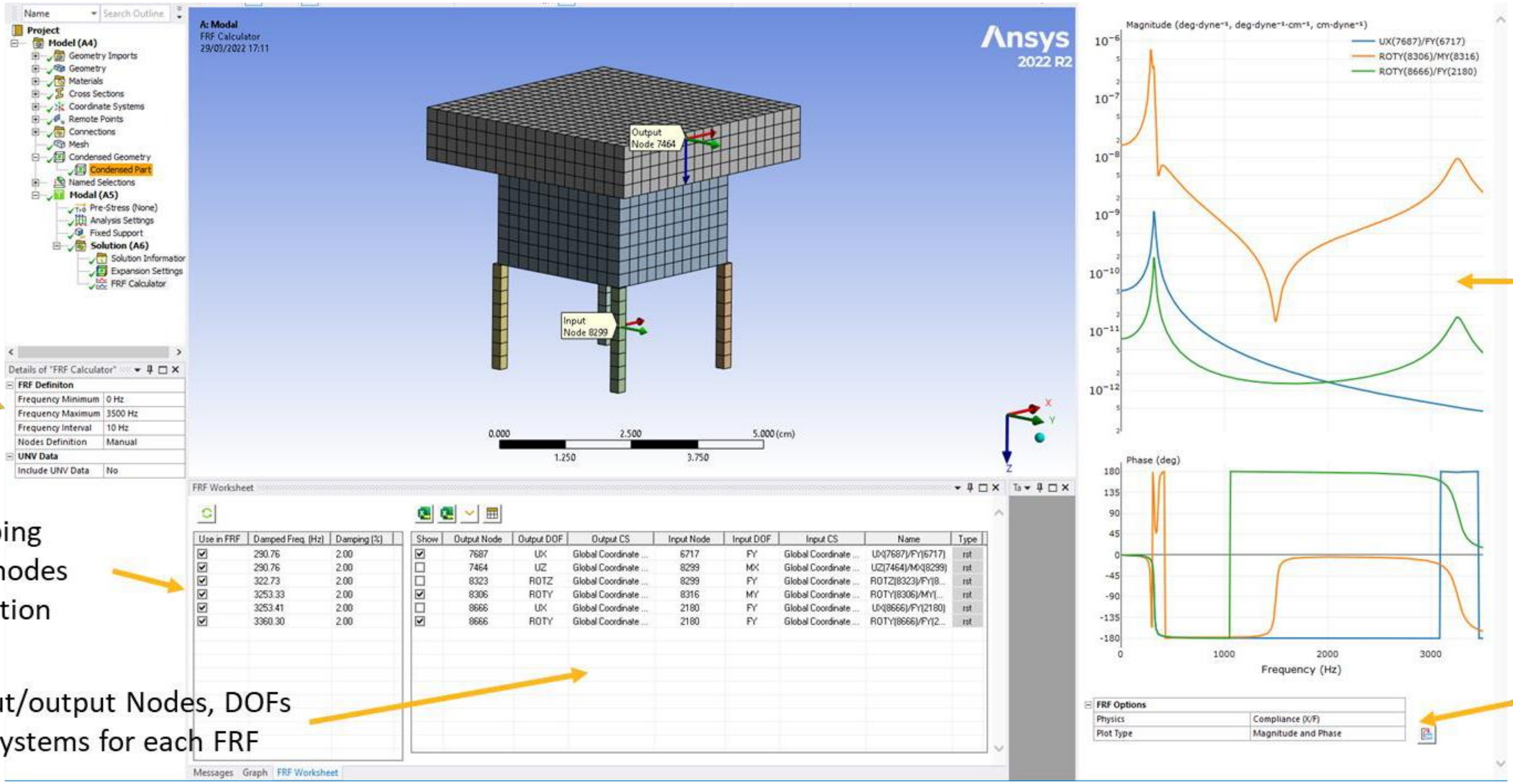
- As MAC, COMAC is bounded between 0 and 1.
- With COMAC, users can identify the potential sources of low MAC (associated with DOF pairs with low COMAC) or the reasons for a high MAC (DOF pairs with high COMAC).

NVH Toolkit Add-on: FRF Calculator

- The Frequency Response Function (FRF) Calculator is a brand-new post object that allows users to compute the relationship between a generalized input force and a generalized output displacement as a function of frequency:

Frequency range and Nodes definition customization

Damping and modes definition



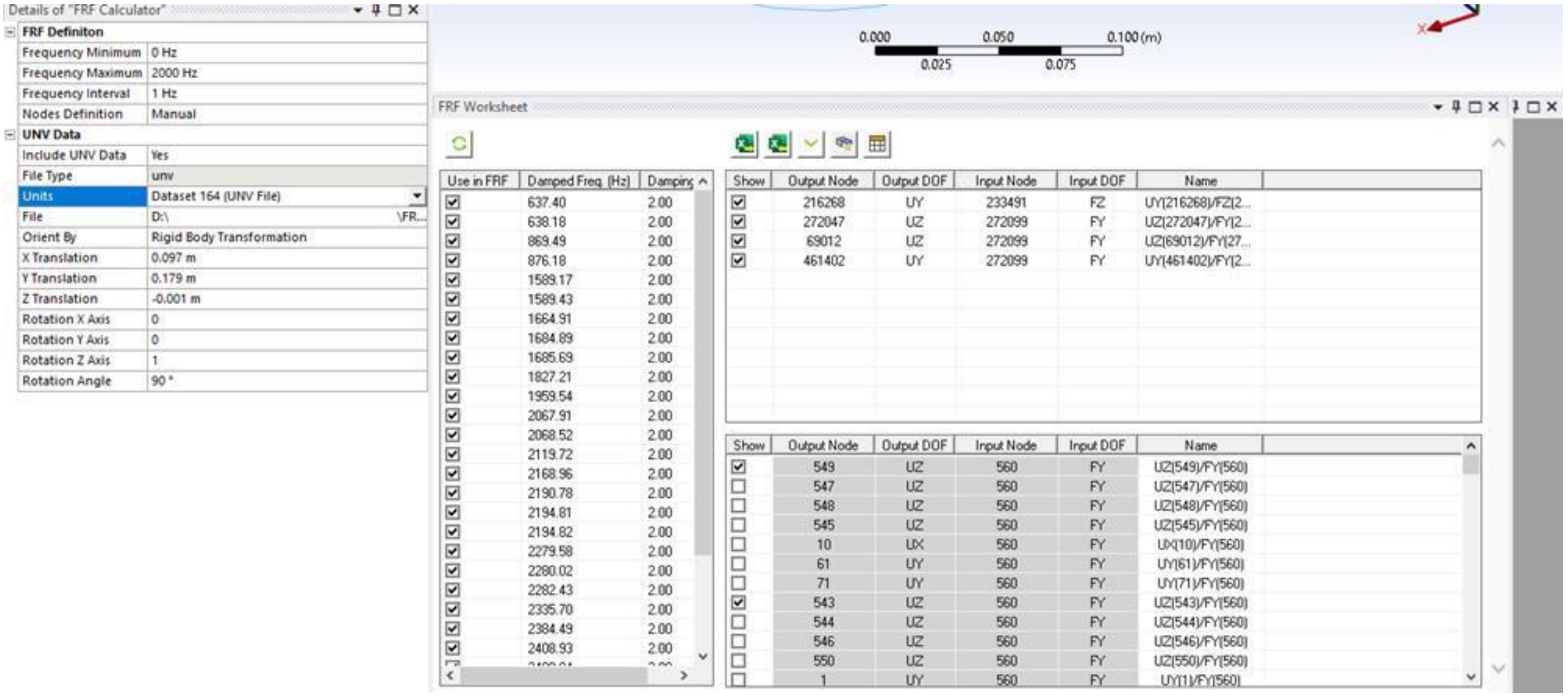
FRF Plot either in Magnitude/Phase or Real/Imaginary formats. The plot is expressed in terms of Compliance (X/F), Mobility (V/F) or Accelerance (A/F)

Definition of input/output Nodes, DOFs and Coordinate Systems for each FRF

Plot definition

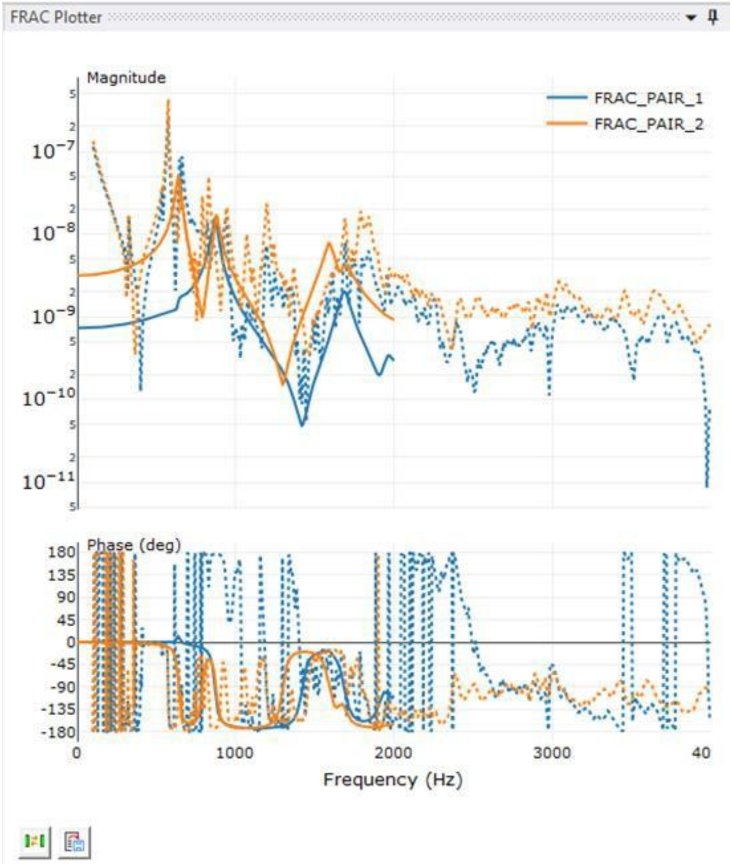
NVH Toolkit Add-on: FRF Calculator, connection with UNV data

- UNV Data has been incorporated into the FRF Calculator to be able to analyze experimental FRFs and compare them with computational FRFs.
- Fully defined model orientation is available (by Coordinate System, Rigid Body Transformation or 3 Node Alignment).



NVH Toolkit Add-on: FRAC Calculator

- A Frequency Response Assurance Criterion (FRAC) Calculator has been incorporated into the FRF Calculator as an additional tool to measure correlation between FRFs.
- FRAC can be computed between two computational FRFs or between a computational and a experimental FRF.
- An automatic algorithm that matches node pairs between the experimental and computational models has been implemented, and users are be able to quickly compare the same FRFs in both models.

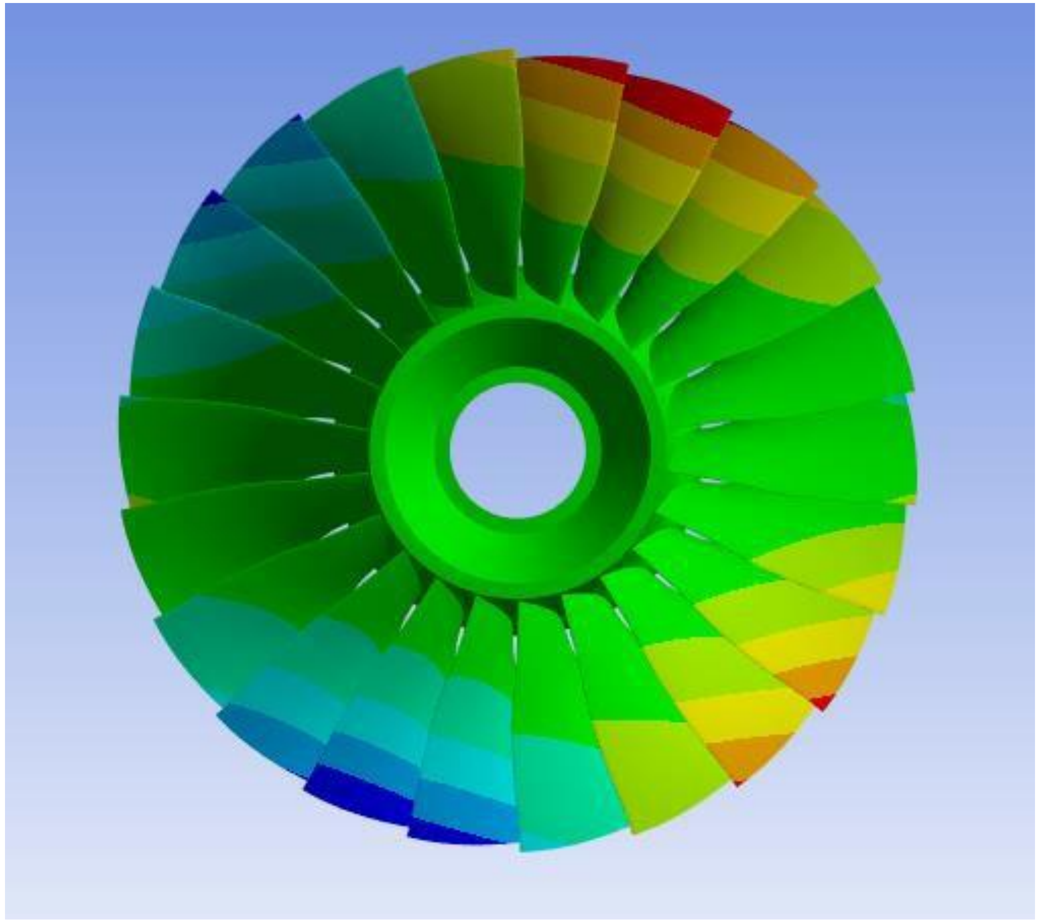


Show	FRF1	FRF2	Name	FRAC
☑	UY(216268)/FZ(233491)...	UZ(543)/FY(560) (unv)	FRAC_PAIR_1	0.0124
☑	UZ(272047)/FY(272099)...	UZ(549)/FY(560) (unv)	FRAC_PAIR_2	5.3934E-03
☐	UZ(69012)/FY(272099) (...	UZ(543)/FY(560) (unv)	FRAC_PAIR_3	0.0339
☐	UY(461402)/FY(272099)...	UY(21)/FY(560) (unv)	FRAC_PAIR_4	0.0569

Forced Response Add-on

Forced Response Add-on

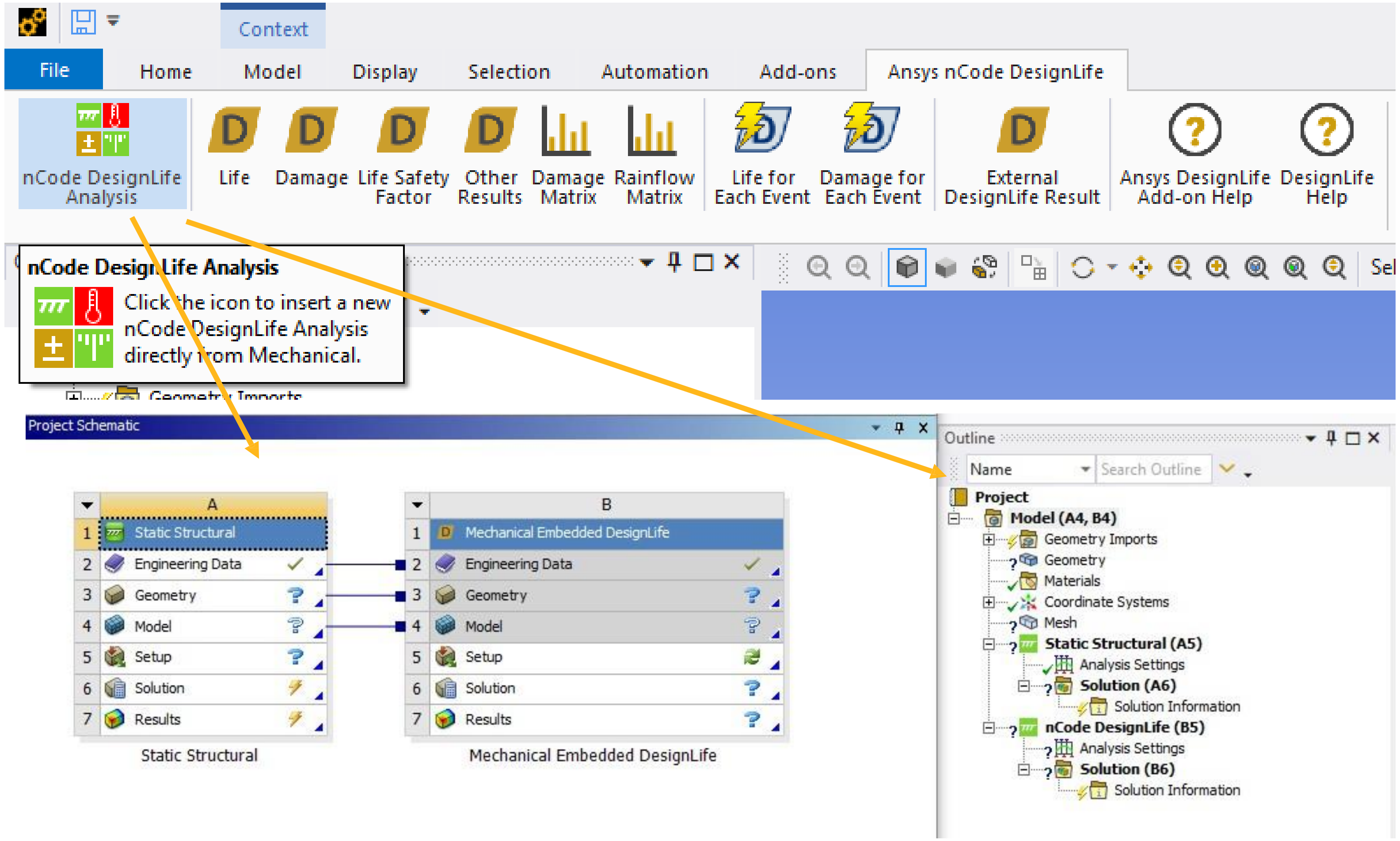
- The Forced Response Tool provides a comprehensive way of investigating the aeromechanics and reliability of turbomachinery blade rows.
- In conjunction with Ansys Computational Fluid Dynamics (CFD) tools, the Forced Response Tool accurately predicts the structural vibrations of industrial components. Advanced physical phenomena such as airfoil aeroelasticity, flutter, deterministic and probabilistic mistuning, as well as an array of loading and boundary conditions can be modeled and studied.
- Efficient high-fidelity modeling is achieved using a combination of core technologies such as cyclic symmetry, mode-superposition, and innovative techniques to model mistuning and aeroelasticity in a reduced space.
- This tool provides an intuitive, reliable, and efficient way to simulate turbomachinery blade rows.



DesignLife Add-on

DesignLife Add-on: Analysis from Mechanical

- Ability to insert the DesignLife analysis directly from Mechanical.



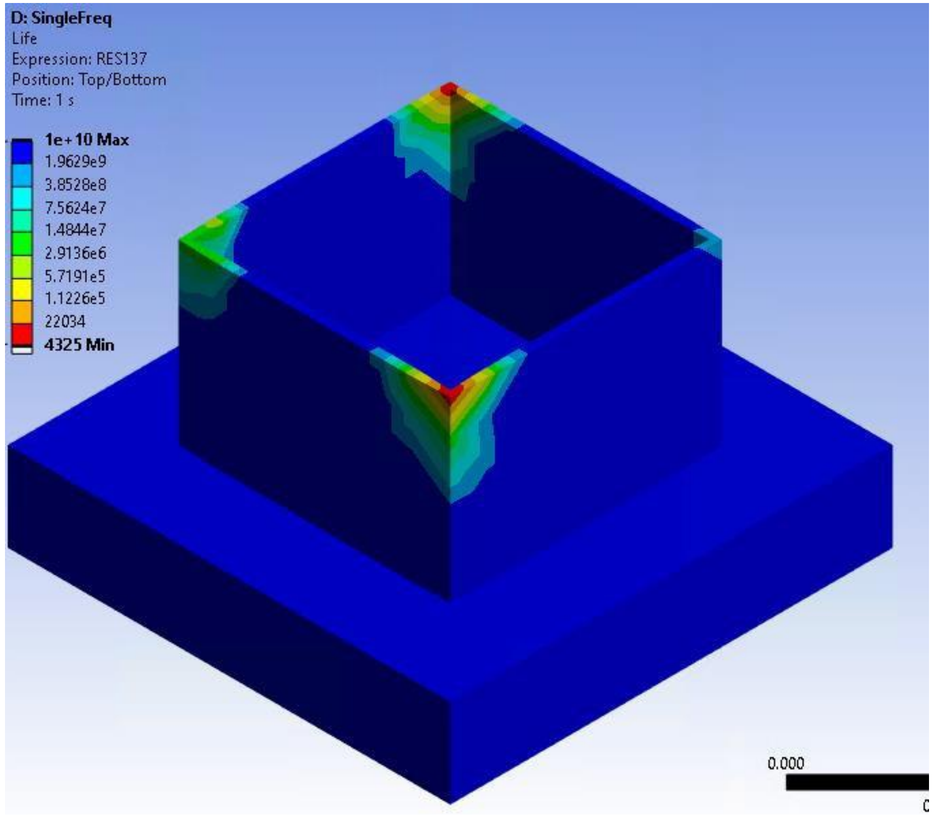
DesignLife Add-on: Harmonic Direct – Vibration Fatigue

Vibration Fatigue Load	Stress	Strain	Shell Seam Weld	Solid Seam Weld
PSD Loading, Including: - Static Offset Case - Single and Multiple Events	✓	✓	✓	✓
Single Frequency Loading, Including: - Static Offset Case - Single and Multiple Events	✓	✓	✓	✓
Frequency Range Loading, Including: - Static Offset Case - Single and Multiple Events	✓	✓	✓	✓
Sine On Random Loading, Including: - Static Offset Case - Single and Multiple Events	✓	✓	✓	✓

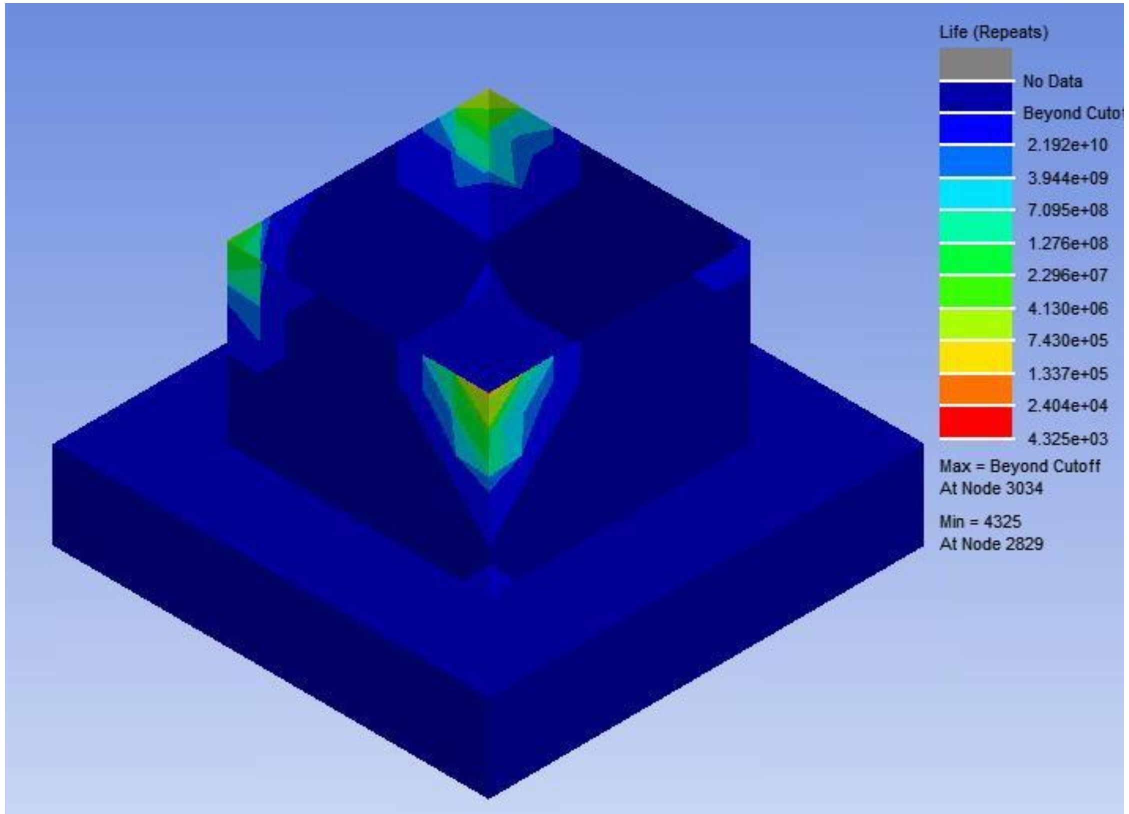
Single Frequency Vibration Load – Stress Case

- Life Result Comparison between DesignLife Add-on and Standalone:

DesignLife Add-on



DesignLife Standalone

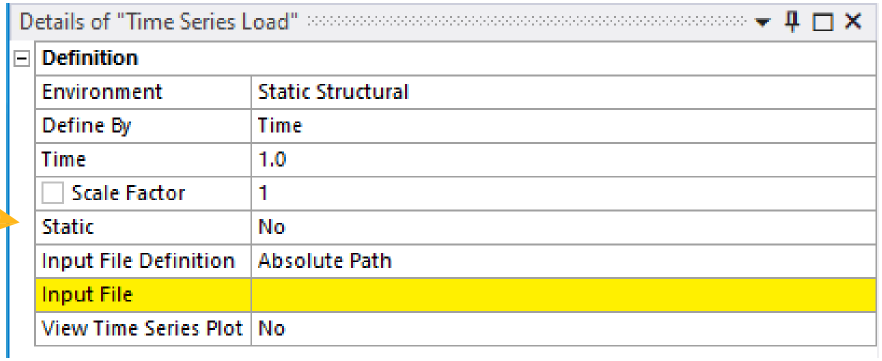


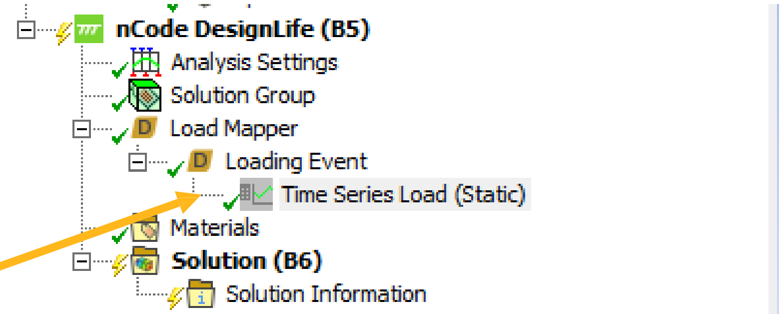
DesignLife Add-on: Expose Static (Preload) Option within Time Series

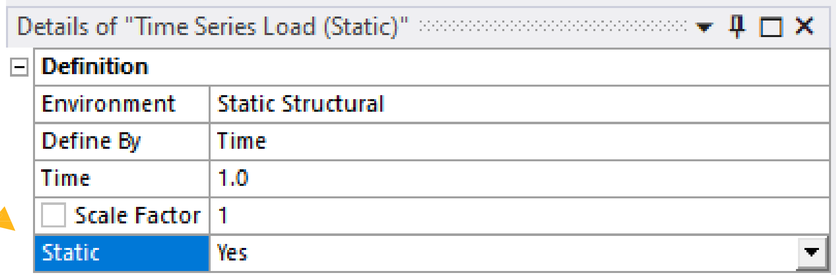
- In order to define the static load case (preload)
  - Set the “Static" option to “Yes”.
  - Then the input file selection will not be required.

•This sets the Static State to “Yes” or “No” within the command SetLoadCaseStaticState inside the input.dcl.

•For clarity, the name of the load will be updated to include (Static) specification.

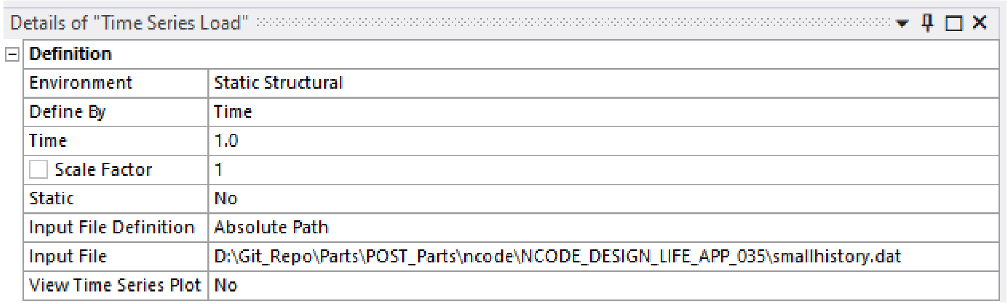






DesignLife Add-on: Time Series Input File (Absolute or Relative Path)

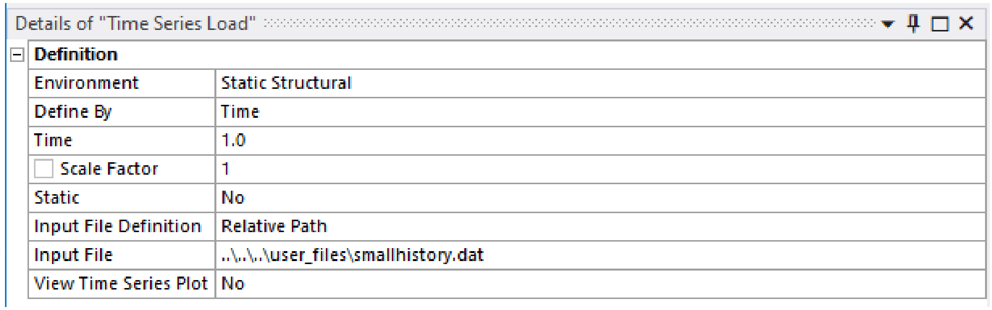
- Input File Definition:
  - Choose between “Absolute Path” or “Relative Path”.
- If “Absolute Path” is used
  - The time series file will be loaded from the path selected in your local device. This means if the file is moved to a different place, the time series input file will no longer be found.



Details of "Time Series Load"

Definition	
Environment	Static Structural
Define By	Time
Time	1.0
Scale Factor	1
Static	No
Input File Definition	Absolute Path
Input File	D:\Git_Repo\Parts\POST_Parts\ncode\NCODE_DESIGN_LIFE_APP_035\smallhistory.dat
View Time Series Plot	No

- If “Relative Path” is used
  - The local file selected will be copied to the user_files folder within the project folder. Instead of loading the file from the local path, it will load it from the user_files. So make sure, if you modify the local file, and you want your project to reflect so, to reload it again so that its updated into the user_files.



Details of "Time Series Load"

Definition	
Environment	Static Structural
Define By	Time
Time	1.0
Scale Factor	1
Static	No
Input File Definition	Relative Path
Input File	..\..\..\user_files\smallhistory.dat
View Time Series Plot	No

DesignLife Add-on: Post Processing Static Failure

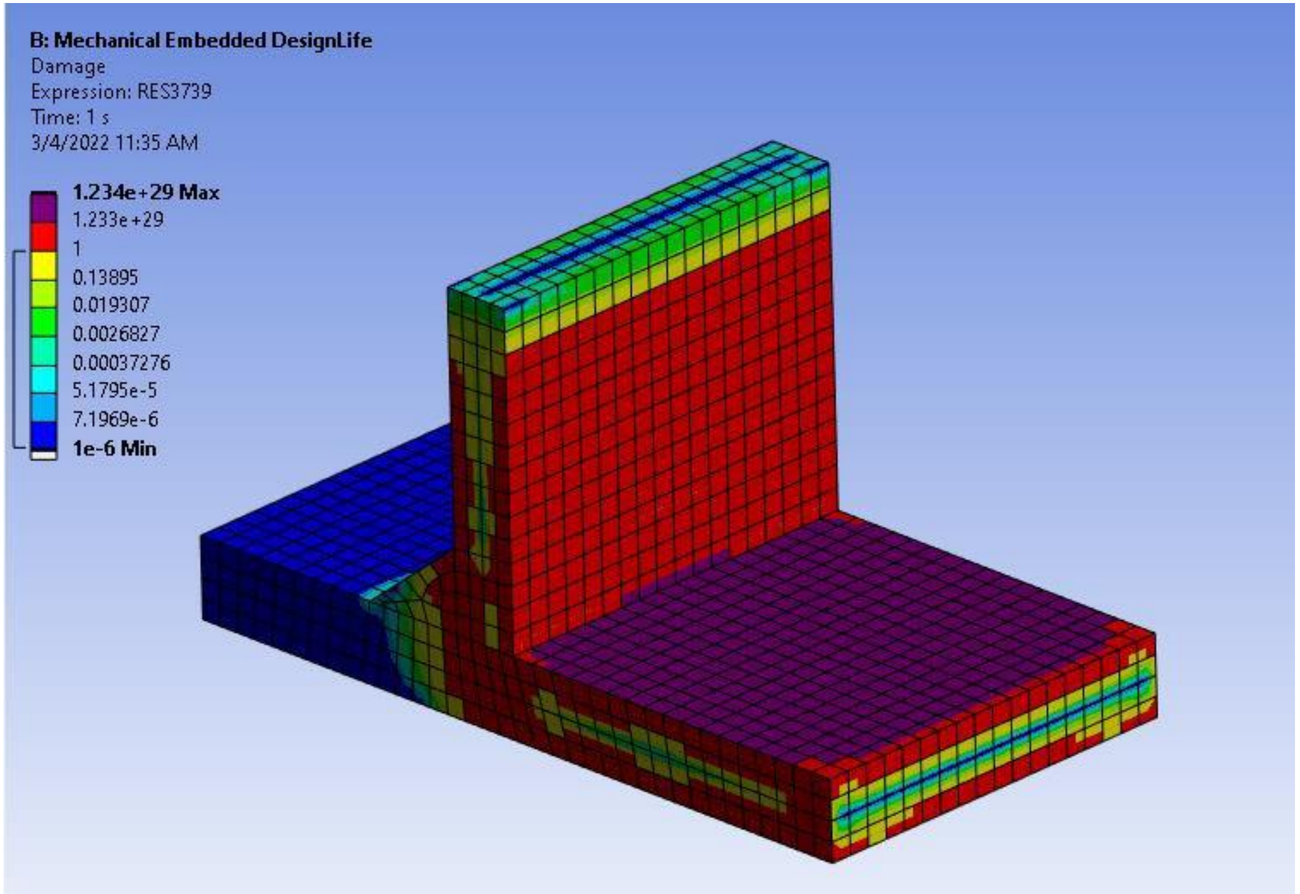
- Life and Damage Results
- For Time Based, Stress-Life Analysis (SN), the Static Failure Damage is set to 1.234e29 in the input.dcl file consumed by nCode solver.
  - *SetProperty("SNEngine_Fatigue",StaticFailureDamage,"1.234E29")*

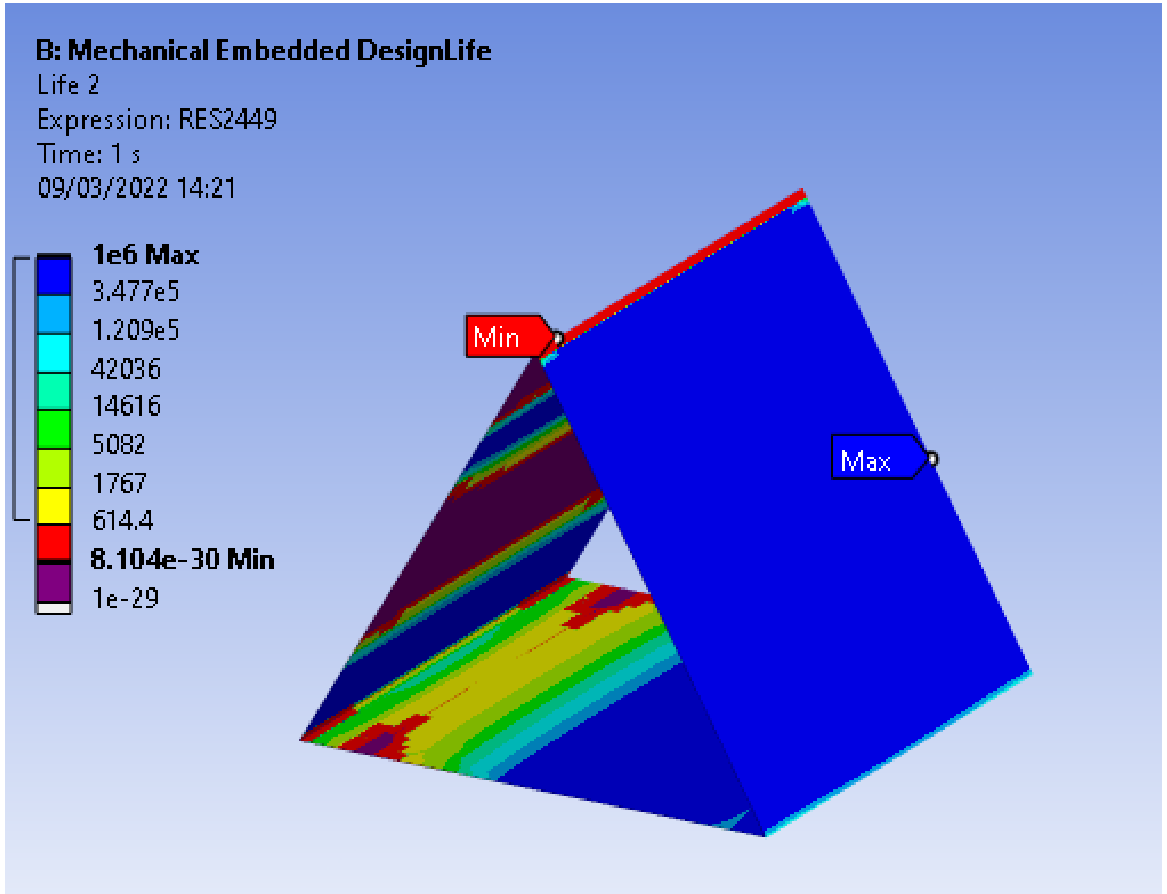
- If Static Failure is detected, Mechanical will issue the warning message 1:

  - <u>Warning message 1:</u> "Calculated alternating stress in some areas exceeded the UTS, which indicates Static Failure. For those areas, Life is set to the Static Failure Life and Damage is set to the Static Failure Damage; plotted as purple contours."

Post Processing Static Failure – Life and Damage Result

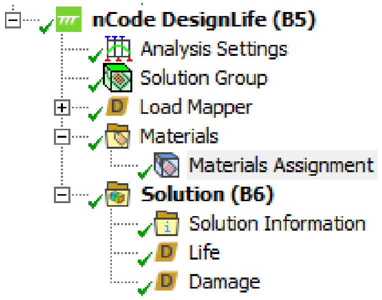
- The purple band in the legend, corresponding to the “Static Failure Life” or “Static Failure Damage” value, represents all the areas where Static Failure is reported.

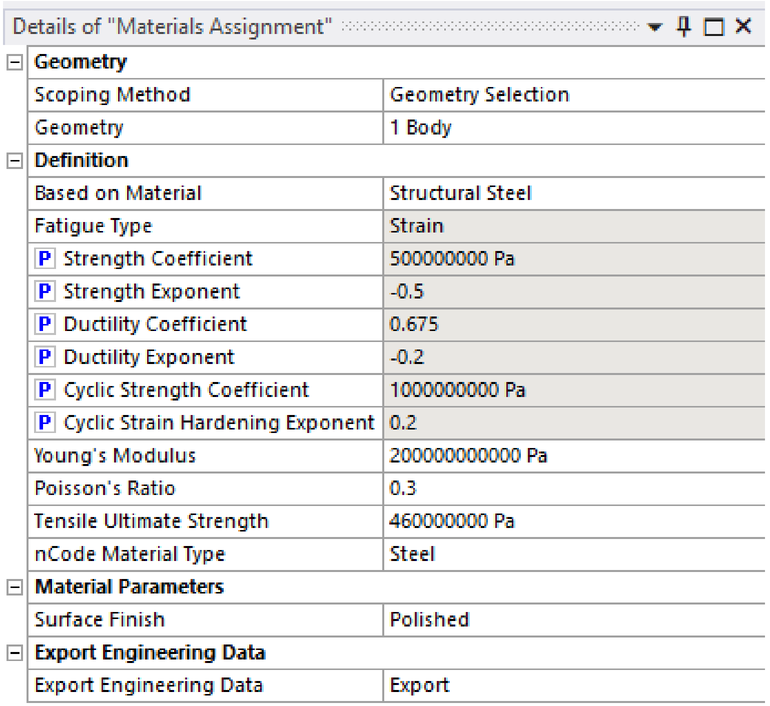




DesignLife Add-on: Exposed Strain Analysis Parameters

- Geometry
  - Scoping method
  - Geometry
- Definition
  - Based on Material (automatically filled from scoped geometry)
  - Fatigue Type (Strain, read only)
  - Strength Coefficient - Parametrizable
  - Strength Exponent - Parametrizable
  - Ductility Coefficient - Parametrizable
  - Ductility Exponent - Parametrizable
  - Cyclic Strain Coefficient - Parametrizable
  - Cyclic Strain Hardening Exponent - Parametrizable
  - Young´s Modulus
  - Poisson´s Ratio
  - Tensile Ultimate Strength
  - NCode Material Type:
    - Grey Cast Iron, Nodular Cast Iron, Malleable Cast Iron, Cast Steel, Steel, Aluminum, Cast Aluminum
- Material Parameters
  - Surface Finish:
    - Polished, Ground, Machined, Poor Machined, As Rolled, AsCast.

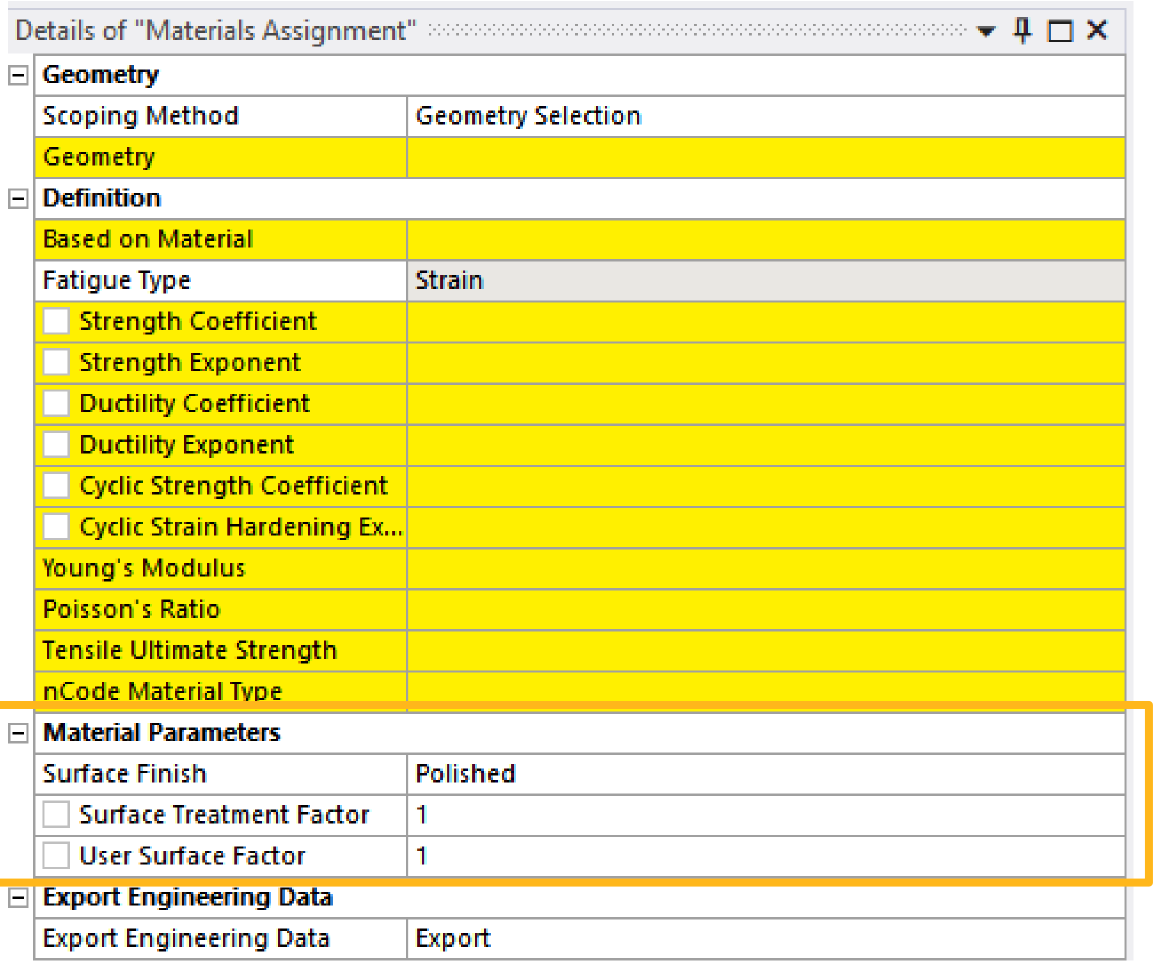



Details of "Materials Assignment"

Geometry	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Based on Material	Structural Steel
Fatigue Type	Strain
P Strength Coefficient	500000000 Pa
P Strength Exponent	-0.5
P Ductility Coefficient	0.675
P Ductility Exponent	-0.2
P Cyclic Strength Coefficient	1000000000 Pa
P Cyclic Strain Hardening Exponent	0.2
Young's Modulus	200000000000 Pa
Poisson's Ratio	0.3
Tensile Ultimate Strength	460000000 Pa
nCode Material Type	Steel
Material Parameters	
Surface Finish	Polished
Export Engineering Data	
Export Engineering Data	Export

DesignLife Add-on: Exposed Material Parameters

- Material Parameters for SN and EN
  - Surface Finish:
    - Polished, Ground, Machined, Poor Machined, As Rolled, As Cast.
  - Surface Treatment Factor
    - Can be parameterized.
  - User Surface Factor
    - Can be parameterized.



New Feature:

Weld Fatigue Workflow

Value Provided:

- Out of the box automation to handle complex weld locations
- Use expressions based on parent material thicknesses to drive different industry best practices

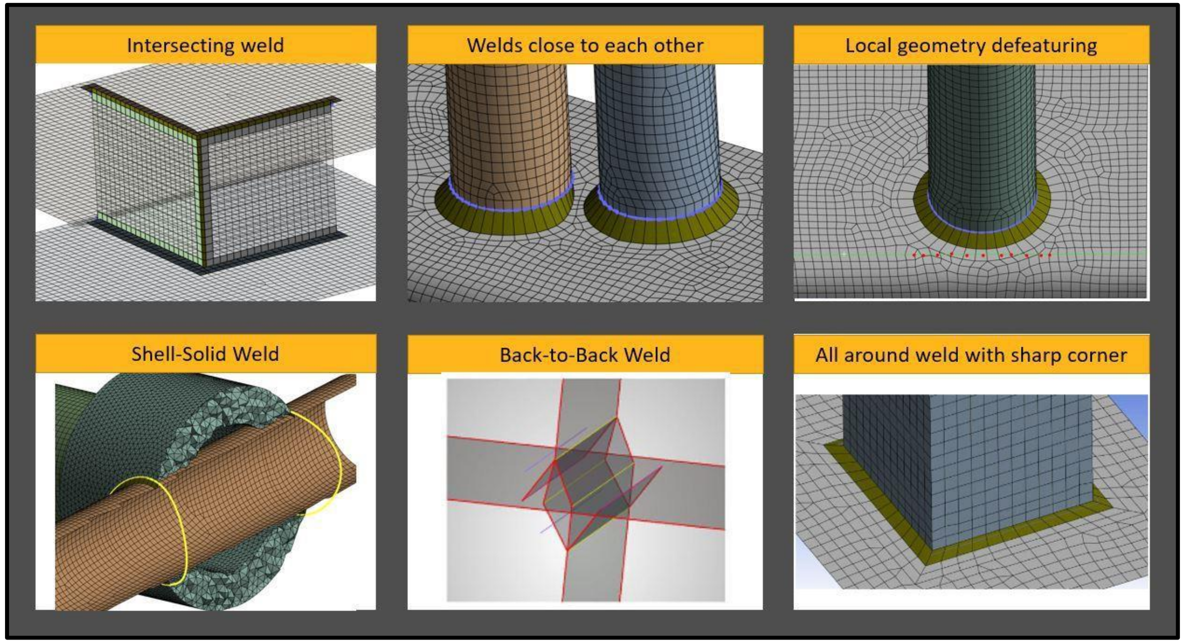
CAD-IT CONSULTANTS

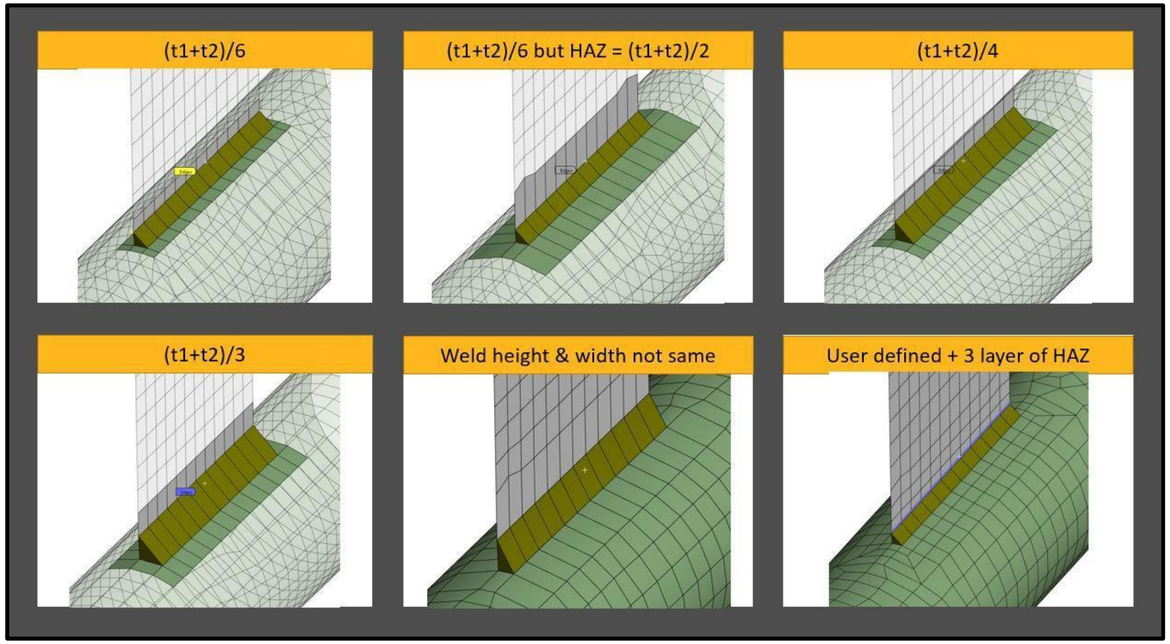
Seam Weld Enhancements

- Handle different complex weld locations with native automation
  - Auto switching of top & bottom faces
  - Back-to-Back welds
  - Handle multiple distinct weld locations part of one weld curve
  - Auto ignore redundant weld curves along thickness
  - Close loop welds
  - Auto retraction &/or squeeze weld &/or HAZ
  - Auto defeaturing

Scope	
Type	Continuous Seam
Source	Mesh
Modeled As	Normal and Angled
Create Using	Curves
Angled Direction	Normal
Use Worksheet	No
Curve Scoping	Geometry Selection
Weld Curve	1 Body
Definition	
Suppressed	No
Adjust Weld Height	Yes
Weld Height (Leg02) Assignment	Expression
Weld Height (Leg02) Expression	(t1+t2)/4
Creation Criteria	Width Based
Weld Width (Leg01) Assignment	Expression
Weld Width (Leg01) Expression	(t1+t2)/4
Edge Mesh Size	Default (3.0 mm)
Create HAZ Layer	Yes
HAZ Distance Assignment	Expression
HAZ Distance Expression	(t1+t2)/4
Number Of HAZ	Default (1)
HAZ Growth Rate	1.2
Generate End-Caps	Yes
Generate Named Selection	No
Intersection Tag (Beta)	
Mechanical Properties	
Material	Structural Steel
Thickness Assignment	Expression
Expression	(t1+t2)/2
Advanced	
Sharp Angle	Default (30.0°)
Connection Tolerance	Default (Program Controlled)
Smoothing	Yes
Lap Weld Angle Tolerance	5.0°

- Use expressions to drive different industry practices
  - Support for expressions (-,+,*,/,(,),^).
  - Available for: Weld Height, Width, HAZ Distance, Thickness





CAD-IT CONSULTANTS

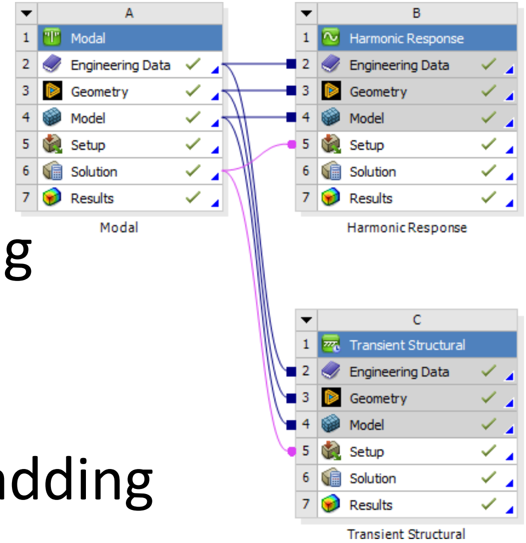
New Feature:

MSUP Harmonic and MSUP Transient workflow Performance Improvement

Value Provided:

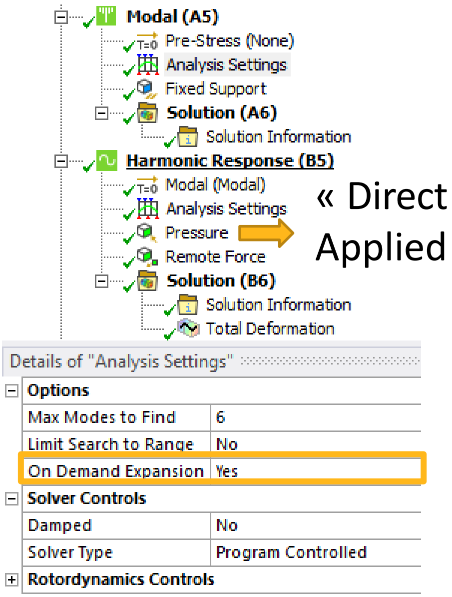
- Reduce overall disk space requirement by 4X
- Solver speed performance improvement
- Data mapping time from structural to acoustics harmonic dropped by 50%

MODDIR for Linked MSUP

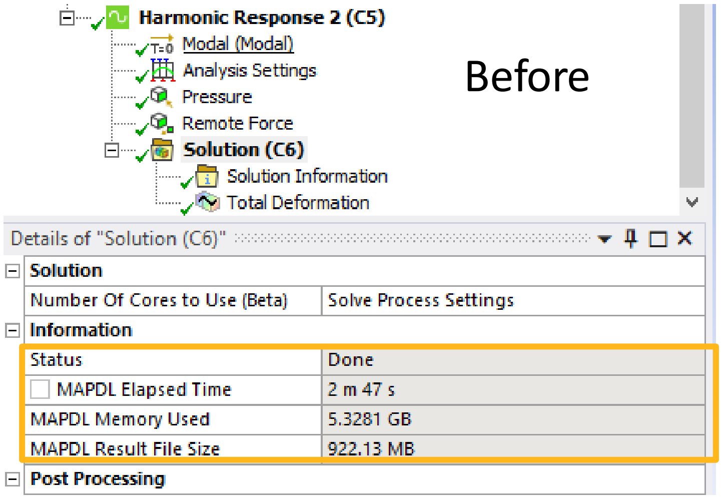


- Reduce results file size and decrease overall time to solve by referencing Modal results files instead of copying them.

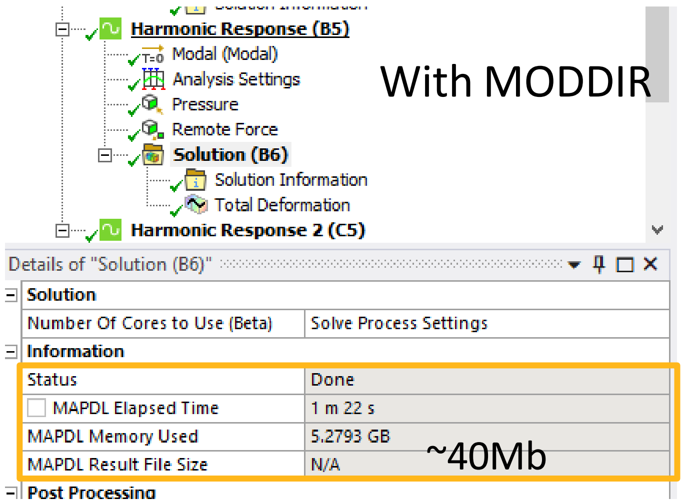
- Needs On Demand Expansion set to Yes in Linked Modal and no loads adding new elements.



« Direct »
Applied By



Before



With MODDIR

~40Mb

Number of elements : ~420 000

Number of modes : 6

Number of frequencies : 100

Modal & MSUP Elemental Temperature

- Element Temperatures is no longer being requested during solution to be stored in the Modal and Mode-superposition analysis results file, which reduces the result file size

Details of "Solution (B6)"

Solution	
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done
MAPDL Elapsed Time	1 m 22 s
MAPDL Memory Used	12.111 GB
MAPDL Result File Size	162.31 MB

Details of "Solution (B6)"

Solution	
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done
MAPDL Elapsed Time	1 m 15 s
MAPDL Memory Used	12.111 GB
MAPDL Result File Size	119.69 MB

Reduced disk space with less data stored in results file

- Using On Demand Expansion, by default mode shapes are no more stored in the result file. Postprocessing is done extracting the data from the mode file(s) thus removing duplication of stored data:

Details of "Solution (B6)"

Solution	
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done
MAPDL Elapsed Time	1 m 11 s
MAPDL Memory Used	12.111 GB
MAPDL Result File Size	120.88 MB

Details of "Solution (B6)"

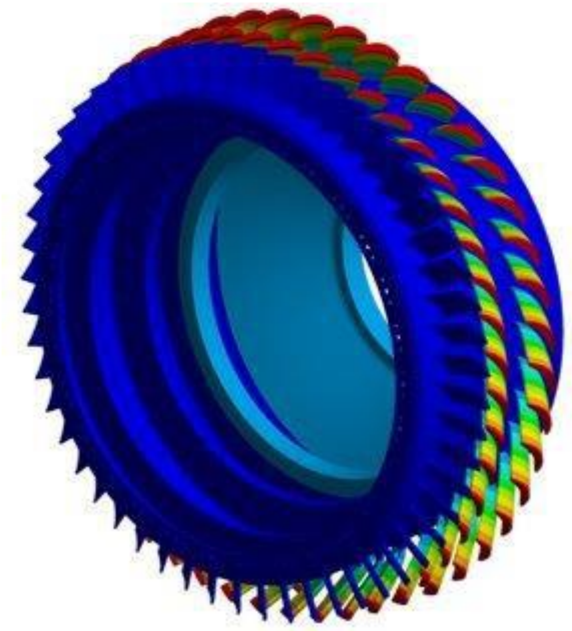
Solution	
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done
MAPDL Elapsed Time	1 m 10 s
MAPDL Memory Used	12.111 GB
MAPDL Result File Size	37.625 MB

Multistage Cyclic Analysis

- Modal and Prestressed Modal analyses now support multiple harmonic indices definition. It allows to improve the accuracy of the results by enriching the solution.

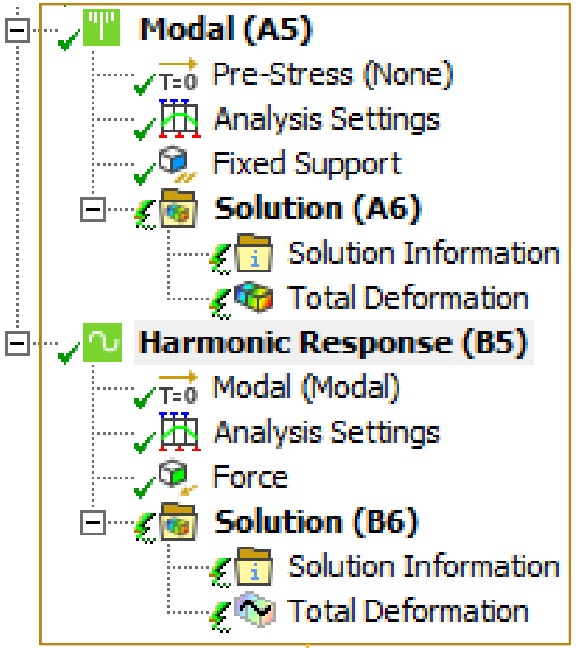
Stage 1					
Nodal Diameters					
HI	HI	N-HI	N+HI	2N-HI	2N+HI
0	0	6	6	12	12
1	1	5	7	11	13
2	2	4	8	10	14
3	3	3	9	9	15

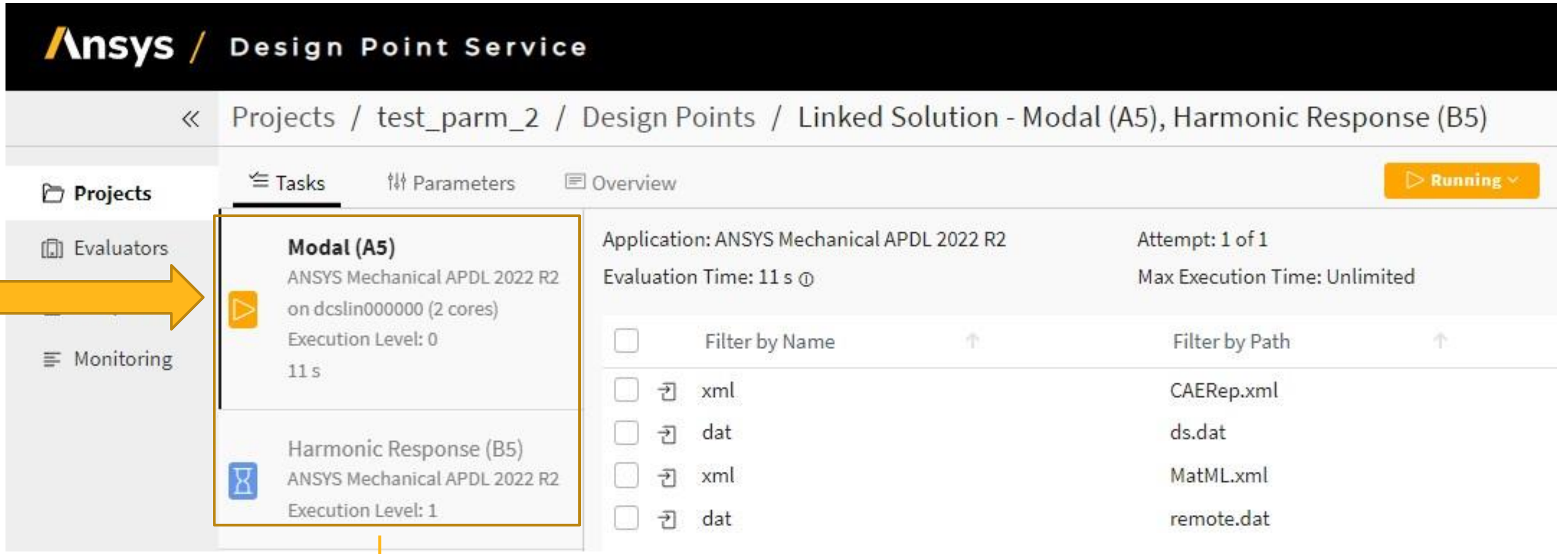
Stage 2					
Nodal Diameters					
HI	HI	N-HI	N+HI	2N-HI	2N+HI
0	0	12	12	24	24
1	1	11	13	23	25
2	2	10	14	22	26
3	3	9	15	21	27
4	4	8	16	20	28
5	5	7	17	19	29
6	6	6	18	18	30



Submit linked analysis job without waiting for completion

- This feature enables the submission of linked analyses on DCS without needing to wait for the upstream system's completion, thus allowing users to close Mechanical and conserve license for other purposes.





The linked jobs are submitted one after the other without waiting

The linked jobs are queued inside DCS

Limitations

- The synchronous submission process is supported for most cases except if an analysis includes any of the following features:
  - Spot Welds
  - Bolt Pretension
  - Weak Springs
  - EM Transducer loading condition
  - Command (APDL) objects
  - Contact Splitting
  - Linking though imported loads

These features may require creation of additional elements or nodes during the solution process or the results to be downloaded and hence, Mechanical would need to wait for the completion of the upstream job on DCS.

New Feature:

Exposure of Hybrid Parallel option to run solutions in Mechanical

Value Provided:

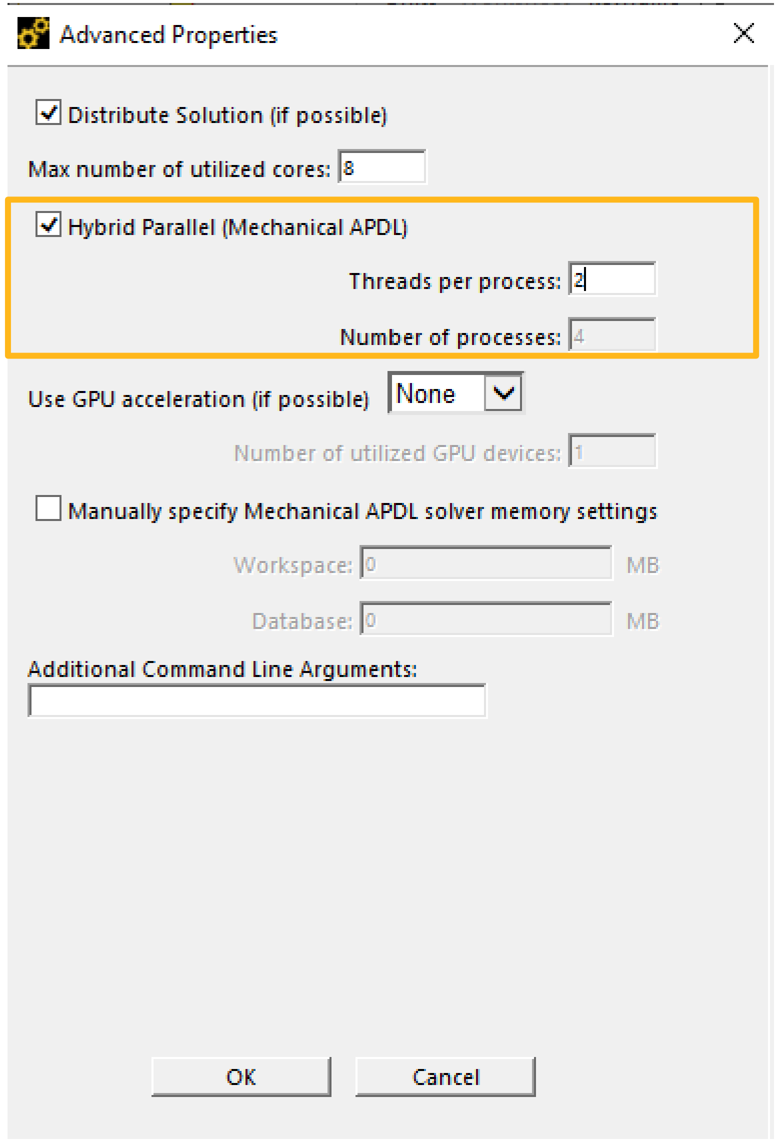
- Reduced memory usage
- Effective use of hardware resource
- Improved scalability of large models with high load balance ratios

Why Hybrid Parallel?

- Reduced Memory usage : Hybrid parallel reduces memory usage by using less MPI processes per compute node compared to distributed memory parallel (DMP)
- Effective use of hardware resource: To use fixed amount of memory with DMP, the cluster must use a fewer number of processes per compute node to increase memory and solution efficiency. Hybrid parallel can be used to address this issue and the cluster resource will be utilized in its full capacity
- Improved scalability of large models with high load balance ratios: Hybrid parallel can improve efficiency and scalability of large models where the load balance ratio is not optimum and such examples may include the cases where the contact pairs cannot be split, or re-mesh happens in certain regions due to mesh adaptivity (Nonlinear Adaptive Region supported in Mechanical)

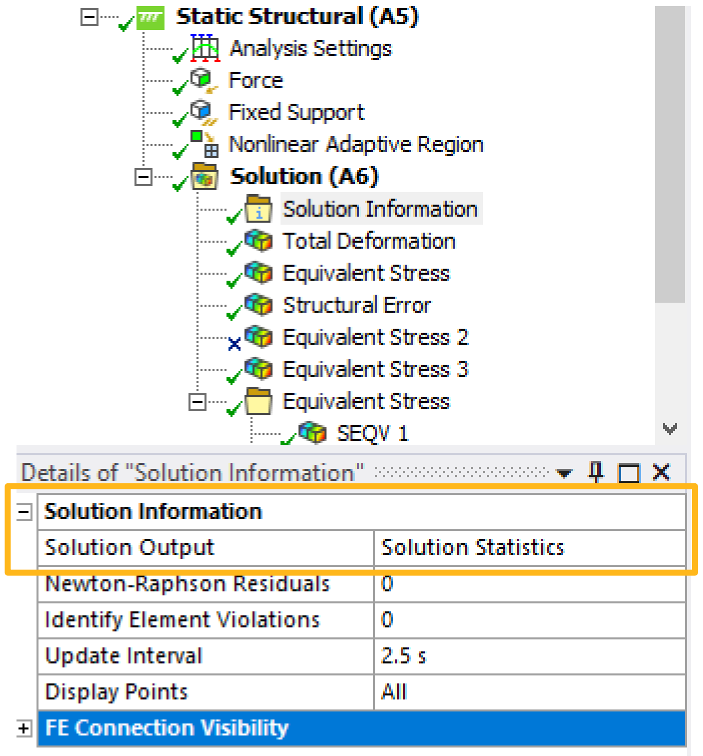
Hybrid Parallel exposure in Mechanical

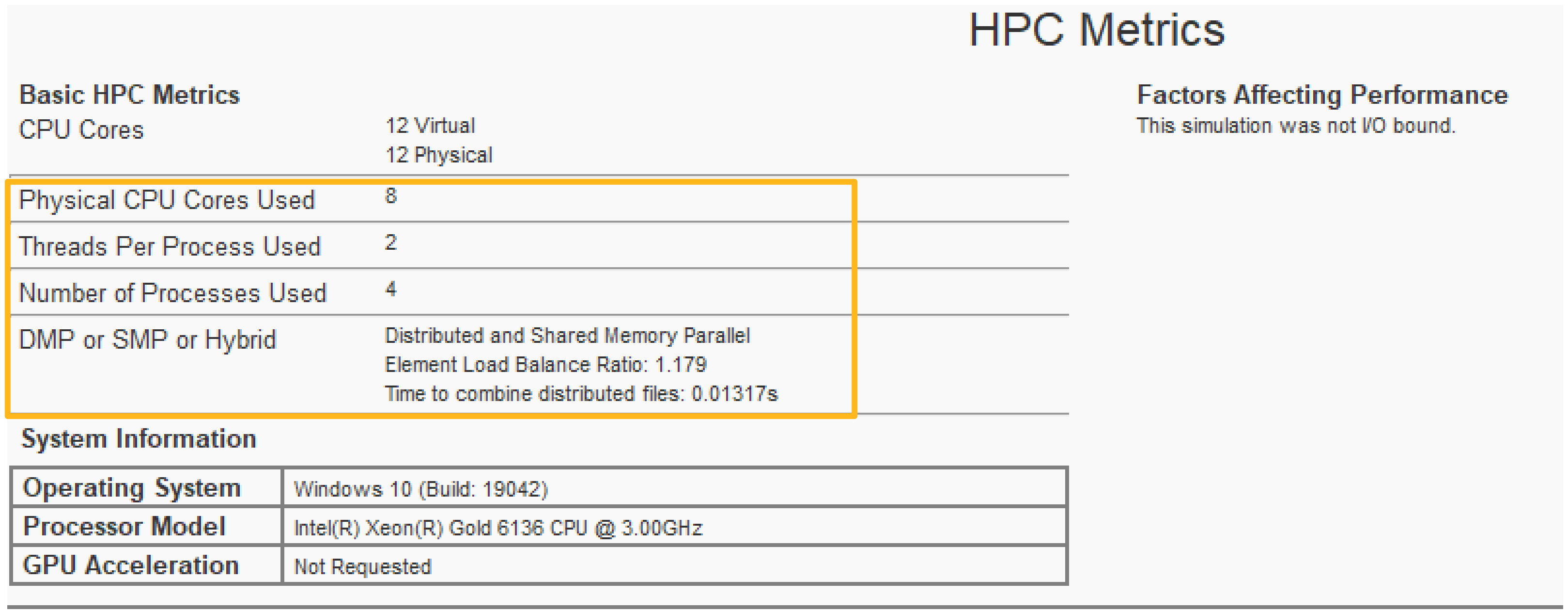
- Hybrid Parallel solution is supported inside Mechanical for MAPDL solution. To enable it, the user need to go to Solve Process Settings and Advanced Properties option.
- Once enabled, the user can specify the number of Threads per process and based on the cores and number of threads per process, the number of processes required will be displayed. The number of processes will be cores divided by number of threads per process



Hybrid Parallel exposure in Mechanical

- Once the solution is executed in Hybrid parallel mode, the solution statistics will display that the solution was done using the Distributed and Shared Memory parallel option and it will also show the cores, processes and threads per process used for the simulation





Hybrid Parallel limitation in Mechanical

- The solver can automatically switch to Hybrid parallel mode for some cases where the distributed solution is turned on and Hybrid parallel mode is not enabled inside Mechanical.
  This happens for cases where solver analyze that Hybrid parallel mode is efficient than Distributed parallel. In this mode, there is limitation that user cannot run the downstream linked analysis.
  Hence, for this rare scenarios, it is recommended to explicitly turn on the Hybrid parallel option in Mechanical and re-run the upstream analysis before proceeding for downstream analysis.
  For example, for pre-stress full harmonic analysis, if the upstream static analysis has automatically enabled the hybrid parallel mode, then the downstream harmonic analysis may not run successfully in Mechanical

New Feature:

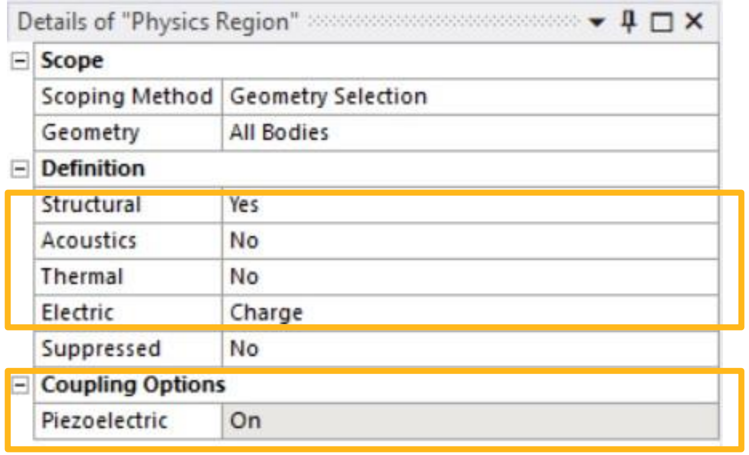
Coupled Field analysis exposure in Mechanical

Value Provided:

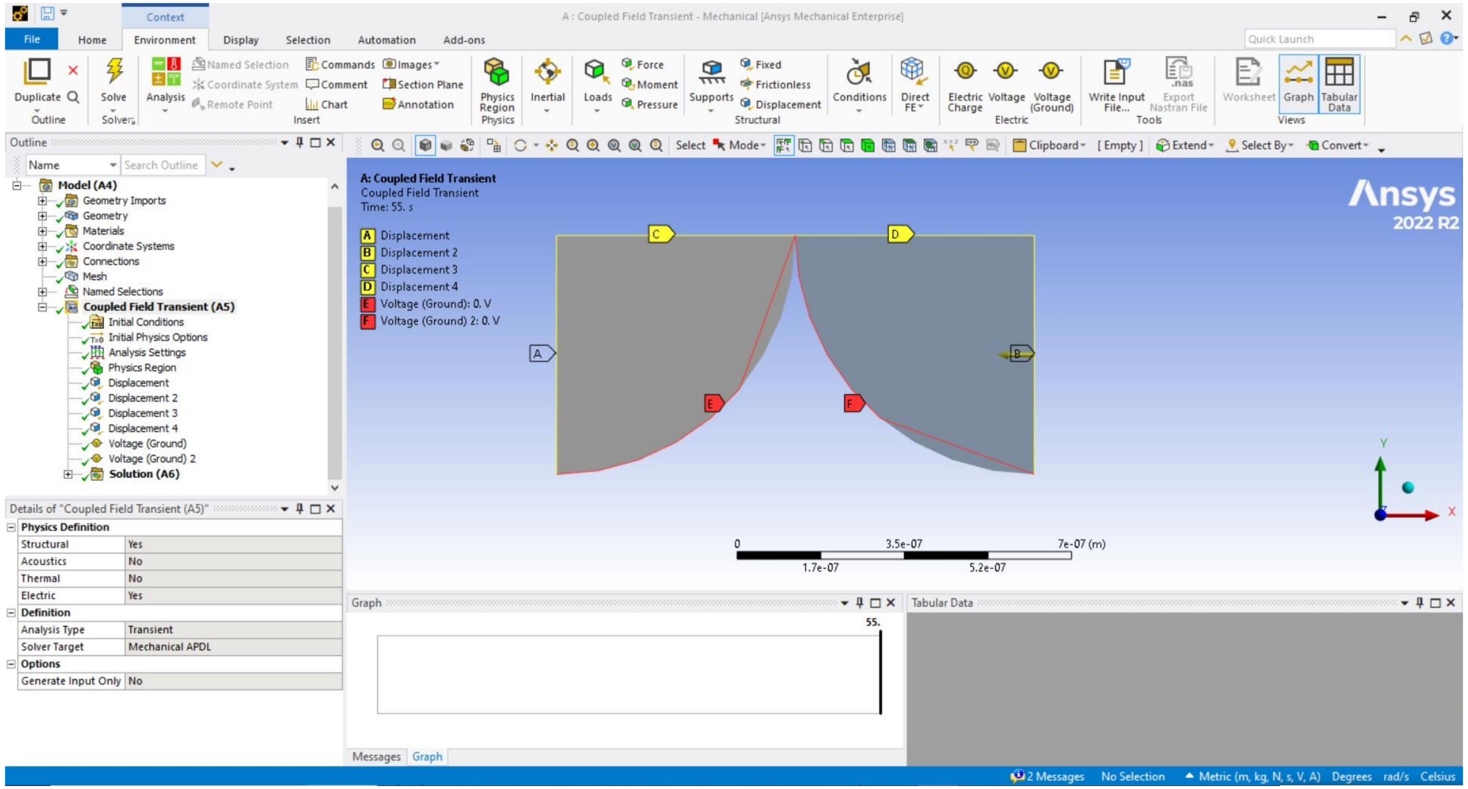
- Enables easier workflows for sensor design, MEMs devices and actuators
- Ability to model piezoelectric and acoustic degrees of freedom
- Includes a library of commonly used piezoelectric materials used in sensor design

Piezoelectric coupling in Coupled Field Transient

- Structural and Electric (Charge based) physics interaction through Piezoelectric coupling is supported for Coupled Field Transient Analysis. The physics region can be inserted to apply this physics interaction. Structural and Electric Boundary conditions are available for the analysis

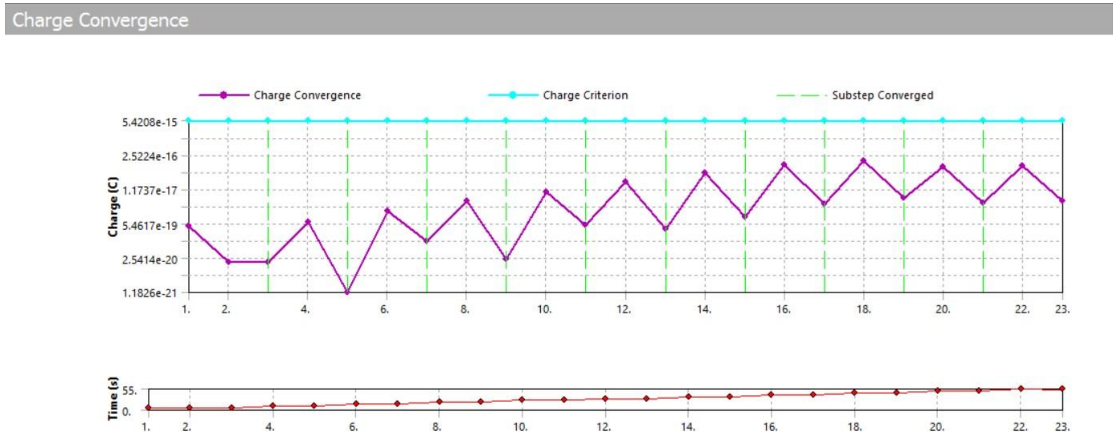

Details of "Physics Region"

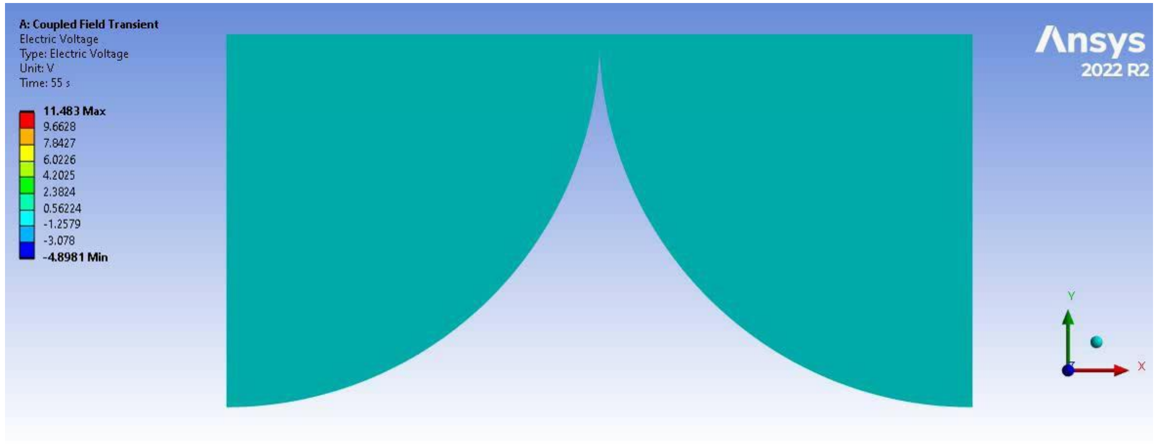
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Structural	Yes
Acoustics	No
Thermal	No
Electric	Charge
Suppressed	No
Coupling Options	
Piezoelectric	On

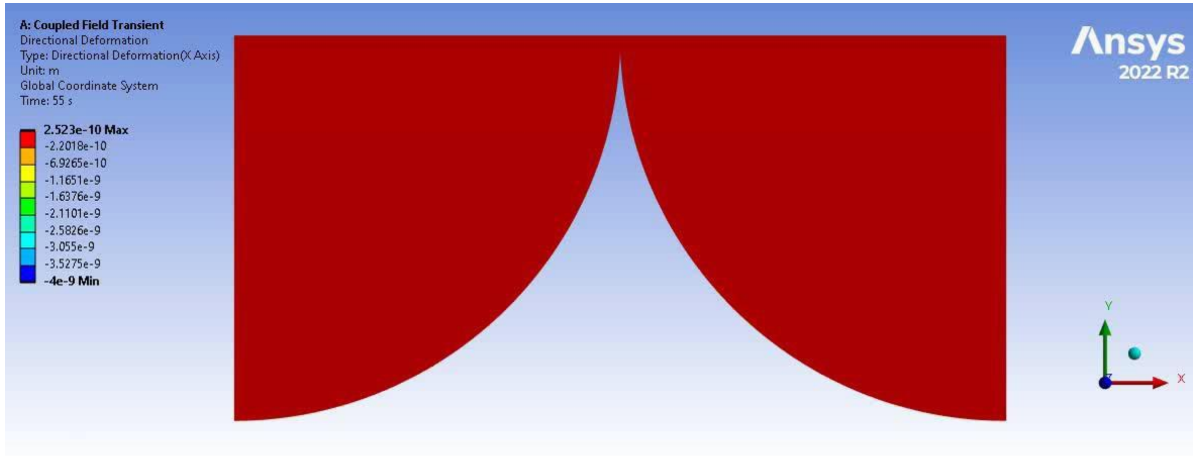


Piezoelectric coupling in Coupled Field Transient

- Structural and Electric results and probes are supported
- The user can also plot the Charge convergence under Solution output
- As shown in the example, Voltage is being generated as a results of applied mechanical stress due to Direct piezoelectric effect







Piezoelectric-Acoustics coupling in Coupled Field Transient

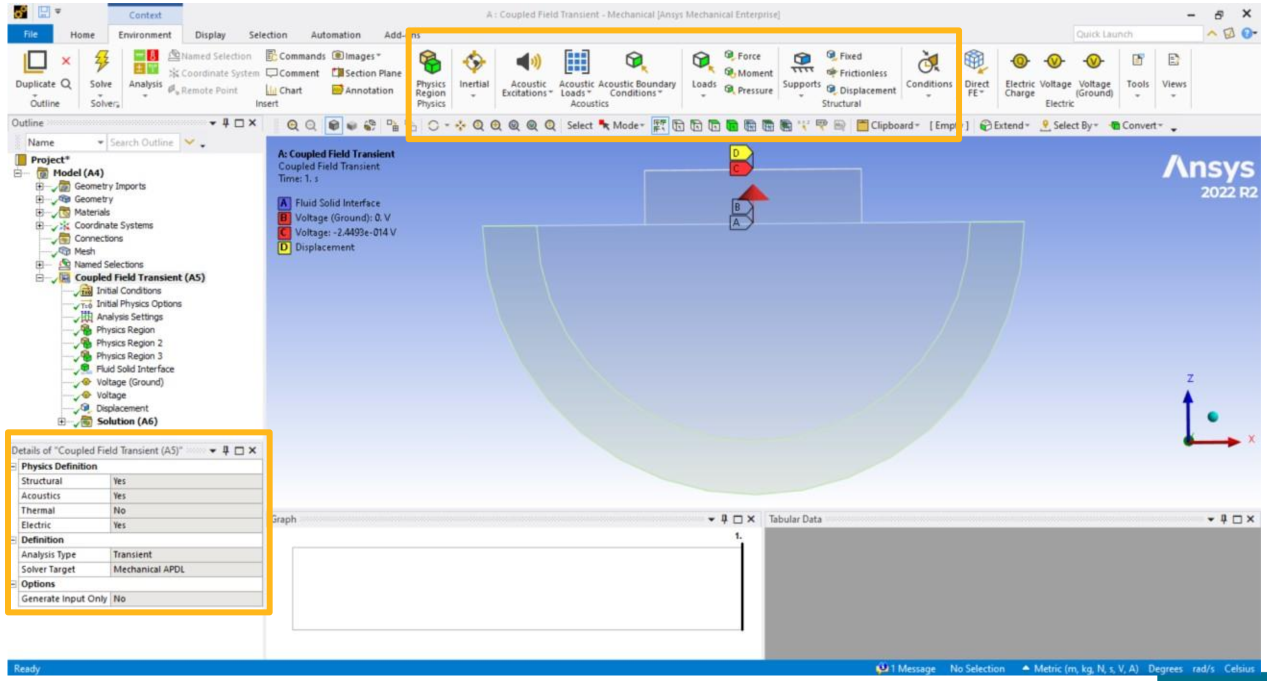
- Piezoelectric coupling along with interaction of Acoustics physics using the Fluid Solid interface can be performed in Coupled Field Transient analysis
- Physics region object is used to define bodies with Acoustics, Structural or Structural-Electric (Charge based) physics. The boundary conditions associated to Acoustics, Structural and Electric physics can be defined in the analysis

Details of "Physics Region"

Scope	
Scoping Method	Named Selection
Named Selection	piezo_body
Definition	
Structural	Yes
Acoustics	No
Thermal	No
Electric	Charge
Suppressed	No
Coupling Options	
Piezoelectric	On

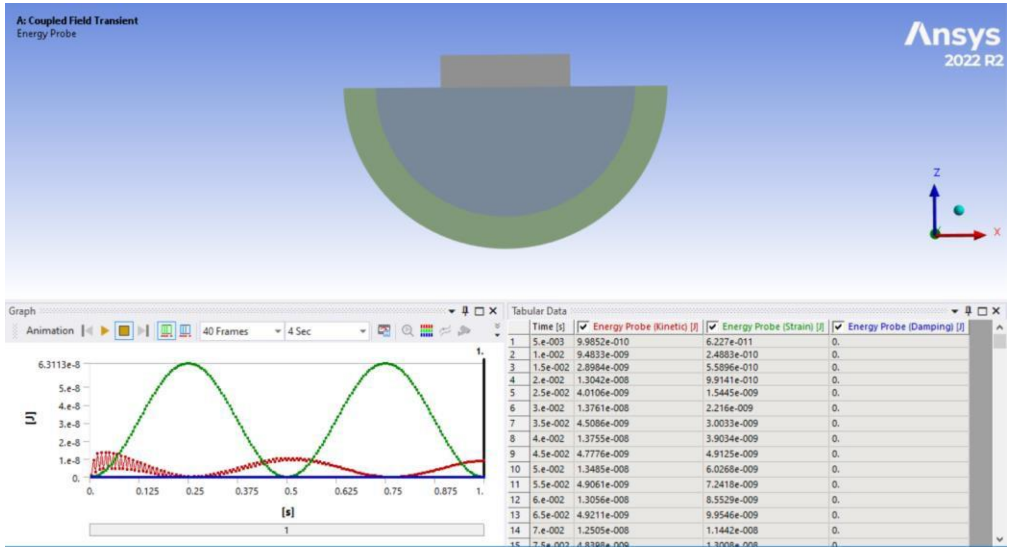
Details of "Physics Region 2"

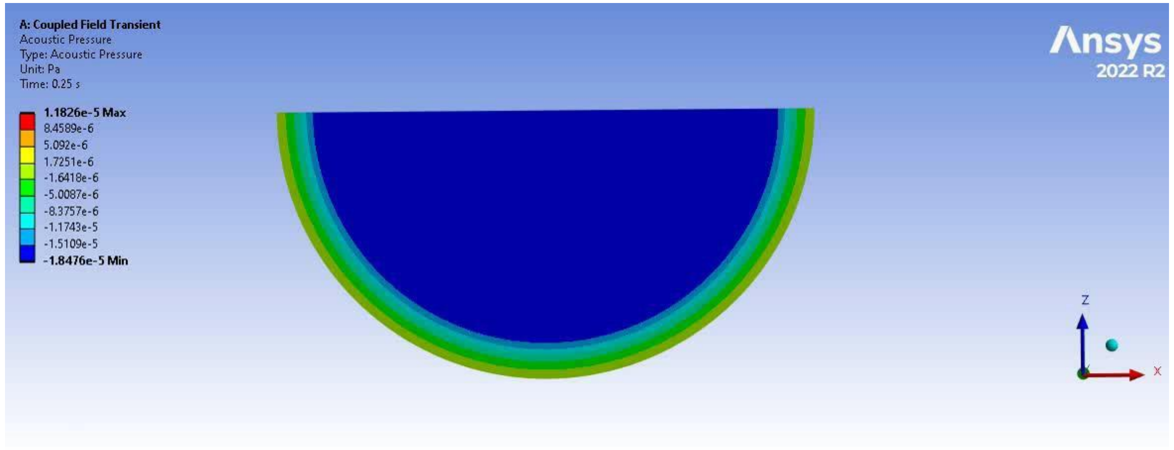
Scope	
Scoping Method	Named Selection
Named Selection	fluid_body
Definition	
Structural	No
Acoustics	Yes
Thermal	No
Electric	No
Suppressed	No
Acoustic Domain Definition	
Artificially Matched Layers	Off
Advanced Settings	
Reference Pressure	2.e-005 Pa
Reference Static Pressure	1.0133e+005 Pa
Fluid Behavior	Compressible



Piezoelectric-Acoustics coupling in Coupled Field Transient

- Acoustics, Structural and Electric results and probes are available. Plots of Charge convergence, Heat convergence, Force convergence can be seen on Solution output
- For example, the Acoustic pressure distribution can be seen on Acoustic body. Energy probe is also shown.





CAD-IT CONSULTANTS

Sample piezoelectric material properties in Engineering Data

- A library of common piezoelectric materials is provided in Engineering Data Sources
- These can be readily added to the project for the analysis and updated with material supplier provided data

Engineering Data Sources

	A	B	C	D
1	Data Source		Location	Description
4	General Materials			General use material samples for use in various analyses.
5	Additive Manufacturing Materials			Additive manufacturing material samples for use in additive manufacturing analyses.
6	Geomechanical Materials			
7	Composite Materials			
8	General Non-linear Materials			
9	Explicit Materials			
10	Hyperelastic Materials			
11	Magnetic B-H Curves			
12	Thermal Materials			
13	Fluid Materials			
14	Piezoelectric Materials			

Outline of Piezoelectric Materials

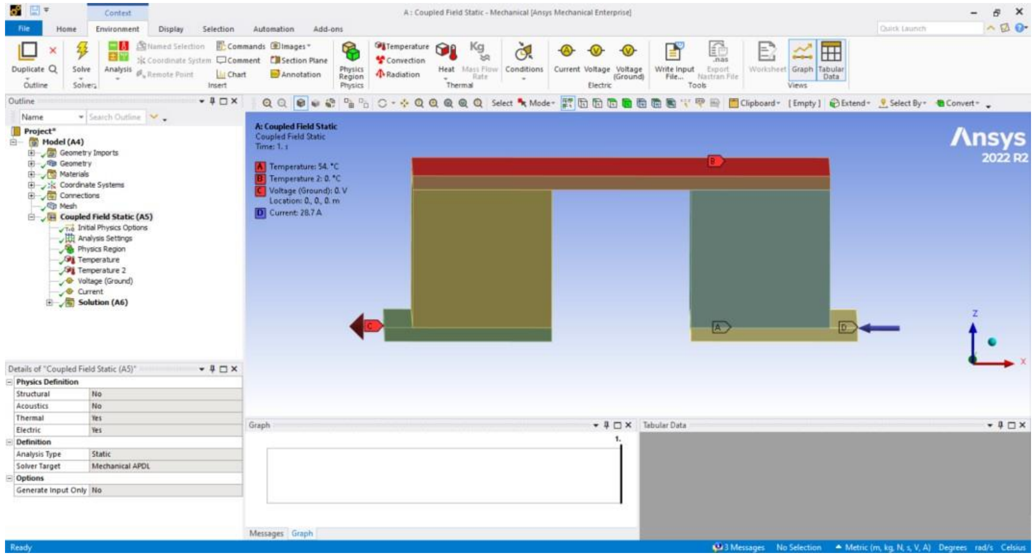
	A	B	C	D	E
1	Contents of Piezoelectric Materials	Add		Source	Description
2	Material				
3	Barium Titanate Crystal ($BaTiO_3$)			Piezoelectric_Materials.xml	Berlincourt D., Jaffe H. (1958) Elastic and piezoelectric coefficients of single-crystal barium titanate. Phys. Rev., vol 111. pp. 143-148
4	Lithium Niobate ($LiNbO_3$)			Piezoelectric_Materials.xml	Tiersten H.F "Linear Piezoelectric Plate Vibrations," Plenum Press, New York (1969)
5	Lithium Tantalate ($LiTaO_3$)			Piezoelectric_Materials.xml	Tiersten H.F "Linear Piezoelectric Plate Vibrations," Plenum Press, New York (1969)
6	Lithium Teraborate ($Li_2B_4O_7$)			Piezoelectric_Materials.xml	V. Petrov,R. Komatsu and T. Sugawara. Temperature tuned noncritical phase-matchingin $Li_2B_4O_7$ for generation of cw laser radiation at 244 nm. Electron.Lett. 35 (1999) 721-2
7	PZT-26			Piezoelectric_Materials.xml	C. Bricault, C. Pezerat, M. Collet, A. Pyskir, P. Perrard, et al. . Multimodal reduction of acoustic radiation of thin plates by using single piezoelectric patch with a negative capacitance shunt. Applied Acoustics, Elsevier, 2019,145,pp.320-327
8	PZT-4			Piezoelectric_Materials.xml	J Yang, Analysis of piezoelectric Devices (Appendix II), World Scientific Publication, Hackensack N.J. ISBN 9789812568618
9	PZT-5H			Piezoelectric_Materials.xml	H. A. Kunkel, S. Locke and B. Pikeroen, "Finite-element analysis of vibrational modes in piezoelectric ceramic disks," in IEEE Transactions on Ultrasonics, Ferroelectrics, and Frequency Control, vol. 37, no. 4, pp. 316-328, July 1990, doi: 10.1109/58.56492.
10	PZT-8			Piezoelectric_Materials.xml	J Yang, Analysis of piezoelectric Devices (Appendix II), World Scientific Publication, Hackensack N.J. ISBN 9789812568618
11	Quartz (alpha)			Piezoelectric_Materials.xml	Basic Material Quartz and Related Innovations. In: Piezoelectricity. Springer Series in Materials Science, vol 114. Springer, Berlin, Heidelberg

ThermoElectric coupling in Coupled Field Static analysis

- A Thermal Electric coupling can be performed in Coupled Field Static analysis.
- The analysis can account for thermoelectric effects, such as Joule heat, Seebeck, Peltier and Thomson effect. The coupling is included either at Load vector level or Matrix level
- The physics region can be specified with physics as Thermal only, Electric Conduction only or both. Thermal and Electric boundary conditions can be defined

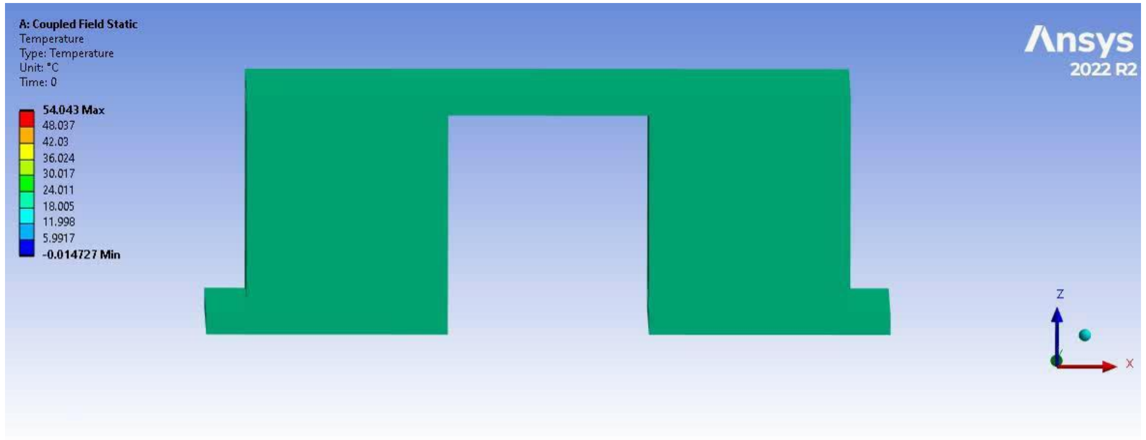
Details of "Physics Region"

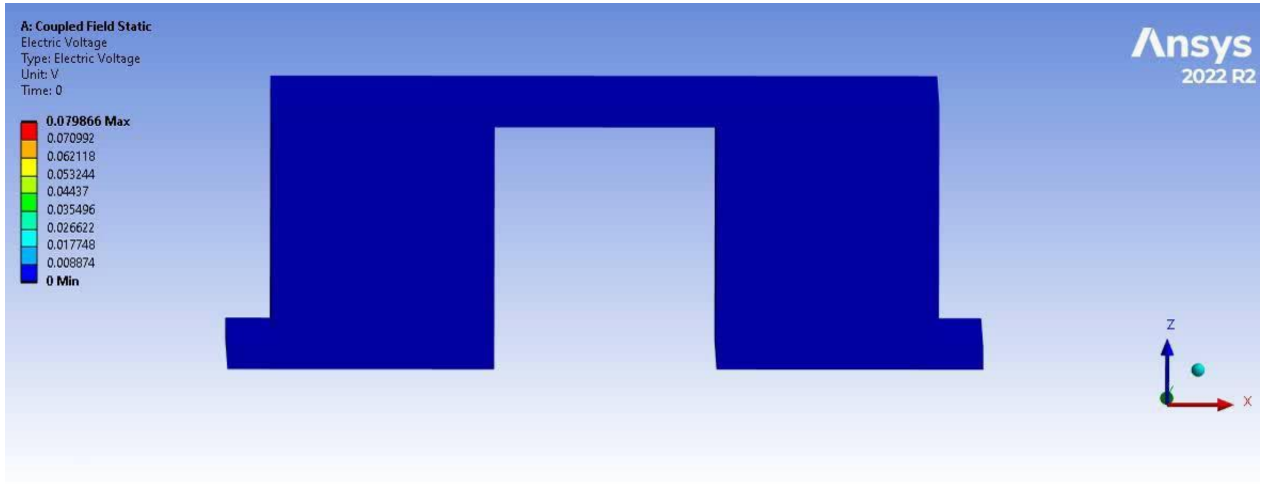
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Structural	No
Acoustics	No
Thermal	Yes
Electric	Conduction
Suppressed	No



ThermoElectric coupling in Coupled Field Static analysis

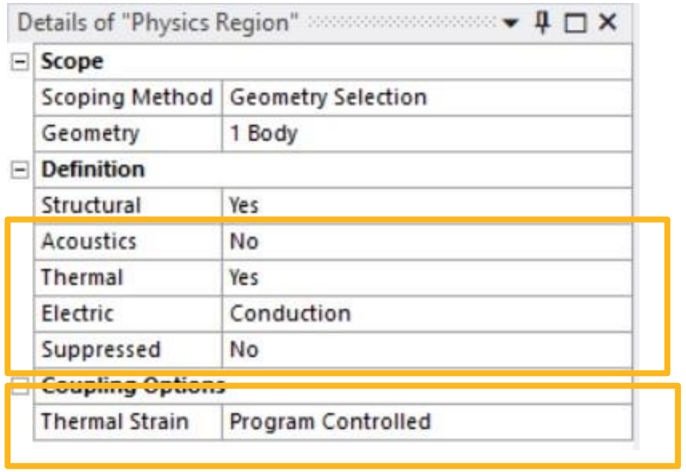
- Thermal and Electric results and probes are available
- Heat Convergence and Current convergence plots can be seen on Solution Output.
- For example, the temperature and voltage distribution of a Thermoelectric cooler is shown below.



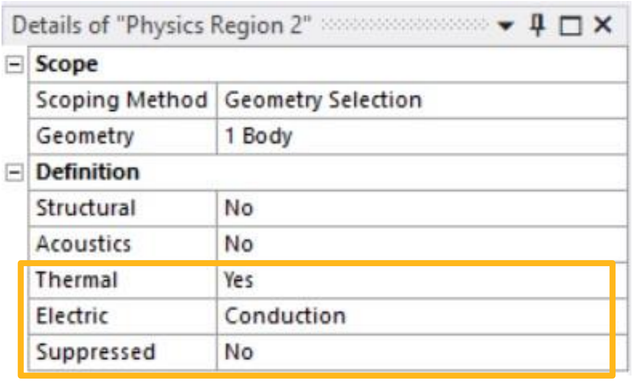


Structural-ThermoElectric coupling in Coupled Field Static analysis

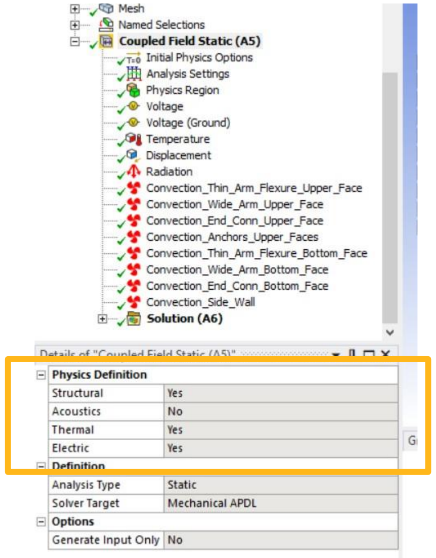
- Coupling of Structural and Thermoelectric physics can be performed in Coupled Field Static analysis. The coupling effects of both Structural-Thermal using thermal strain and Thermal-Electric conduction based of Joule heating, Seebeck, Peltier and Thomson effects will be considered when Structural-Thermal-Electric Conduction based physics is specified using Physics region.
- Structural, Thermal and Electric boundary conditions can be defined in the analysis


Details of "Physics Region"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Structural	Yes
Acoustics	No
Thermal	Yes
Electric	Conduction
Suppressed	No
Coupling Options	
Thermal Strain	Program Controlled


Details of "Physics Region 2"

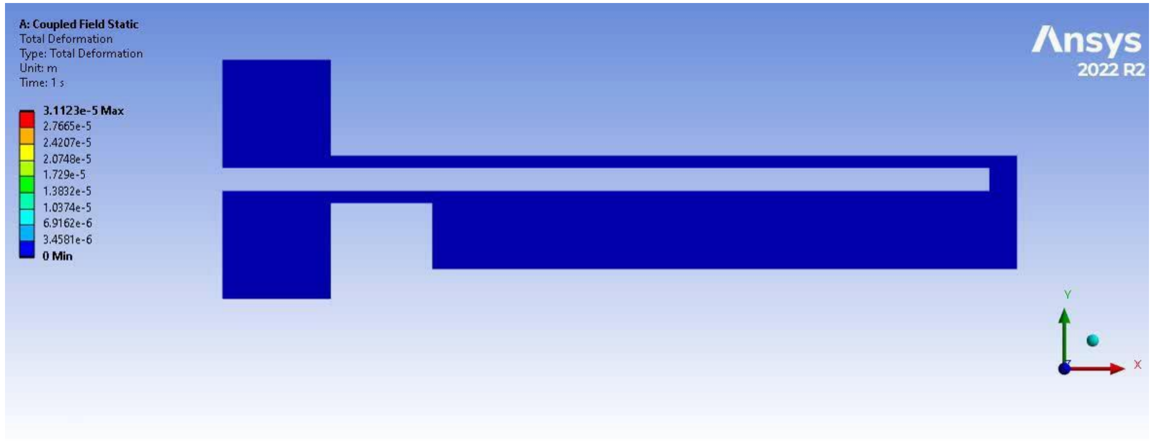
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Structural	No
Acoustics	No
Thermal	Yes
Electric	Conduction
Suppressed	No

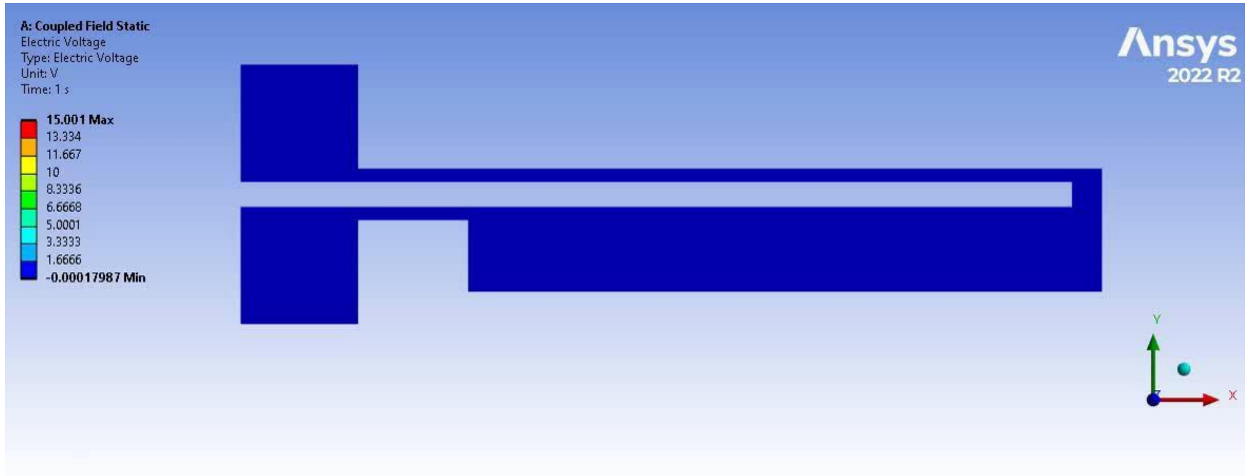


Physics Definition	
Structural	Yes
Acoustics	No
Thermal	Yes
Electric	Yes
Definition	
Analysis Type	Static
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

Structural-ThermoElectric coupling in Coupled Field Static analysis

- Structural, Thermal and Electric results and probes are available. The user can plot Force Convergence, Heat Convergence and Current convergence plots from Solution output.
- The animation of the deformation and voltage of an Electro – Thermal Micro actuator is shown below

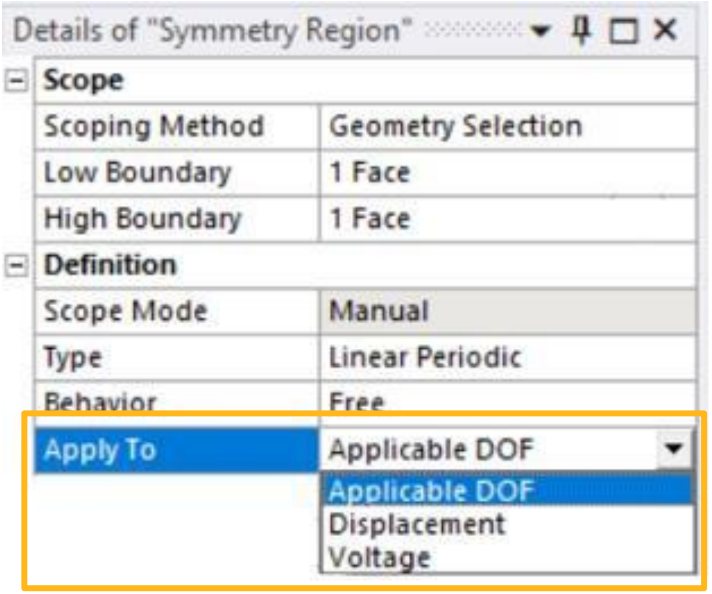




Linear Periodic symmetry for Voltage (Coupled Field Analysis)

- In previous releases linear periodic objects can be scoped to only structural or thermal-electric bodies. From 2022 R2, Linear periodic type symmetry region can be applied in Coupled Field Analysis with Displacement and Voltage degree of freedom .

- Use **Apply To** property to specify the Degree Of Freedom(DOF):
  - **Applicable DOF (Default)**
    - specifies Displacement and Voltage DOFs on the geometries of the Low Boundary and High Boundary.
  - **Displacement**
    - specifies Displacement DOF on the geometries of the Low Boundary and High Boundary.
  - **Voltage**
    - specifies Voltage DOF on the geometries of the Low Boundary and High Boundary.



Summary of Coupled Field systems

	Coupled Field Harmonic	Coupled Field Modal	Coupled Field Static	Coupled Field Transient
Structural-Thermal	[1]	[2]	Thermal strain Thermoplasticity Thermoviscoelasticity	Thermal strain Thermoelastic Damping Thermoplasticity Thermoviscoelasticity
Structural - Acoustics	Fluid Solid Interface	Fluid Solid Interface	Fluid Solid Interface	Fluid Solid Interface
Structural Electric (Charge)	Piezoelectric	Piezoelectric	Piezoelectric	Piezoelectric ☆
Structural Electric Acoustic	Piezoelectric Fluid Solid Interface	Piezoelectric Fluid Solid Interface	Piezoelectric Fluid Solid Interface	Piezoelectric Fluid Solid Interface ☆
Thermal Electric (Conduction)	[2]	[2]	Joule Heating Seebeck effect Peltier effect Thomson effect ☆	[1]
Structural Thermal Electric (Conduction)	[2]	[2]	Joule Heating Seebeck effect Peltier effect Thomson effect Thermal strain ☆	[1]

1 Not Supported in Mechanical
2 Not Applicable

☆ New in 2022R2

Transient piezoelectric and piezo-acoustic
- Piezoelectric haptic feedback devices
- Pulse based piezoelectric transducer design
- Acoustic imaging/NDE through piezoelectric transmitter/receiver
- Piezoelectric micro-speaker design
- Many others

Static structural-thermoelectric
- Joule heating in various applications
- Electric connectors, splice design
- Thermocouple design
- Thermoelectric cooler and generator design
- Thermoelectric Micro-actuators
- Many others

New Feature:

Hyper viscoelastic materials in Linear Perturbation full harmonic analysis

Value Provided:

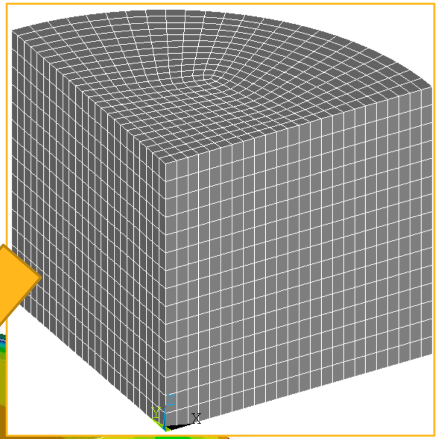
- Allows for the harmonic analysis of preloaded hyper viscoelastic materials.

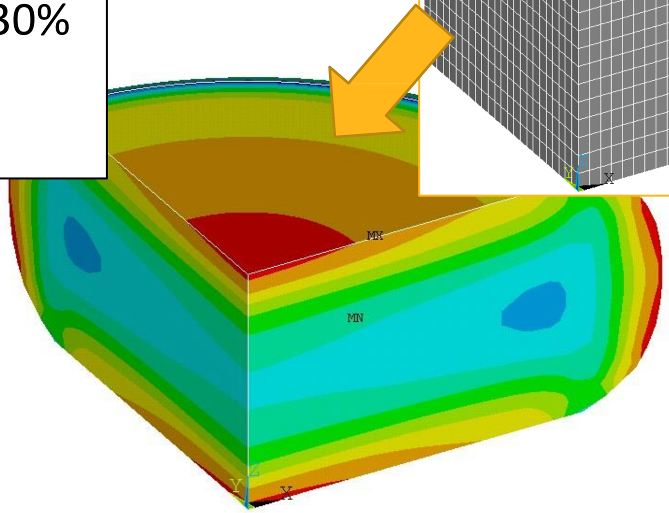
CAD-IT CONSULTANTS

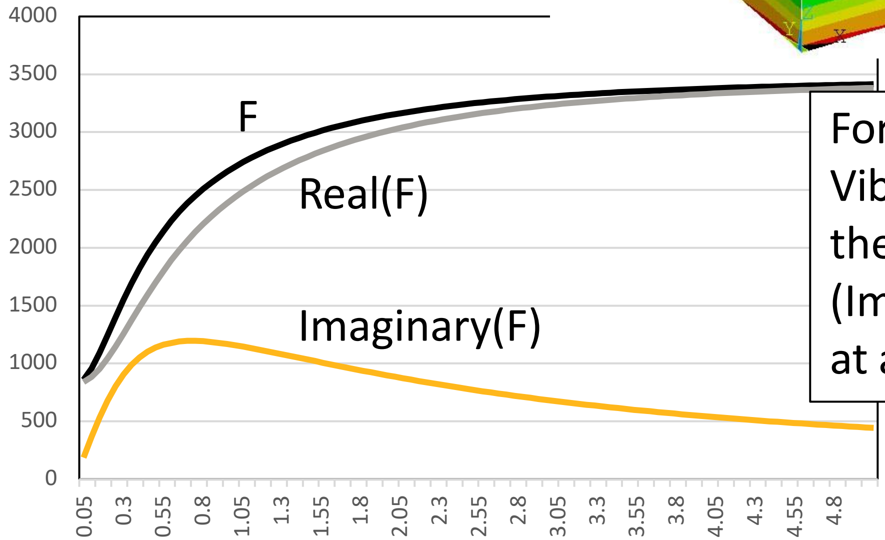
Hyper Viscoelasticity in Perturbed Harmonic Analyses

- Harmonic analysis is important in areas like vibration isolation and noise transmission
- For Noise, Vibration and Harshness (NVH) in vehicle design, the structures include plastics, foams, and other vibration absorbing materials
  - It is important to capture the deformed and prestressed state of the material during harmonic analysis
- This new capability will allow users to simulate the deformation in a static or transient step and then perform a follow-on perturbed harmonic analysis to determine the vibration characteristics of the structures that have hyper viscoelastic materials

Vibration Isolation Mount, Yeoh hyperelastic with Prony series viscoelasticity, compressed 30% and experiencing a fixed-amplitude small vibration







Force Transmission vs. Vibration Frequency shows the damping behavior (Imaginary part) is greatest at about 0.6 Hz

New Feature:

Support for AMD GPUs

Value Provided:

- Faster simulations (HPC value)

GPU Acceleration Enhancements

- Added support for Instinct GPUs from AMD
  - New generation (CDNA 2.0) cards significantly faster than previous generation (CDNA 1.0)
  - Uses HIP/ROCm 5.0 ❼ *requires AMD driver version 21.50 or newer*

AMD ROCm

CAD-IT CONSULTANTS

GPU Acceleration Enhancements

- Added support for Instinct GPUs from AMD
  - Focused on sparse direct solver
  - Targeting Linux only this release

Model	Release Date	Memory Capacity (GB)	Memory Bandwidth (GB/s)	Peak FP64 Compute (TFlops)	Peak FP32 Compute (TFlops)	Peak TF32 Compute (TFlops)
AMD Instinct MI100	November 2020	32	1223	11.5	23.1	N/A
NVIDIA A100	May 2020	40/80	1555/1935	9.7 (19.5[1])	19.5	156
AMD Instinct MI210	December 2021	64	1638	22.6	22.6	N/A
AMD Instinct MI250	November 2021	128	3277	45.3	45.3	N/A

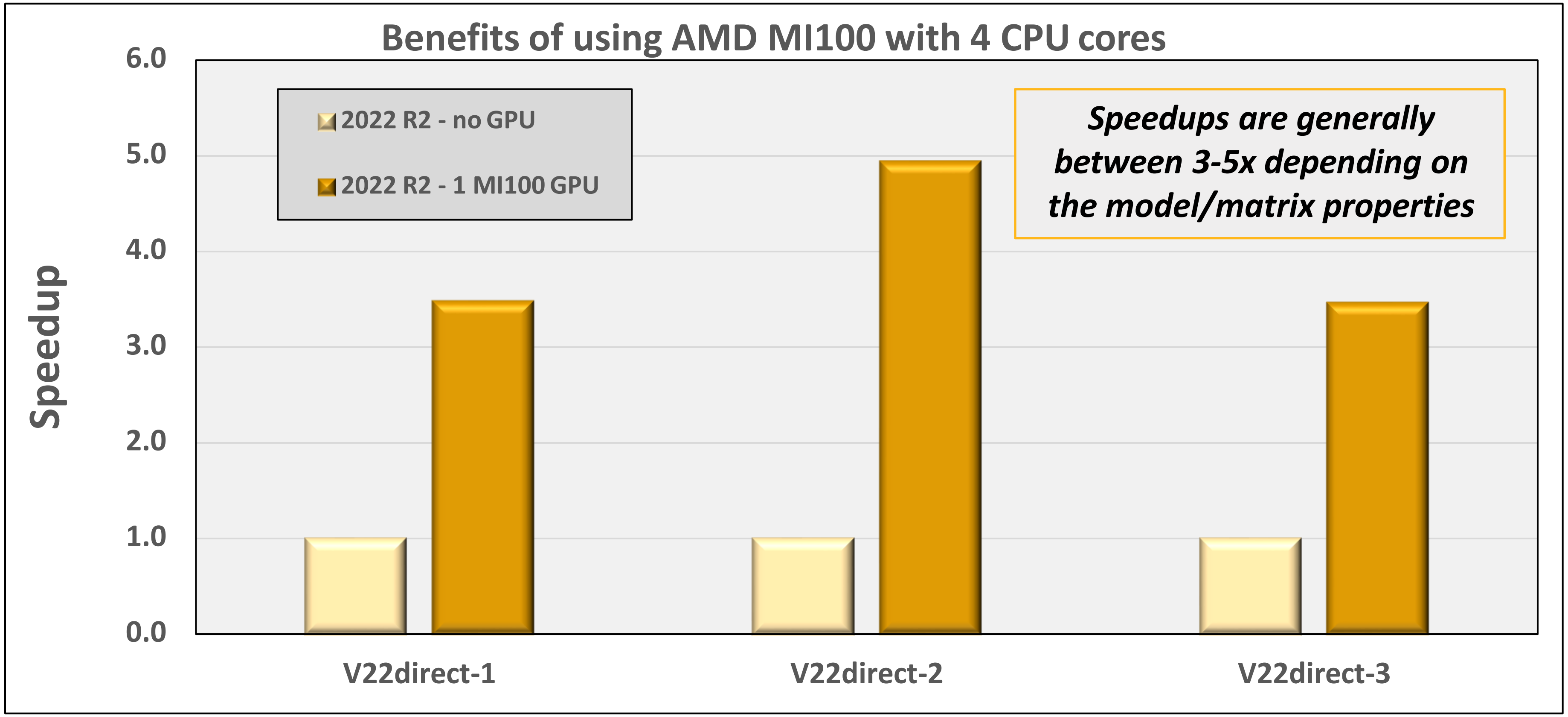
[1] Compute speed when using tensor cores
- Data provided by https://en.wikipedia.org

GPU Acceleration Enhancements

- Significantly faster performance for direct solver benchmarks

Measuring entire matrix factorization time (not just the calculations that are accelerated on the GPU)

•R22 Workstation Benchmark set run on Linux system with 2 AMD 7532 processors, 256 GB RAM, SSD, 1 AMD Instinct MI100 PCIe card, CentOS 8.4

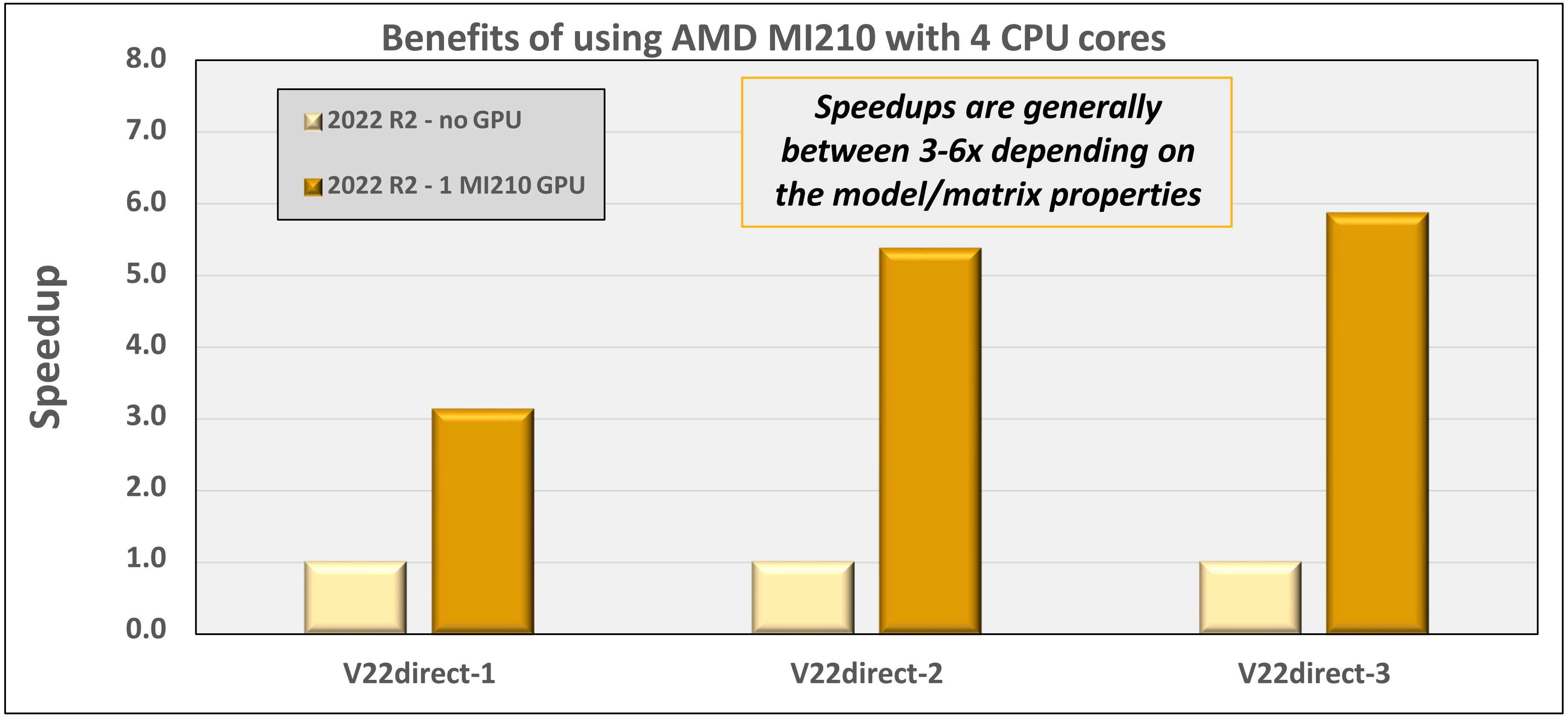


GPU Acceleration Enhancements

- Significantly faster performance for direct solver benchmarks

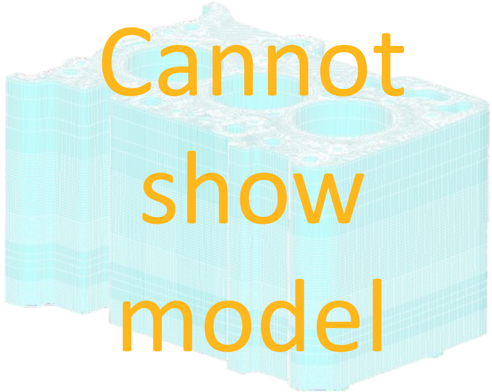
Measuring entire matrix factorization time (not just the calculations that are accelerated on the GPU)

•R22 Workstation Benchmark set run on Linux system with 2 AMD 7643 processors, 512 GB RAM, SSD, 1 AMD Instinct MI210 PCIe card, RHEL 8.5



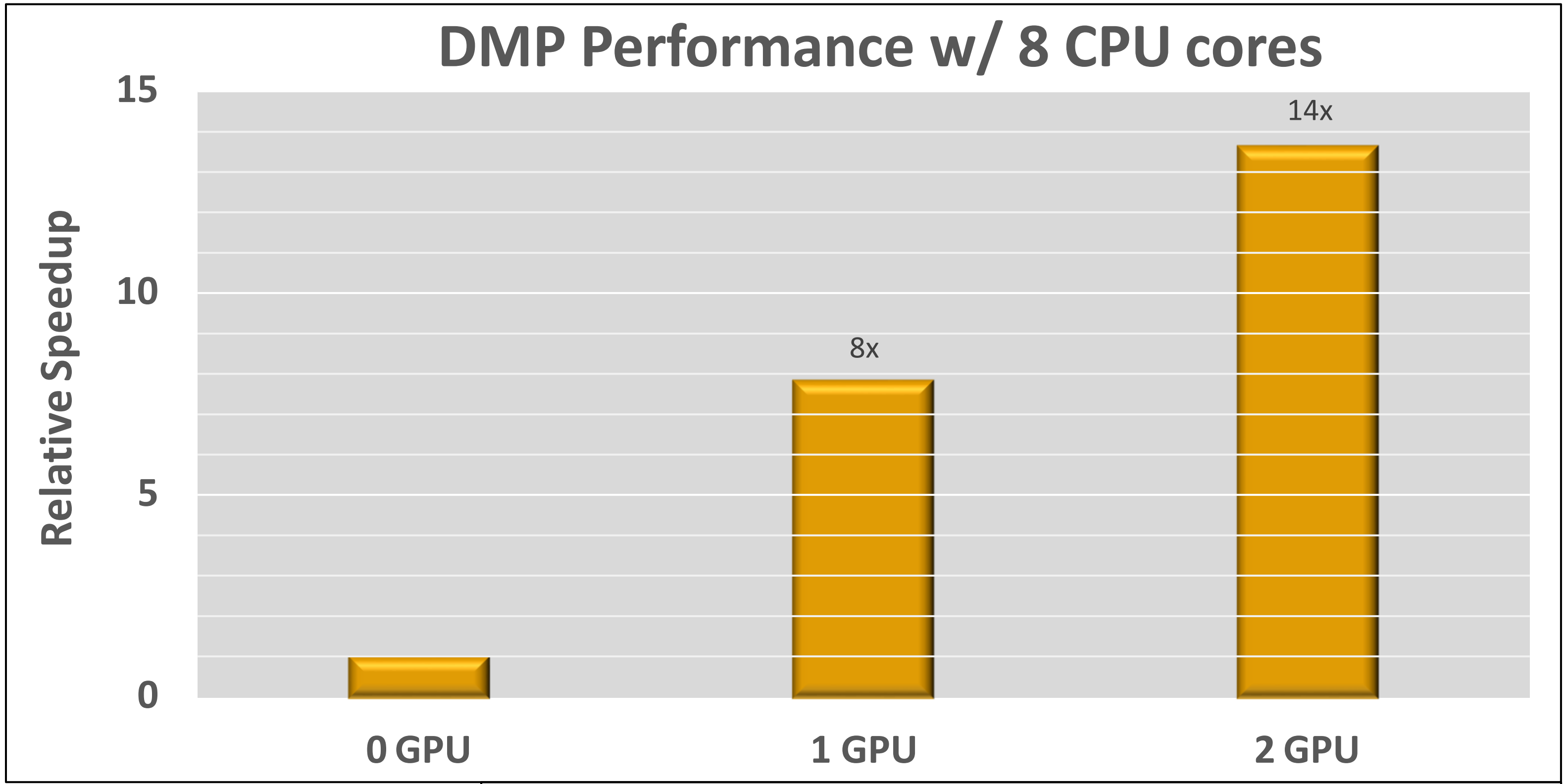
GPU Acceleration Enhancements

- Significantly faster performance (Engine block model)



Measuring just the matrix factorization algorithm that is mostly offloaded to the GPU

- 4.2 million DOF; sparse solver
- Nonlinear static analysis involving contact, plasticity and gasket elements



2 AMD EPYC 7643 processors, 512 GB RAM, AMD Instinct MI210 cards, RHEL 8.5

New Feature:

Default number of cores increased from 2 to 4

Value Provided:

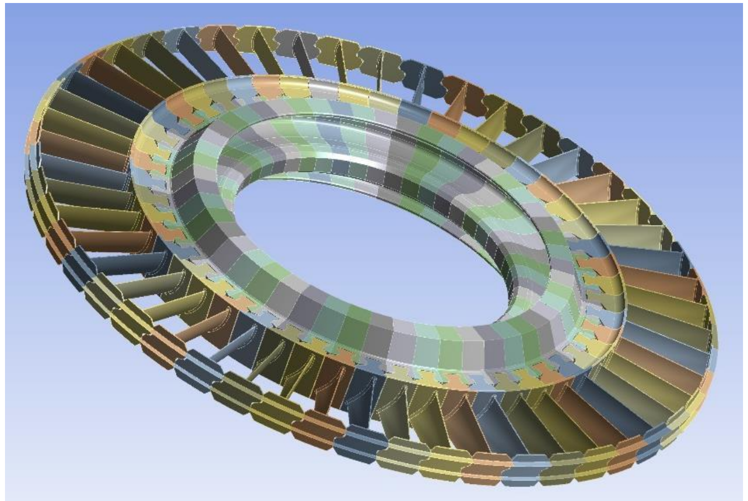
- Better HPC value
- When the product launches by default using only 2 CPU cores, it is underutilizing what they already paid for

Distributed Memory Parallel Enhancements

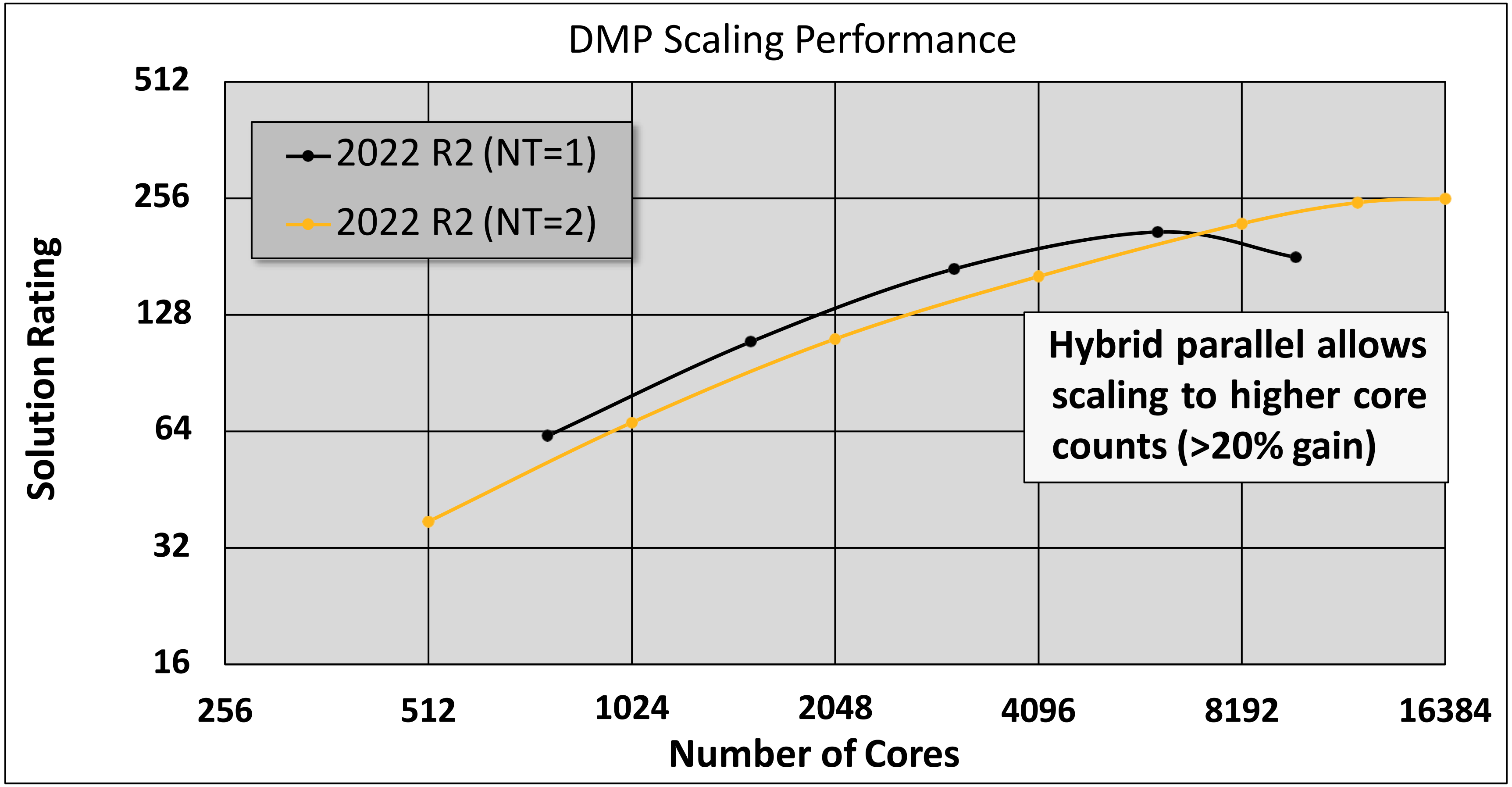
- Default changed from 2 to 4 CPU cores
- Replaced "Distributed ANSYS" naming ❼ "Distributed Memory Parallel"
- MPI library support
  - Upgraded to Intel MPI 2021 Update 6 on Windows and Linux
    - Improves performance, scalability and robustness
    - Linux clusters using (older) Mellanox Infiniband 4.x ❼ (older) Intel MPI 2018 is automatically chosen
  - Microsoft MPI v10.0 is unchanged at this release on Windows
  - Open MPI v4.0.5 is unchanged at this release on Linux

Distributed Memory Parallel Enhancements

- Improved scaling at higher core counts



- 50 MDOF; Sparse direct solver
- Nonlinear, static analysis involving large deflections, 20 equilibrium iterations, bonded contact pairs
- Linux cluster; each compute node contains 2 Intel Xeon Platinum 8260L processors (48 cores), 192GB RAM, SSD, CentOS 7.7, Mellanox HDR Infiniband



New Feature:

SMART enhancements: Automatic Crack Initiations and Remeshing Mesh Sizing Control

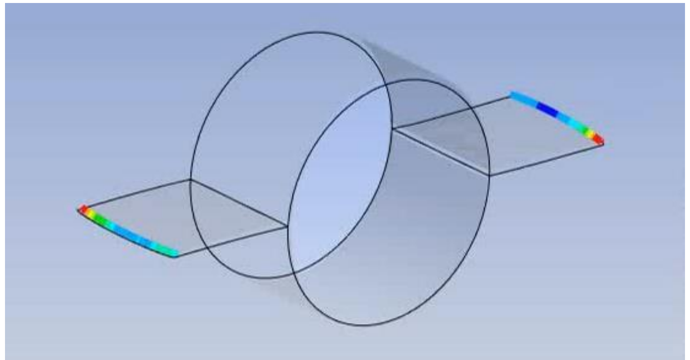
Value Provided:

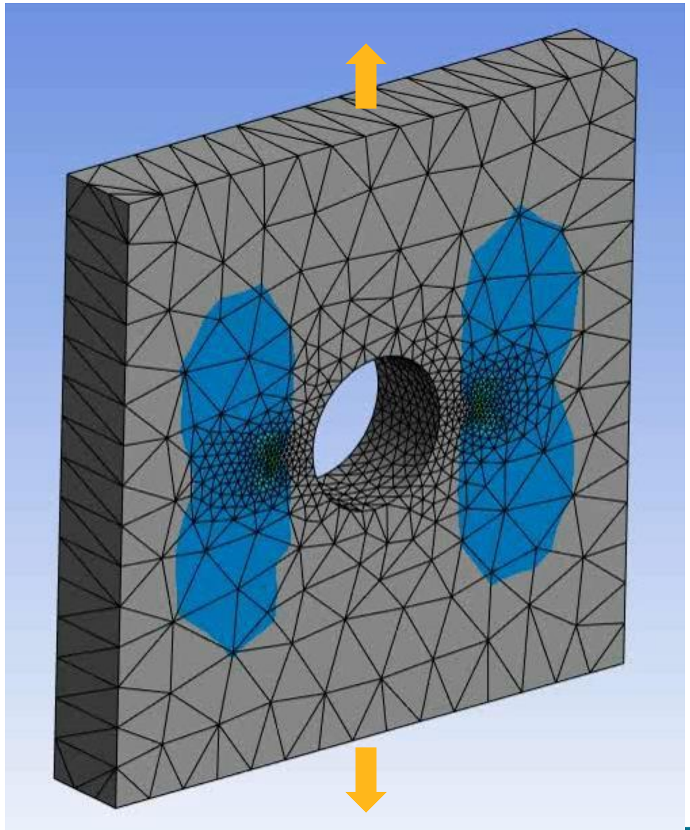
- Broaden simulation scopes
- Enable users to better control meshes
- Improve solution accuracy and robustness

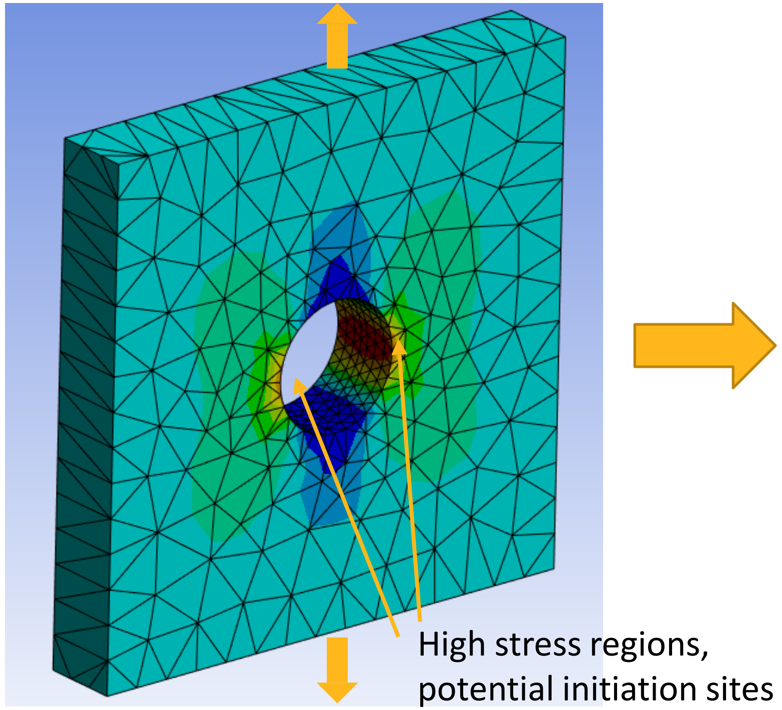
CAD-IT CONSULTANTS

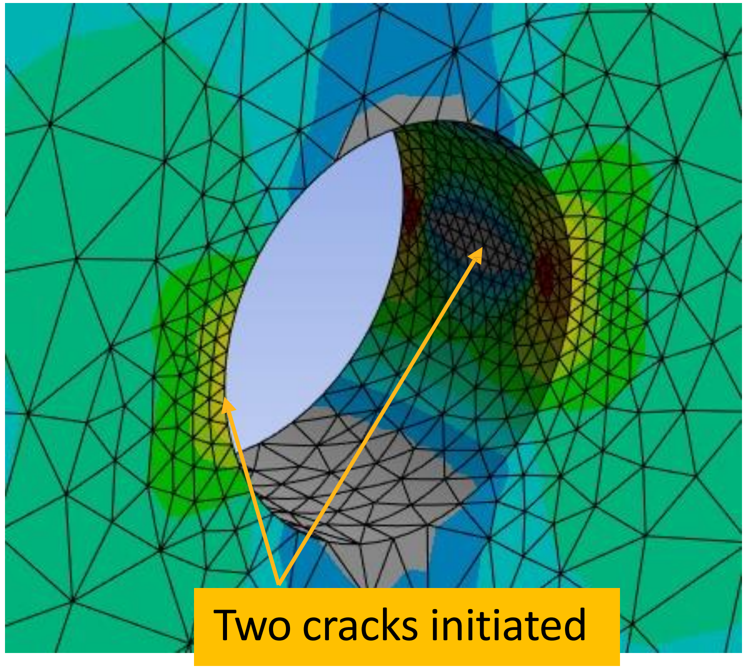
SMART Enhancement: Multi Cracks Initiation

- Multiple crack initiations
  - Multiple cracks can be initiated simultaneously or sequentially
- Maximum principal stress based initiation criteria
- Sizing control of initiated crack
- Support both fatigue and static crack growth



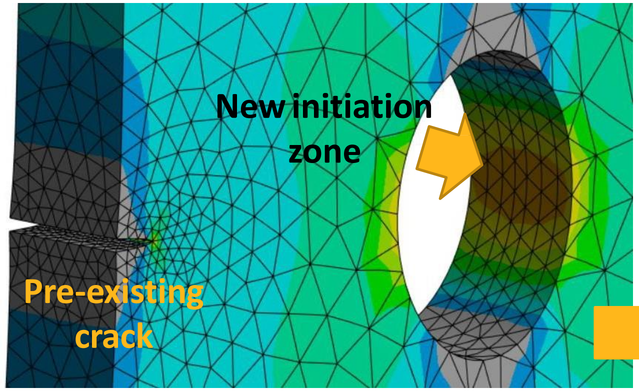




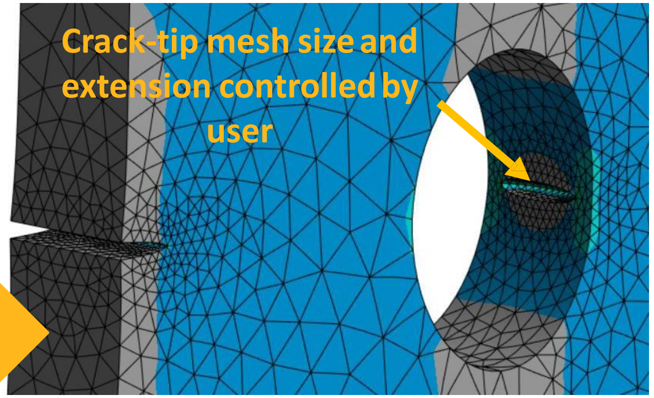


SMART Enhancement: Automatic Crack Initiation

- Automatic crack initiation with pre-existing crack(s)
  - Automatic crack initiation supports now structures with pre-existing crack(s)
- Sizing control of initiated crack
- Maximum principal stress-based initiation criteria
- Support both fatigue and static crack growth

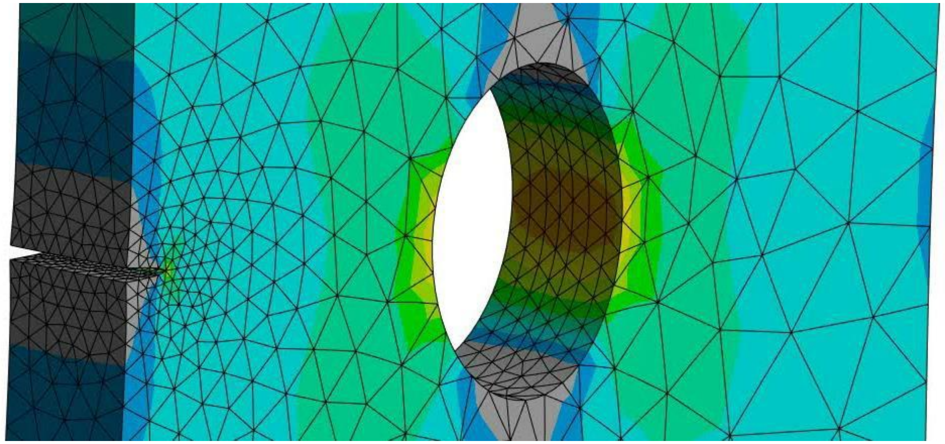


Stress solution of initial model



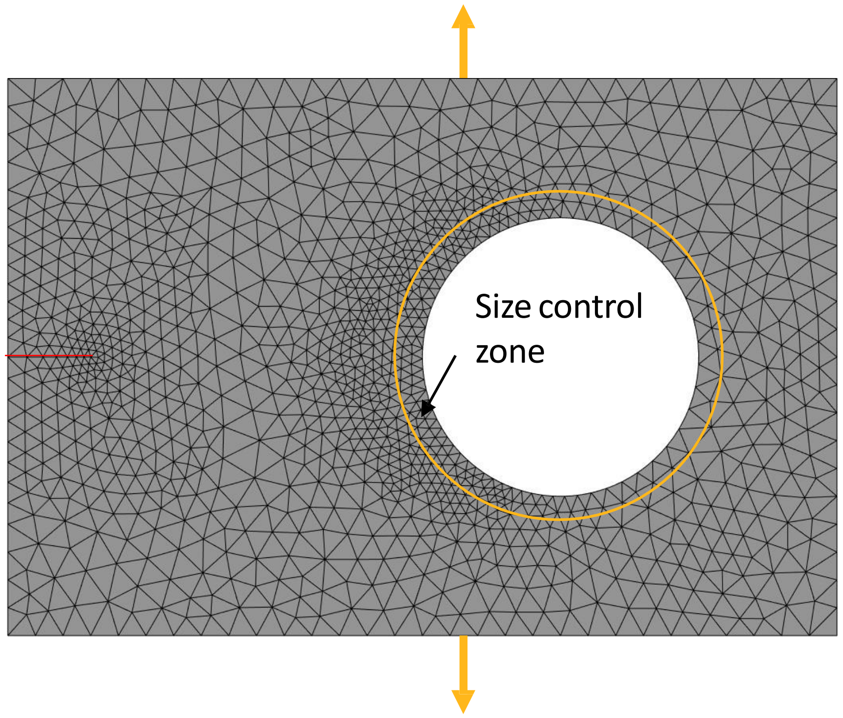
After crack-initiation

Support multiple pre-existing cracks and multiple crack initiations



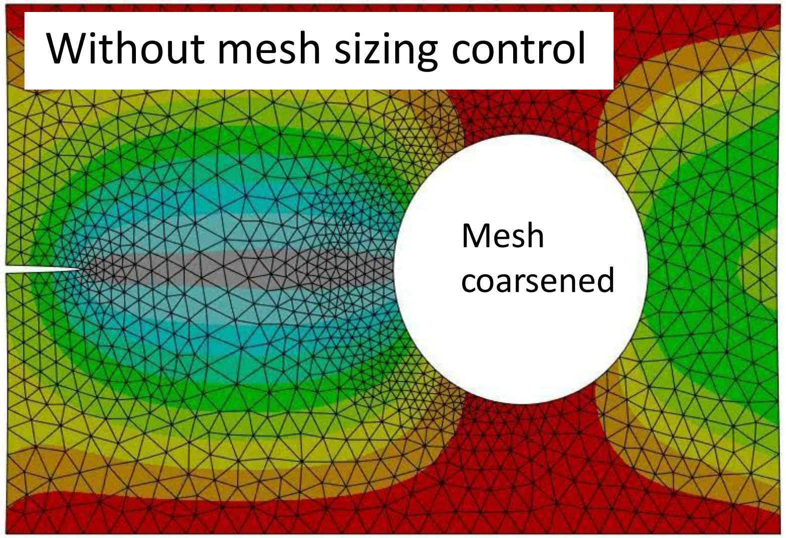
SMART Enhancement: Mesh Sizing Control

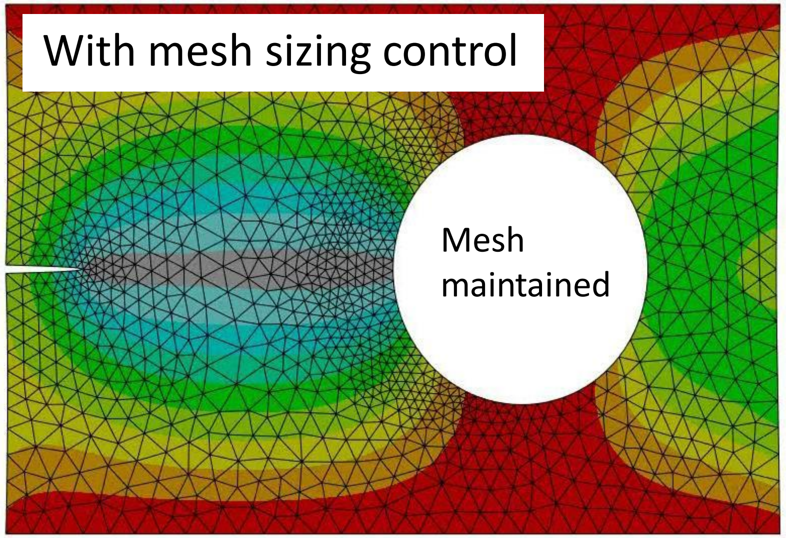
- SMART remeshing mesh sizing control
  - Sizing control with specific node or element components
  - Sizing control with specific locations (coordinates)



Mesh sizing control to maintain geometry

Fatigue crack growth
Mesh coarsening option: CONS

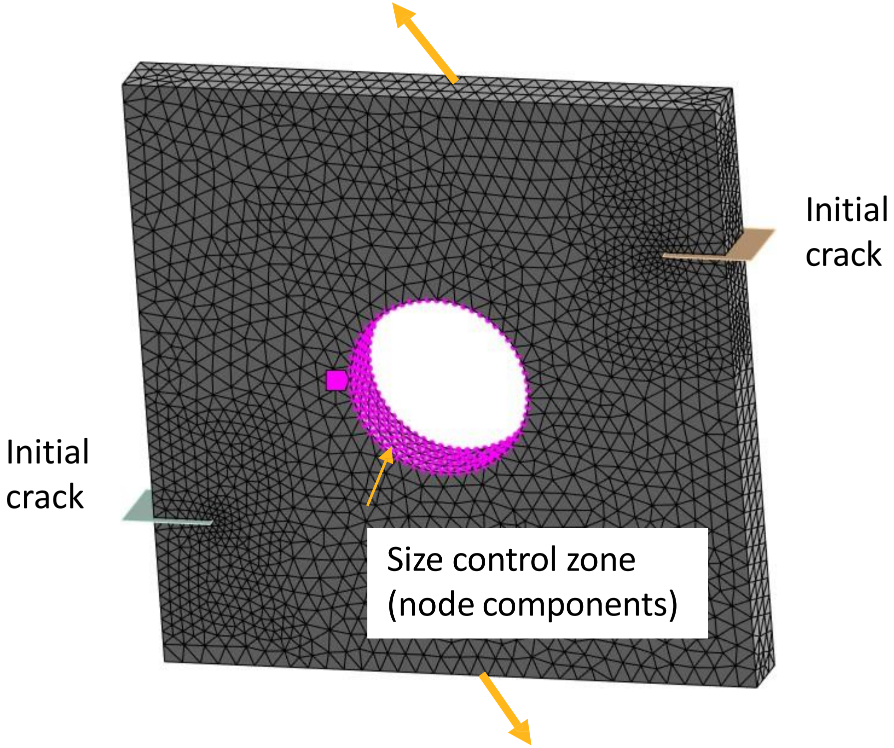




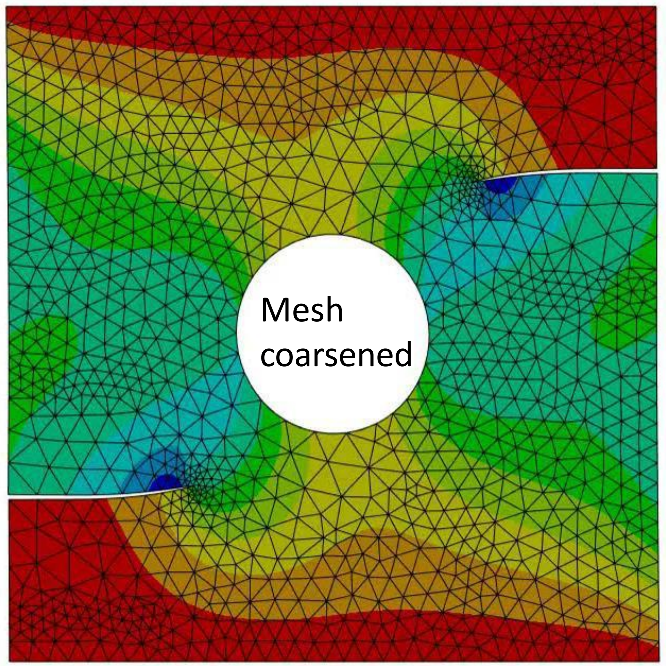
SMART Enhancement: Mesh Sizing Control

- Perforated plate with two edge cracks

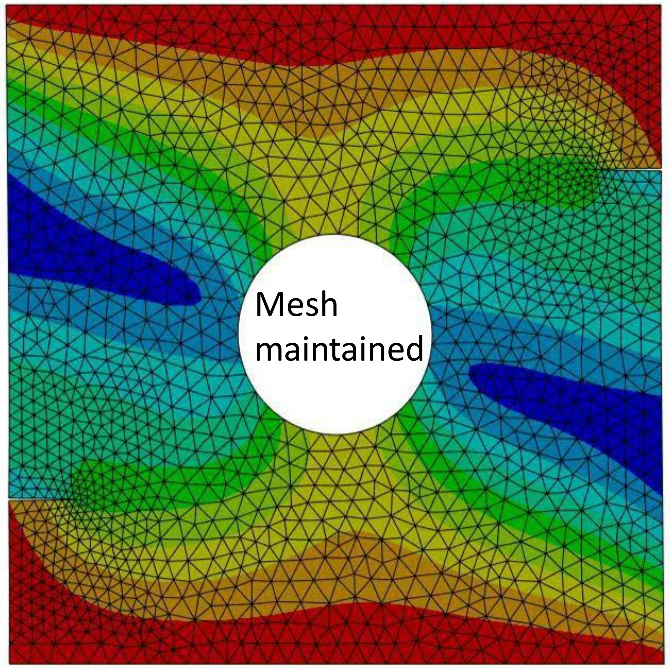
Fatigue crack growth
Mesh coarsening option: AGGR



Without mesh sizing control



With mesh sizing control



New Feature:

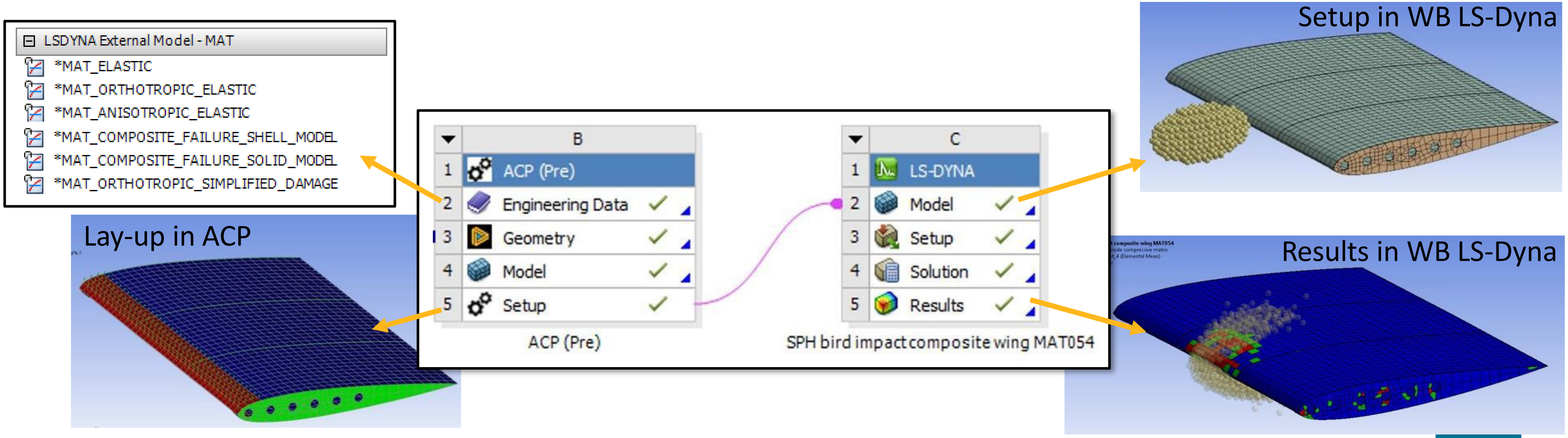
Composite post-processing functionalities moved into DPF plugin

Value Provided:

- More efficient post-processing
- More efficient and robust
- Easy to expose the composite post-processing functionalities in different contexts such as pyAnsys
- Flexibility for the future

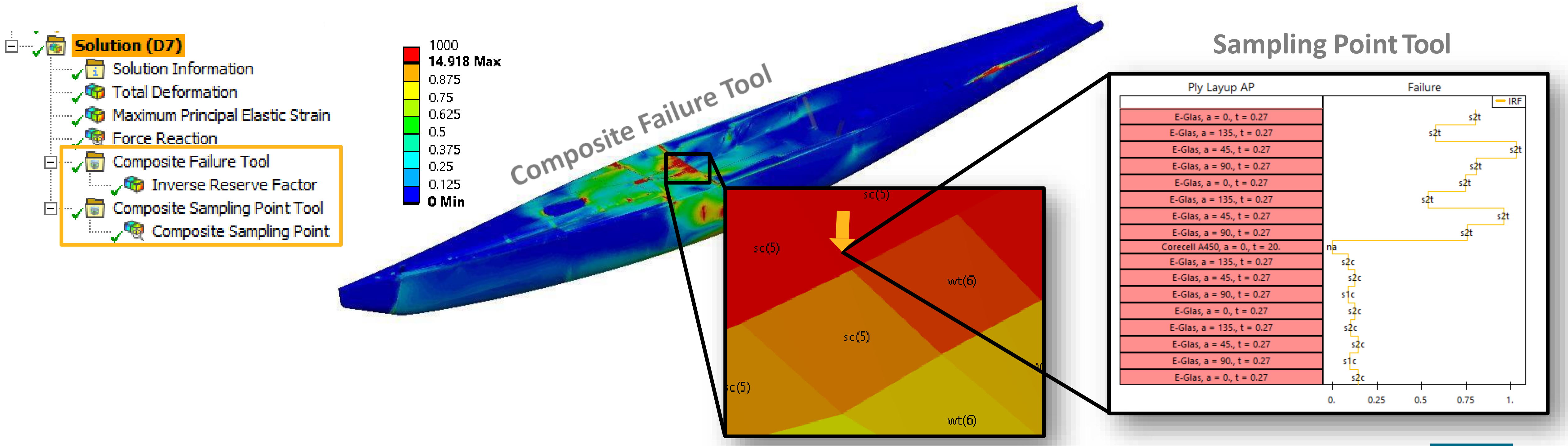
ACP – LS-DYNA Workflow in Workbench

- Analyze composite structures with respect to crash and impact
- Supports shell models as *ELEMENT_SHELL, **solid models** as *ELEMENT_TSHELL (beta) and any kind of **assemblies** (new)
- Many LS-DYNA material cards are now available in **Engineering Data** allowing the definition of a material card per ply material



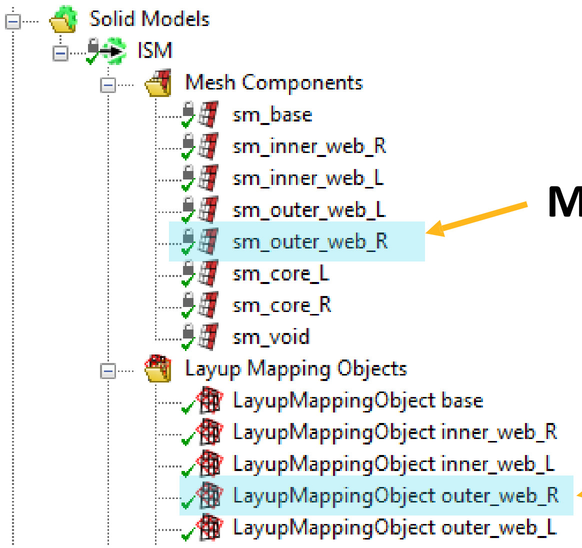
Composite Post-Processing in Mechanical

- Composite post-processing is a new plugin of DPF
- Used by the existing Composite Failure Tool and Sampling Point Tool in Mechanical
- More efficient and more robust
- Allows for more flexible post-processing in the future (pyAnsys)

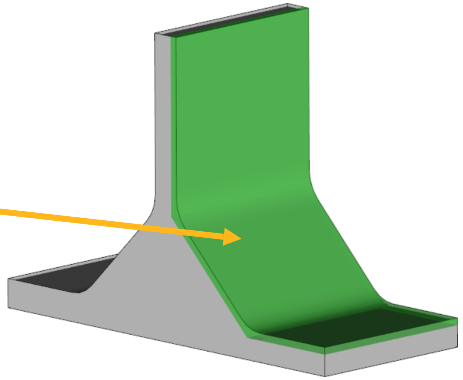


Transferring the Lay-up on a Solid Mesh

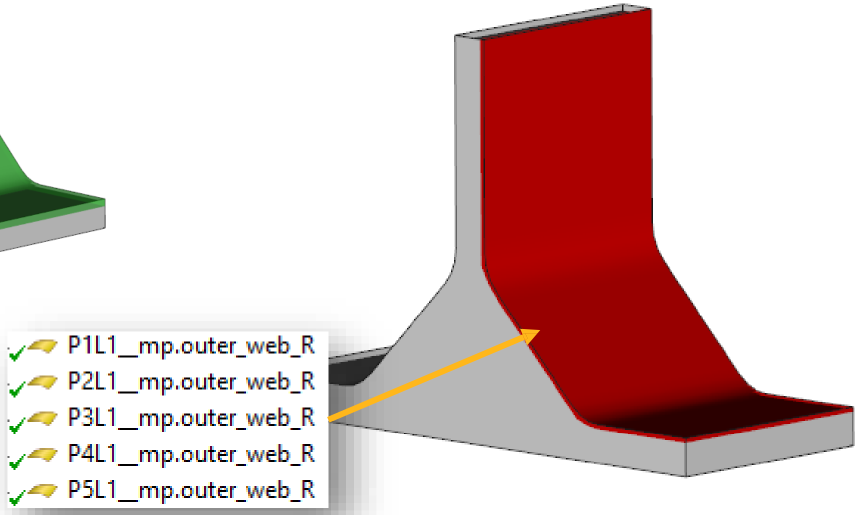
- The mapping can now be restricted to sub-domains of the solid mesh.
- With the new **Lay-up Mapping Objects** it is possible to define exactly which plies are transferred onto which regions (**Mesh Components**) of the volume mesh.
- This improves the handling of complex and even hybrid structures



Mesh Components

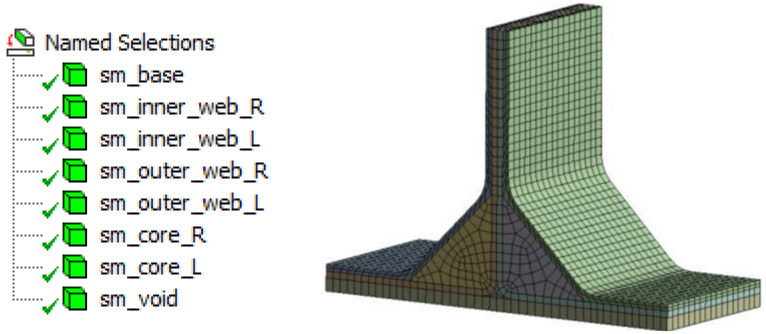


Lay-up Mapping Objects
Selection of plies to map onto mesh component(s)

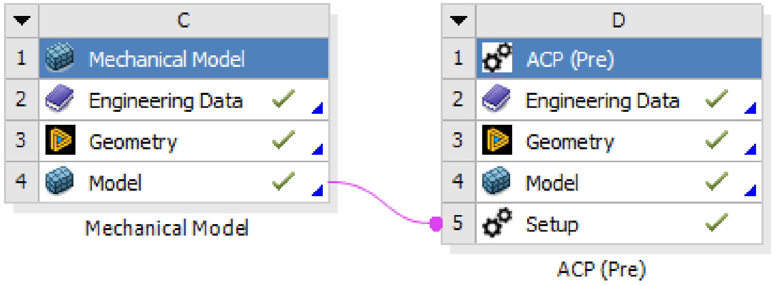


Lay-up Mapping: Modeling a T-Joint *

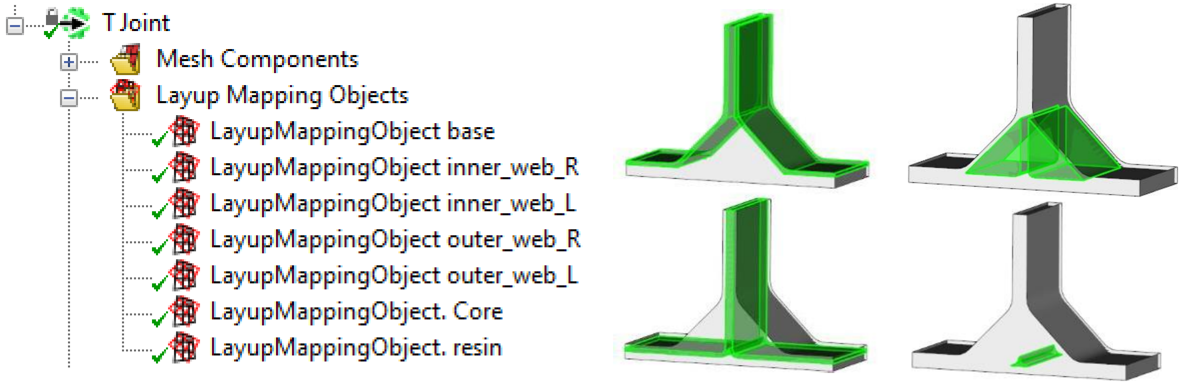
1. Create a solid mesh with Named Selections



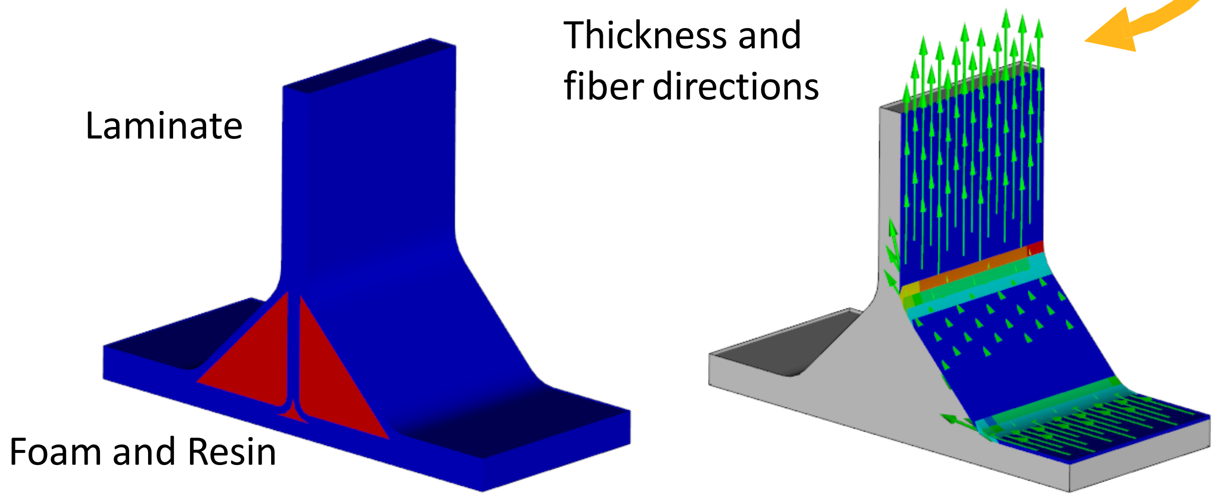
2. Pass the solid mesh (C4) to ACP Pre (D5)



3. Define lay-up mapping: source (plies) and target (Mesh Components)



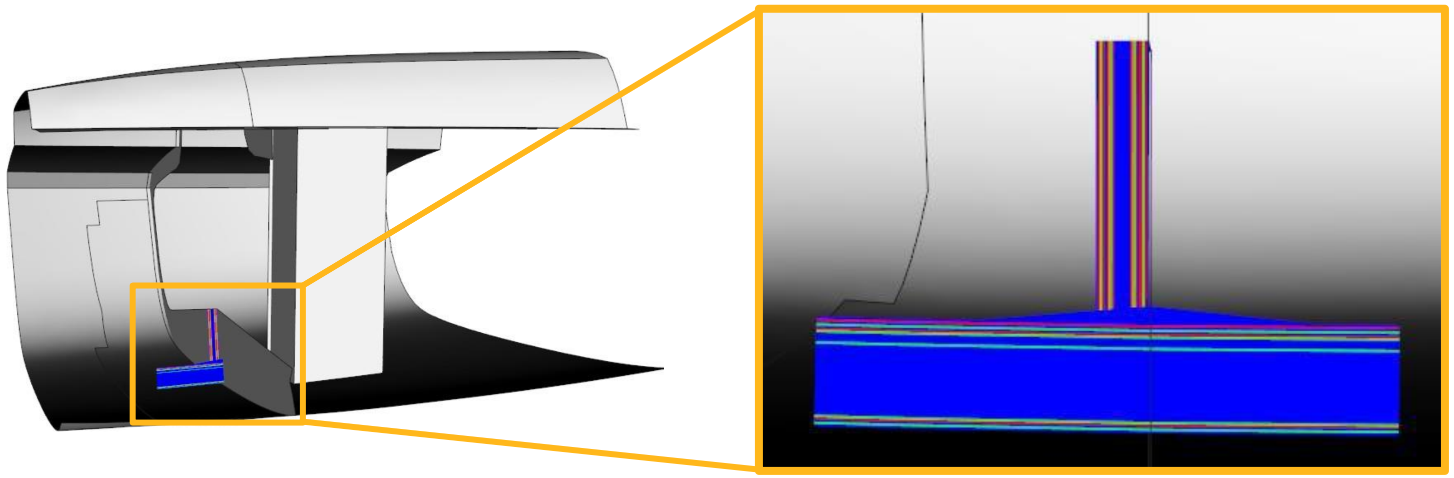
4. Output: layered solid model

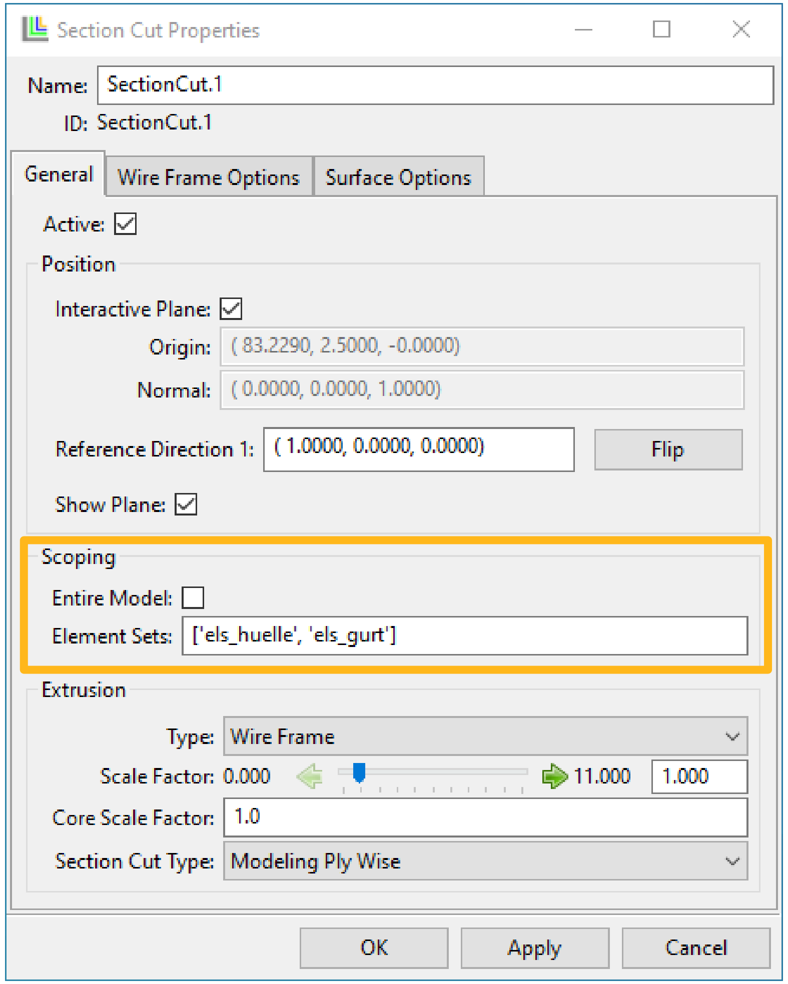


Section Cut Scoping

- Section Cuts allow visualizing and exporting the composite lay-up definition on an arbitrary section plane
- An option was added to restrict the scope of a Section Cut to selected Element Sets
- This makes Section Cuts easier to interpret, especially in complex geometries

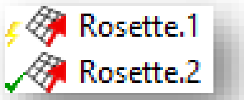
Section Cut with ply-wise angles



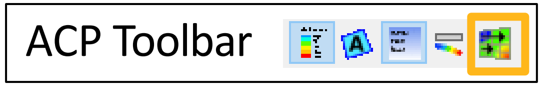


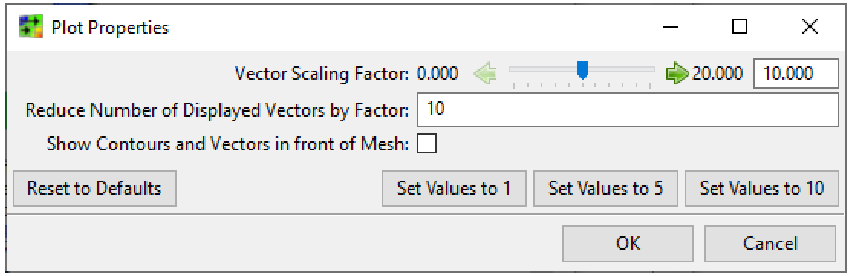
Graphical User Interface

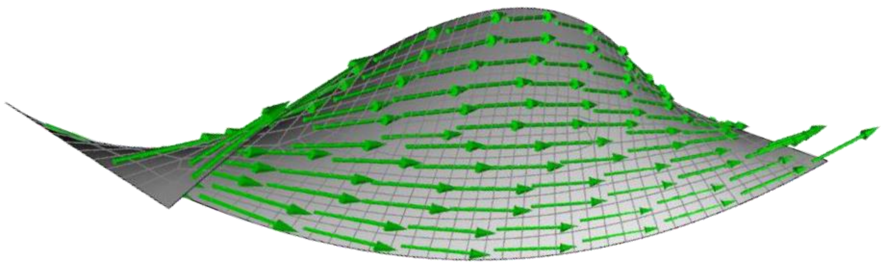
- The UI of ACP has been improved in these areas
  - Use **F2** for renaming objects directly via the Object Tree.
  - Hot keys were added for Copy & Paste (**Ctrl + C, Ctrl + V**). Multi-selection is also supported.
  - The Object Tree now always shows automatically the latest state (up-to-date, out-of-date) of the objects.



  - Vector plots such as fiber directions or orientations were further refined. Use the Plot Properties dialog from the toolbar to manually adjust the settings (scaling factor and density).







New Feature:

Multizone Hex Meshing

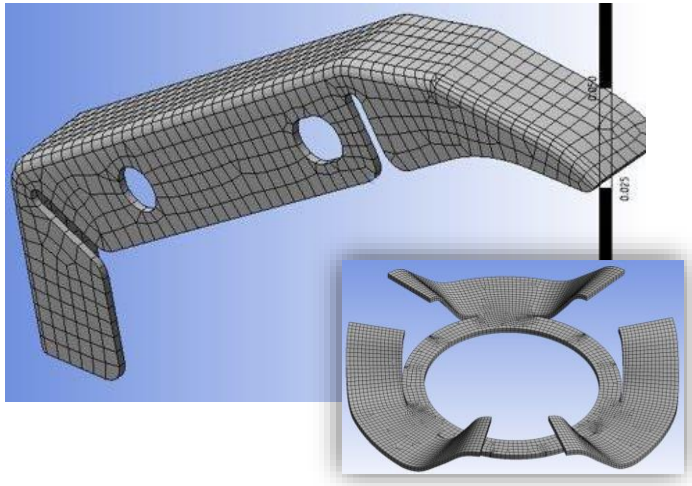
Value Provided:

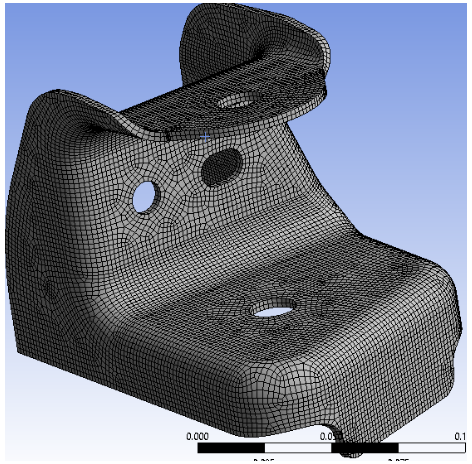
- Obtain hex elements with less requirements with CartSweep and ThinSweep decomposition
- Improved robustness of automatic-thickness

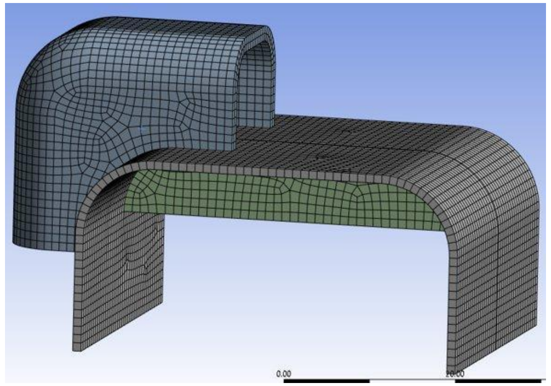
Thin Sweep Decomposition for Multizone

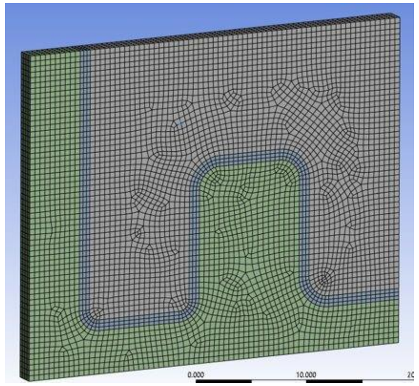
Definition	
Suppressed	No
Method	MultiZone
Decomposition Type	Thin Sweep

- **Thin Sweep** is a New Multizone Decomposition option for thin bodies
- Blocking technology is used to create and mesh 2D block on one side and pull automatically into 3D block
- In 22R2 we begin to support some simple bodies where connectivity is required
  - Share topology connections from two sides working on simple cases





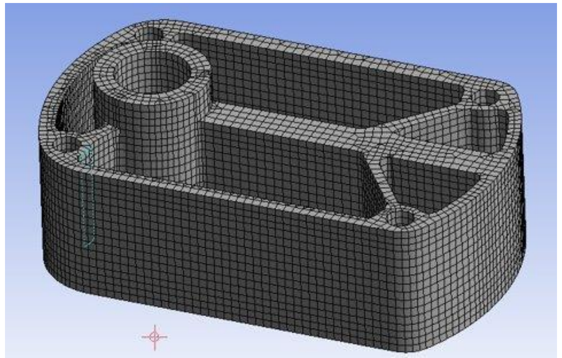


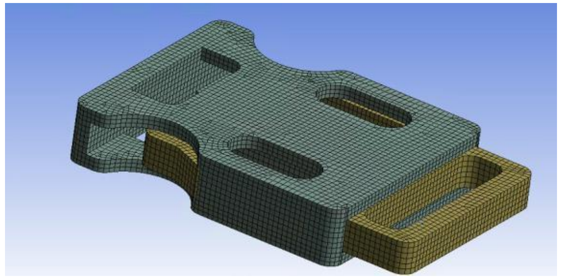


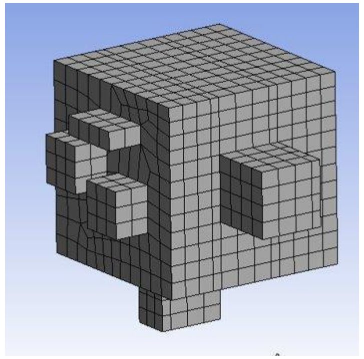
Cart Sweep Decomposition for Multizone

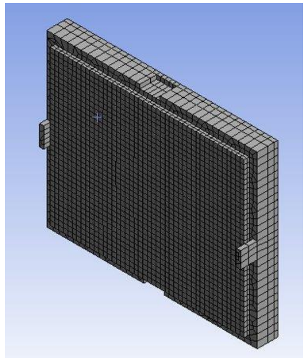
Definition	
Suppressed	No
Method	MultiZone
Decomposition Type	Cart Sweep

- **Cart Sweep** is a New Multizone Decomposition option for 2.5D bodies
  - Sweepable in one Cartesian direction
  - Direction can be specified by picking source faces
- A Cartesian background grid is used to create blocks
- Allows users to mesh with hex elements with less requirement to decompose into hex-able volumes in geometry tools









New Feature:

Enhanced Material Support for the LS-DYNA Solver

Value Provided:

- Simplifies the modeling process
- Enables Advanced Material Modeling for a wide range of applications including Drop Test, Impact, and Composites Application

Material Improvements

Improved Material Support

- Additional fields are made available for the definition of LS-DYNA specific material MAT_SIMPLIFIED_RUBBER/FOAM
- Materials added:
  - MAT_OGDEN_RUBBER including all input card variations and optional cards
  - MAT_GENERAL_VISCOELASTIC

	A	B	C	D	E
1	Property	Value	Unit		
2	*MAT_GENERAL_VISCOELASTIC				
3	Definition				
4	Tensile Pressure Elimination Flag, pcf	0			
5	Elastic Flag, ef	0			
6	Number of Terms in Shear Fit, nt	0			
7	Shear Fit Coefficient, bstart	0	s^-1		
8	Optional Ramp Time for Shear Loading, tramp	0	s		
9	Number of Terms in Bulk Fit, ntk	0			
10	Bulk Fit Coefficient, bstartk	0	s^-1		
11	Optional Ramp Time for Bulk Loading, trampk	0	s		
12	Initial Shear Modulus, G0	1	Pa		
13	Initial Bulk Modulus, K0	1	Pa		

LSDYNA External Model - MAT

- *MAT_ELASTIC
- *MAT_ORTHOTROPIC_ELASTIC
- *MAT_ANISOTROPIC_ELASTIC
- *MAT_PLASTIC_KINEMATIC
- *MAT_BLATZ-KO_RUBBER
- *MAT_HIGH_EXPLOSIVE_BURN
- *MAT_NULL
- *MAT_JOHNSON_COOK
- *MAT_POWER_LAW_PLASTICITY
- *MAT_PIECEWISE_LINEAR_PLASTICITY
- *MAT_ENHANCED_COMPOSITE_DAMAGE
- *MAT_LAMINATED_COMPOSITE_FABRIC
- *MAT_CRUSHABLE_FOAM
- *MAT_OGDEN_RUBBER
- *MAT_SIMPLIFIED_JOHNSON_COOK
- *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY
- *MAT_SIMPLIFIED_RUBBER/FOAM
- *MAT_BILKHU/DUBOIS_FOAM
- *MAT_FABRIC
- *MAT_ADD_EROSION
- *MAT_ADD_DAMAGE_GISSMO
- *MAT_GENERAL_VISCOELASTIC

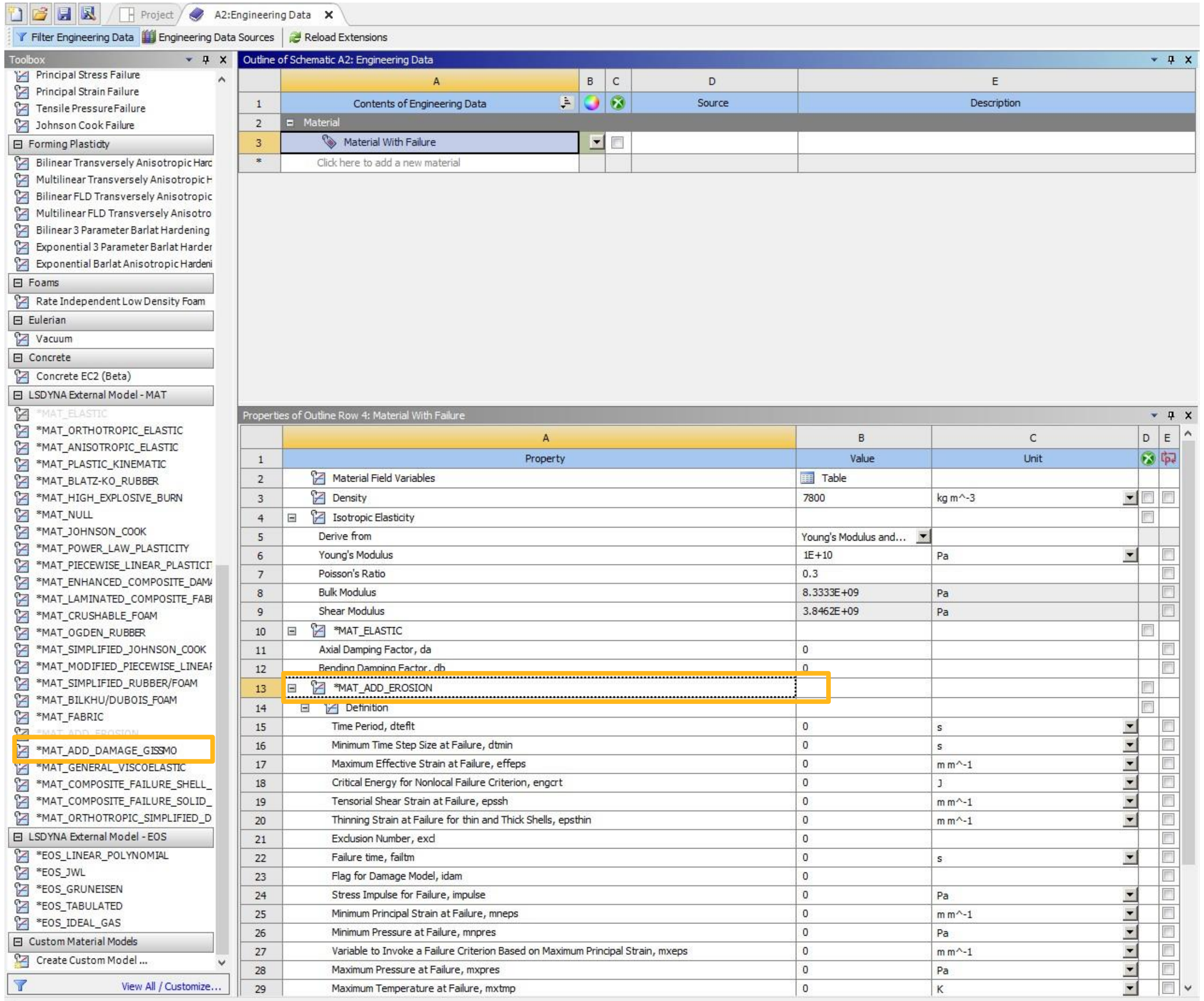
Improved Material Support

Many of the constitutive models in LSDYNA do not incorporate failure and erosion in the main model.

❼ The following Material Models additions provide a way of including damage and failure in these models.

*MAT_ADD_EROSION

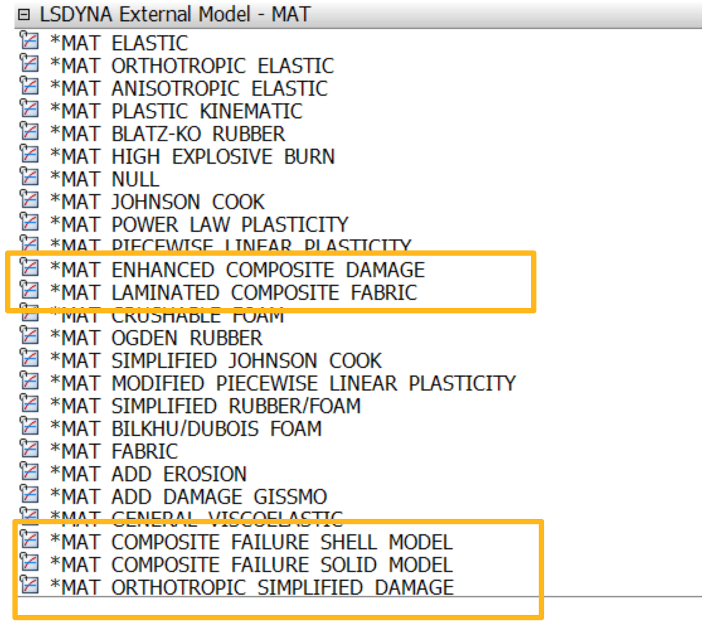
*MAT_ADD_DAMAGE_GISSMO



Improved Material Support

•Materials models targeting composite and turbomachinery applications have been Added. They enable advanced ply modelling and failure

- *MAT_ENHANCED_COMPOSITE_DAMAGE
- *MAT_LAMINATED_COMPOOSITE_FABRIC
- *MAT_COMPOSITE_FAILURE_SHELL_MODEL
- *MAT_COMPOSITE_FAILURE_SOLID_MODEL
- *MAT_ORTHOTROPIC_SIMPLIFIED_DAMAGE



ALE Improvements

Improved Support for ALE

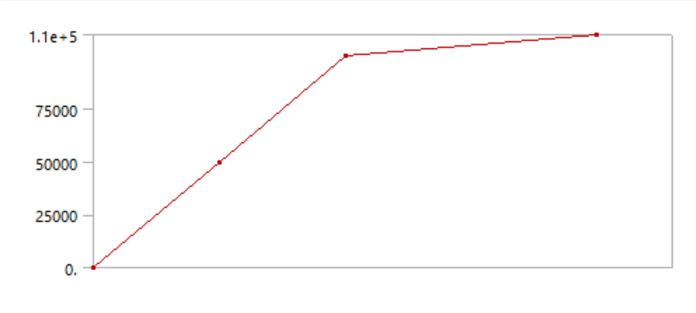
- Field added to specify reference pressure under analysis settings
- Option to use newer coupling algorithm (ALE_Structured_FSI) for S-ALE
- Ability to define coupling stiffness using tabular data

ALE Controls	
Continuum Treatment	Use Alternate Advection Logic
Cycles Between Advection	1
Advection Method	Donor Cell + Half Index Shift
Simple Average Weighting Factor	-1
Volume Weighting Factor	0
Isoparametric Weighting Factor	0
Equipotential Weighting Factor	0
Equilibrium Weighting Factor	0
Advection Factor	0
Start	0 s
End	1E+20 s
Reference Pressure	0 Pa
Advanced	
Output Controls	

Details of "Coupling"

Lagrange Bodies	
Scoping Method	Geometry Selection
Geometry	1 Body
ALE Bodies	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Fluid Structure Interaction Type	Program Controlled
Fluid Structure Coupling Method	Program Controlled / Constrained Lagrange in Solid / ALE Structured FSI
Coupling Direction	
Number of Coupling Points	2
Lagrange Normals Point Toward ALE Fluids	Yes
Leakage Control	None
Stiffness Type	Tabular
Stiffness Load Curve	Tabular Data
Minimum Volume Fraction to Activate Coupling	0.5
Friction	0
Birth Time	0 s
Death Time	1E+20 s

Graph



Tabular Data

	Displacement [m]	Coupling Pressure
1	0.	0.
2	5.e-003	50000
3	1.e-002	1.e+005
4	2.e-002	1.1e+005
*		

Structured FSI Coupling Method

- The new structured FSI coupling method can now be used for S-ALE simulations

Details of "Coupling"

Lagrange Bodies	
Scoping Method	Geometry Selection
Geometry	
ALE Bodies	
Scoping Method	Geometry Selection
Geometry	
Definition	
Fluid Structure Interaction Type	Program Controlled
Fluid Structure Coupling Method	Program Controlled Constrained Lagrange In Solid ALE Stuctured FSI
Coupling Direction	
Number of Coupling Points	2
Lagrange Normals Point Toward ALE Fluids	Yes

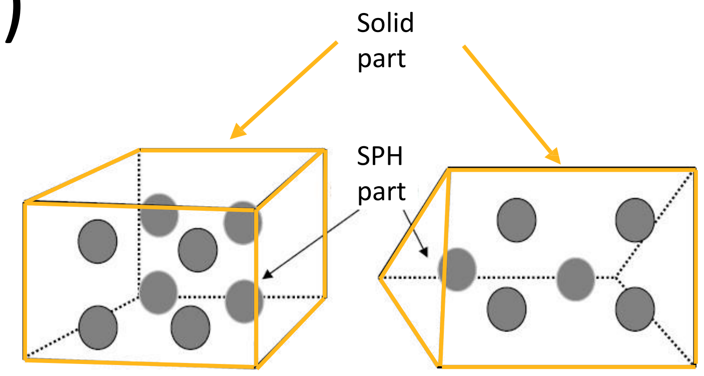
```
*ALE_STRUCTURED_FSI
$     slave    master     sstyp     mstyp    unused    unused    unused     mcoup
          1        45         1         1                                    -2
$     start       end      pfac    unused    unused      flip    unused    unused
          0     1E+20       0.1                             0
```

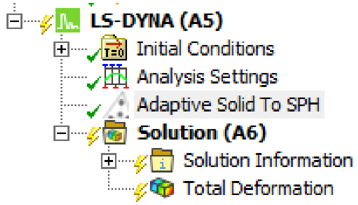
SPH Improvements

Adaptive Solid to SPH (Material Assignment)

- SPH particles existing inside the solid element are inactive initially and become active when the solid element reaches failure criteria defined by the material of the solid part.
  - The SPH particles are created inside the solver.

If the user want the generated particles (after failure) to have different material, he/she should set material Assignment to User Defined. In the field Material, we can access all materials defined in the model.





Details of "Adaptive Solid To SPH"

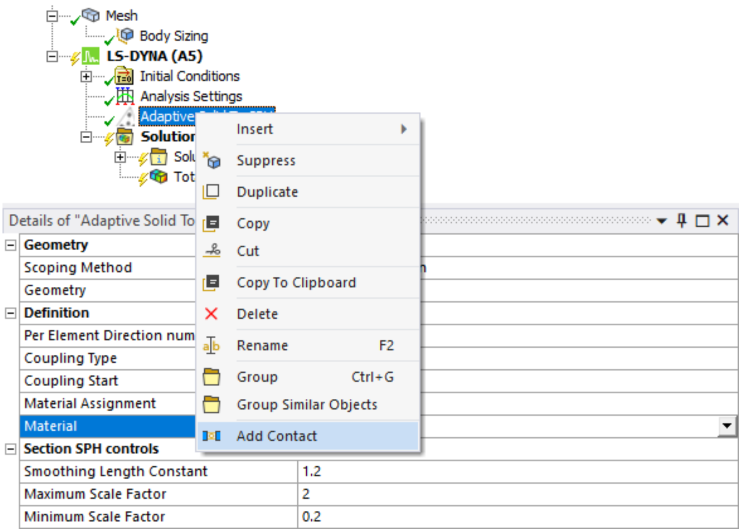
Geometry	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Per Element Direction number of particles	1
Coupling Type	Debris
Coupling Start	From Beginning
Material Assignment	User Defined
Material	Structural Steel
Section SPH controls	
Smoothing Length Constant	1.2
Maximum Scale Factor	2
Minimum Scale Factor	0.2

Adaptive Solid to SPH (Material Assignment)

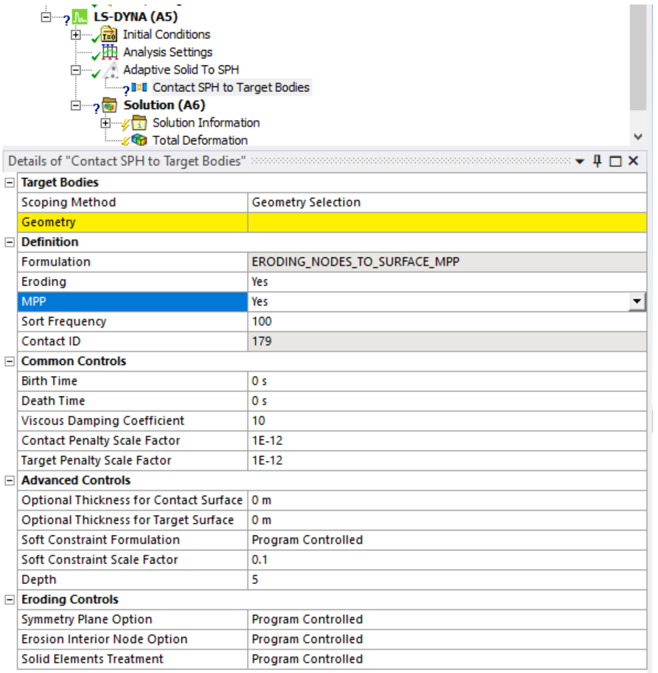
- The Adaptive Solid To SPH particles are created inside the solver
- The SPH part is not visible on mechanical and cannot be scoped

in contact.

- A child object under the corresponding adaptive solid to SPH called Contact SPH to Target Bodies can be added.



- The user should set the target Bodies and the contact properties.
- The user can run the contact algorithm in MPP configuration by setting the processing type to MPP in analysis settings
- The user can add erosion to contact by Eroding property to Yes



CAD-IT CONSULTANTS

New Feature:

Fully automated element embedding workflow for modeling intersecting components of elements

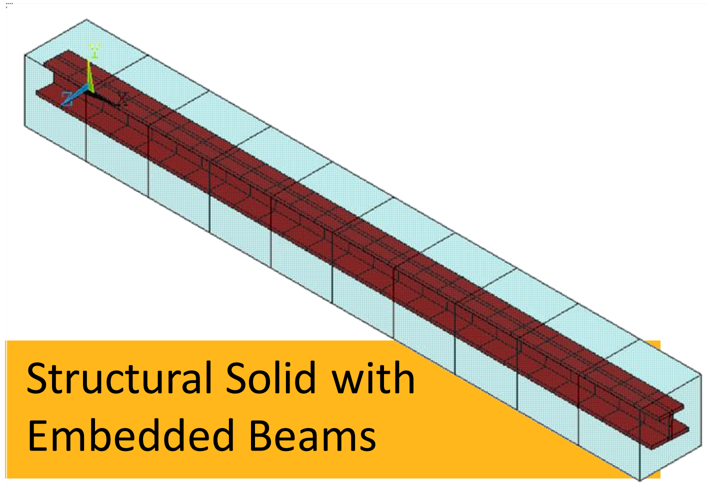
Value Provided:

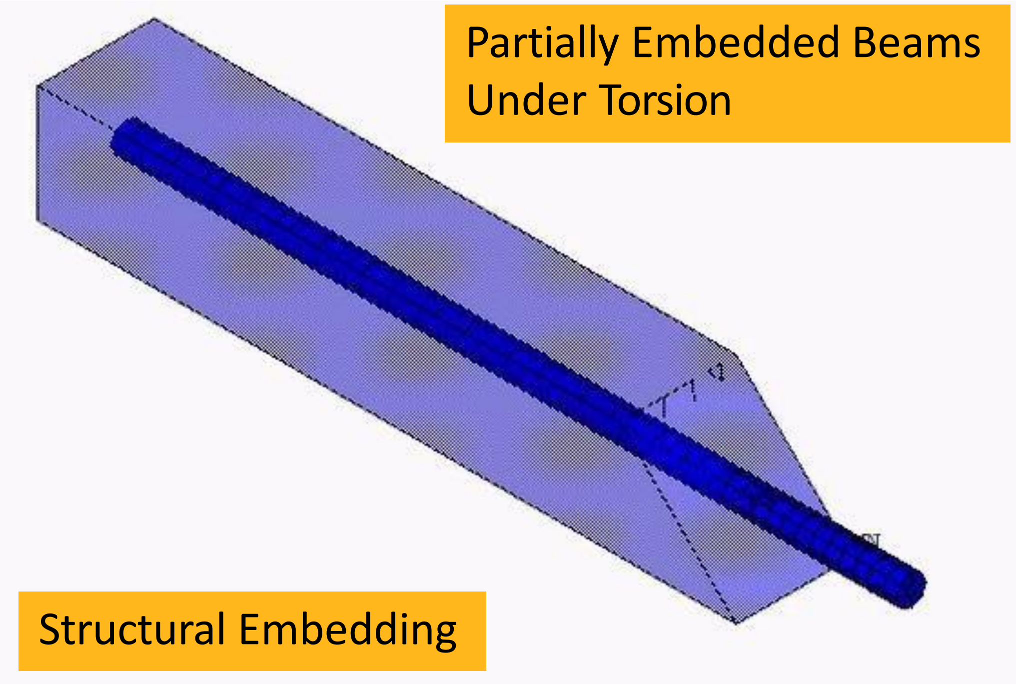
- New workflow greatly reduces the model preparation time and computational costs
- Integrated seamlessly with the released Reinforcing capability
- Powerful usability and accuracy enhancements by harvesting all embedded element features

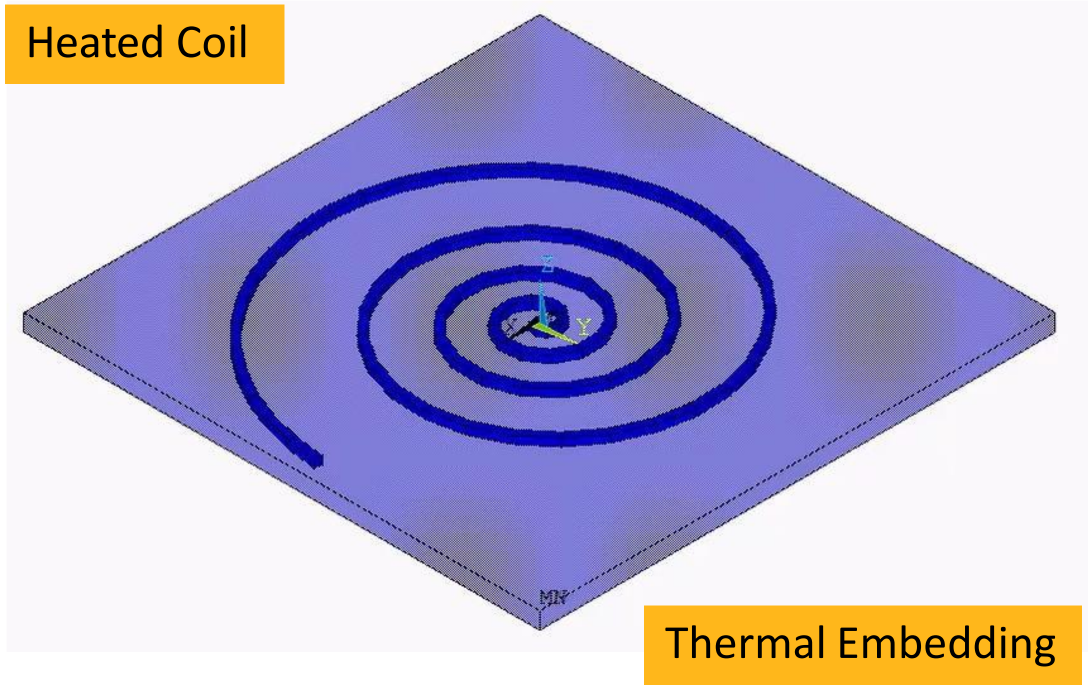
CAD-IT CONSULTANTS

Direct Element Embedding Workflow

- The connections between two intersecting elements or element components (embedded and base elements) can now be automatically established with a new Direct Embedding procedure. A new command **EEMBED** is introduced
- Structural beams/links and thermal links are supported for embedded members
- Extensive applications: electronics reliability, composites, biomedical

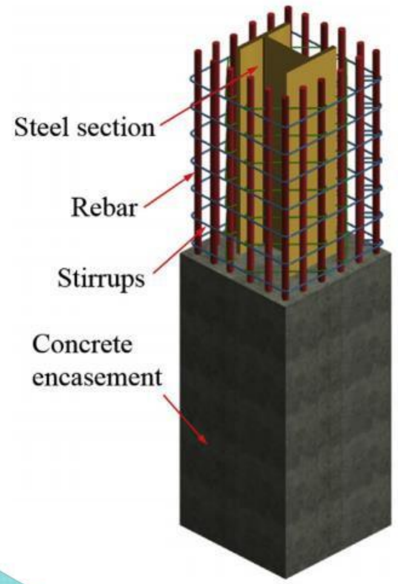


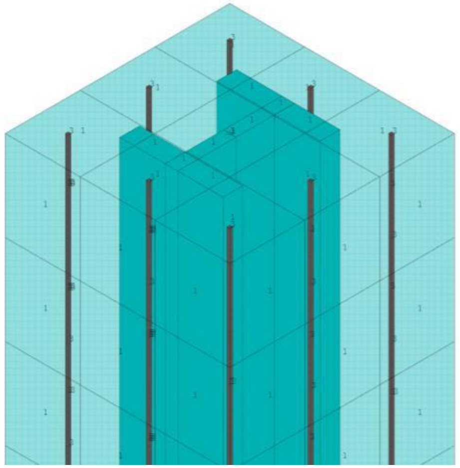




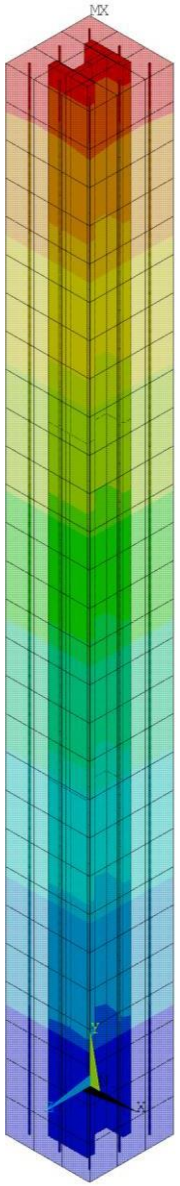
Advantages of Direct Element Embedding

- Can be seamlessly integrated with current Reinforcing workflow (see example on the right)
- Allows both embedded members and base materials to be modeled with standard structural or thermal elements
- Accounts for the actual cross-section geometry of the embedded members in embedded / base element connections.
- Improves the modeling accuracy (for instance, inclusion of bending and torsional stiffness for embedded members is now feasible using standard beam elements)
- Effectively eliminates the solution sensitivity with respect to the mesh density.





SOLID185 – Concrete Encasement
Reinforcing - Rebars
Direct embedding – Steel Section



New Feature:

Co-Simulation of Aqwa, Rigid Dynamics and AeroDyn for the offshore wind turbine

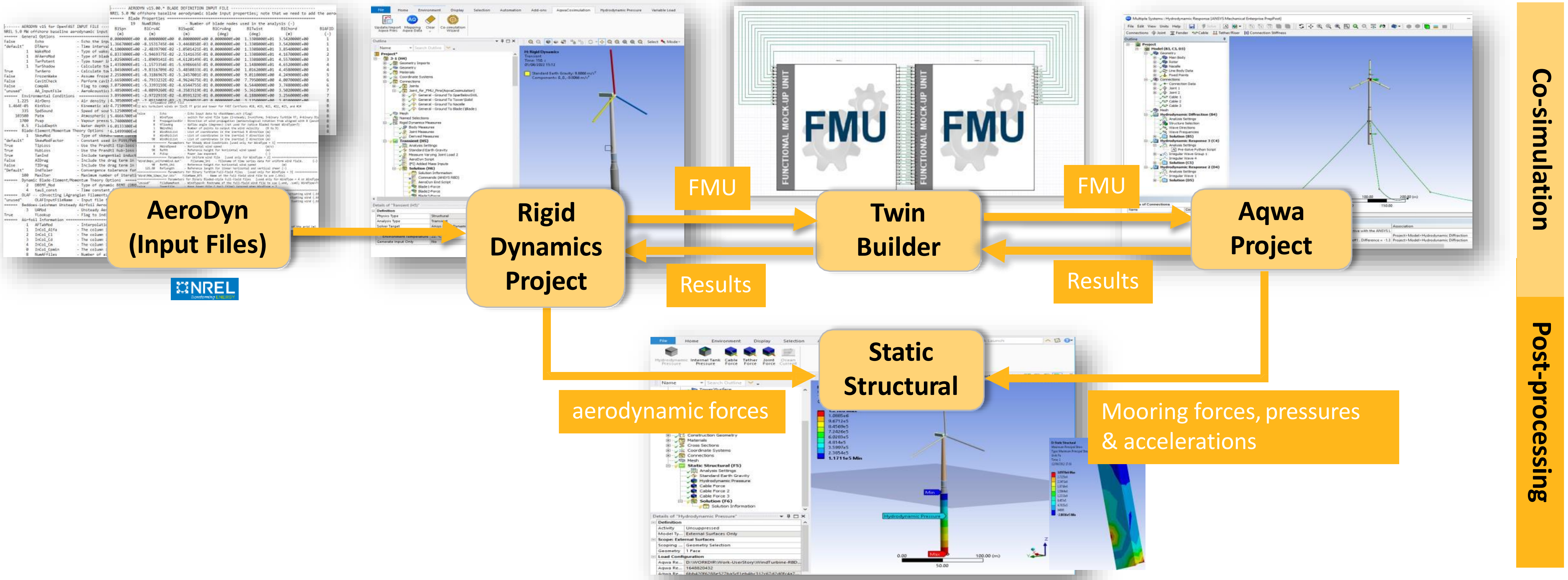
Value Provided:

- Enables user to simulate the coupling system of hydrodynamics, mooring dynamics, aerodynamics, and structural dynamics of the offshore floating wind turbine, providing the transient responses and stresses on the floating wind turbine system under various sea and wind conditions

CAD-IT CONSULTANTS

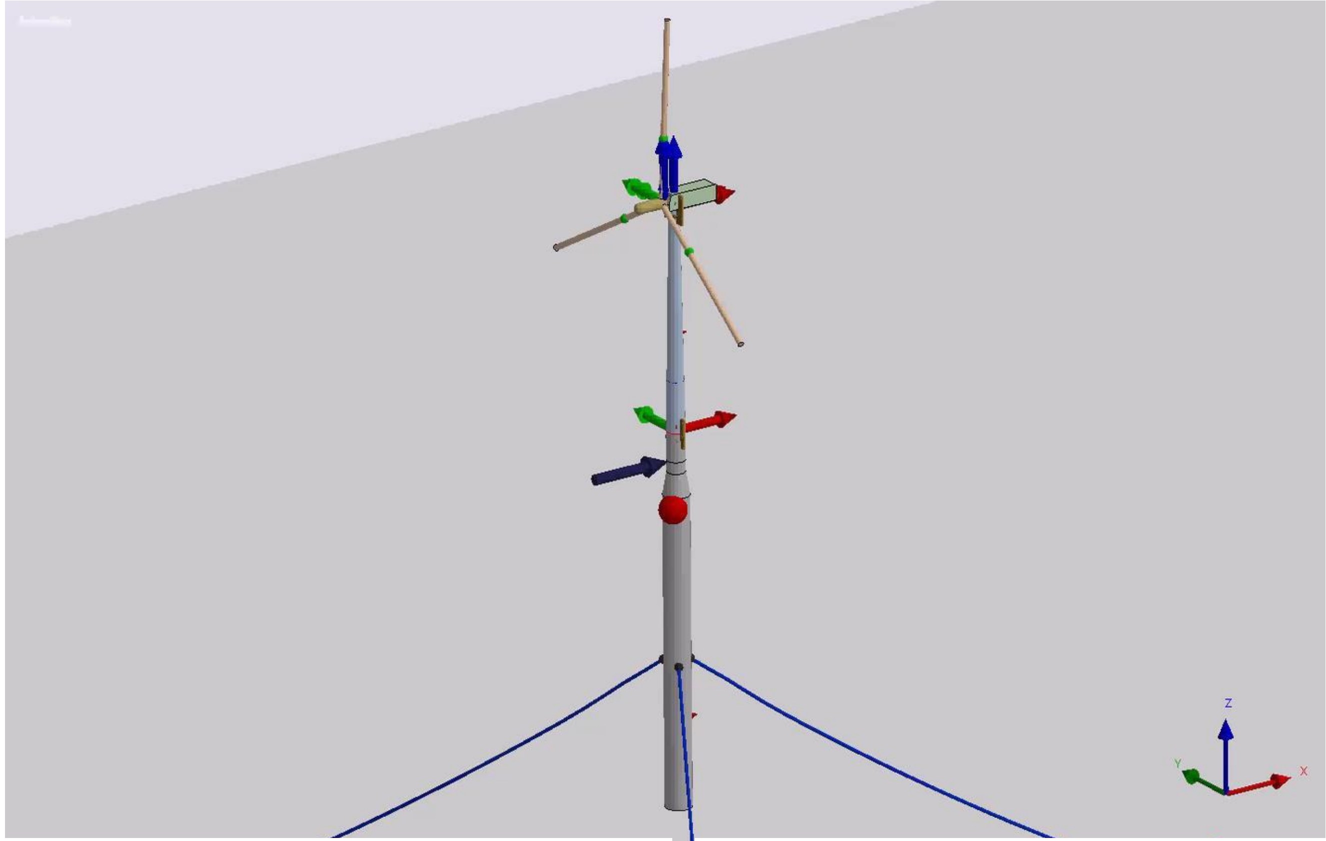
Floating Wind Turbine Co-simulation Workflow

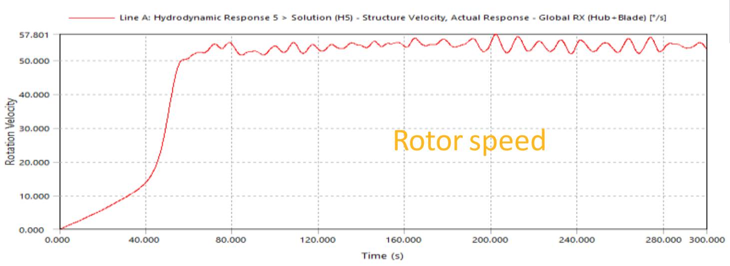
- Provides an integrated procedure for the time domain hydrodynamics, mooring dynamics, aerodynamics and structural dynamics response analysis for floating wind turbine systems

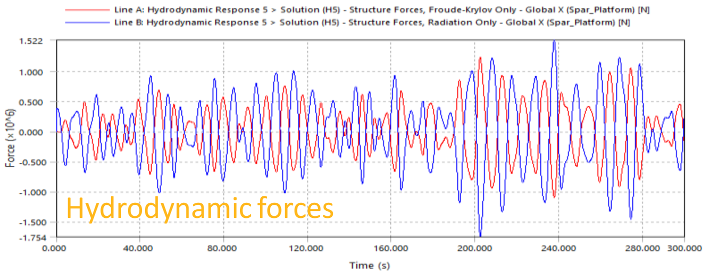


Pre and Post Processing of Co-simulation in Workbench

- Installation:
  - AqwaCosimulation Extension pre-installed with Workbench
  - Manually loaded from Extension > Manage Extensions
- Inputs:
  - Model geometry and mooring definition
  - Ocean environment settings
  - Blade and tower sectional aerodynamic data
  - Generator torque vs. Speed curve
- Results:
  - Time history of responses
  - Flexible structural analysis







New Feature:

Structural Optimization

Value Provided:

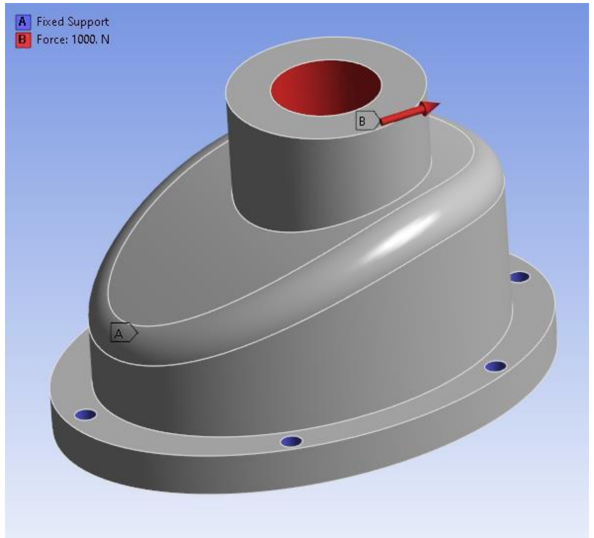
- New optimization methods and constraints open up more possibilities for customers to create optimized designs.

New Features & Capabilities

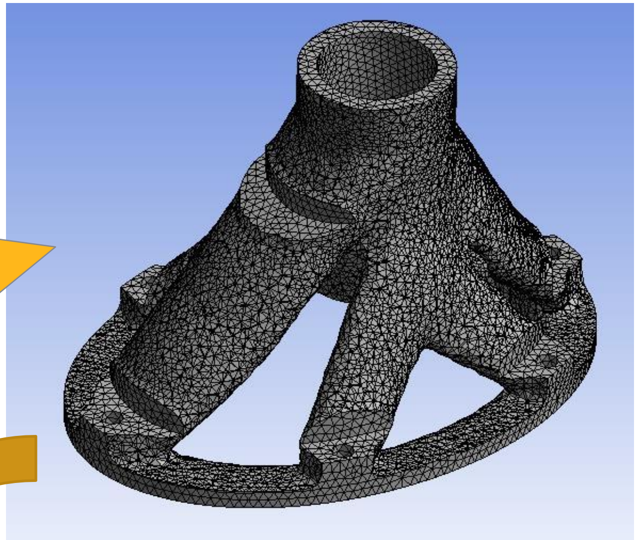
- New manufacturing constraint
- Design constraint

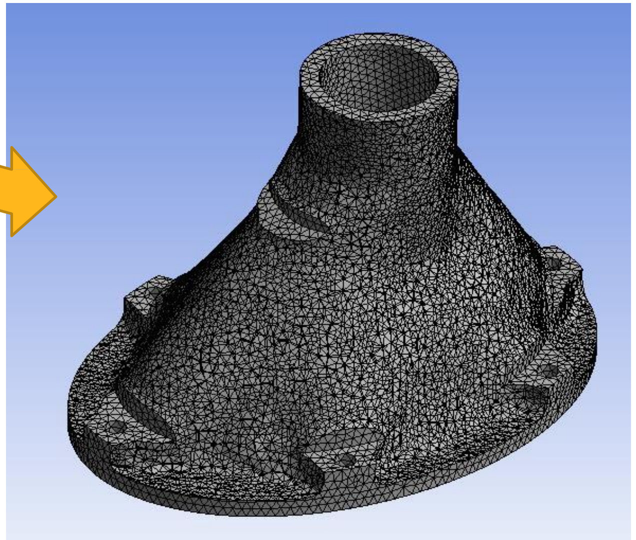
Topology Optimization - Housing

$$\begin{cases} \min\limits_{\Omega} compliance \\ vol < 40\% \end{cases}$$

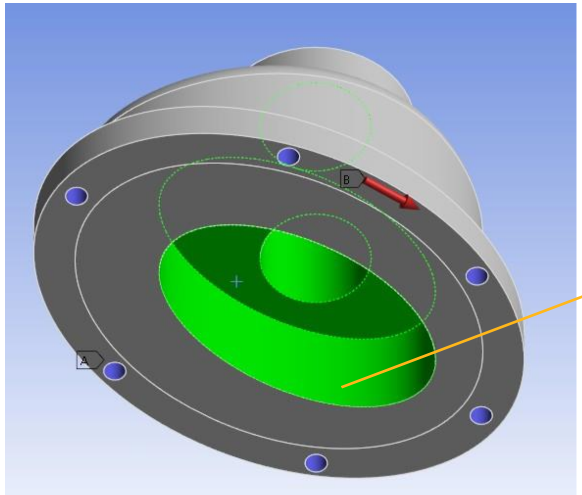


- Topology Optimization delivers the lightest solution. However, designs are almost systematically perforated which makes them inappropriate for some contexts.
- For example, consider a housing that is clamped on another component and contains a liquid.
- **Would Topology optimization be capable to deliver a watertight design?**
- **Where should the envelop be optimally located?**

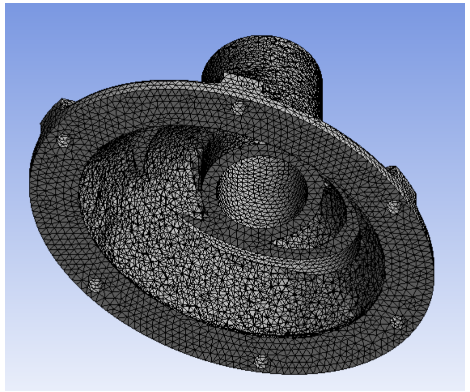




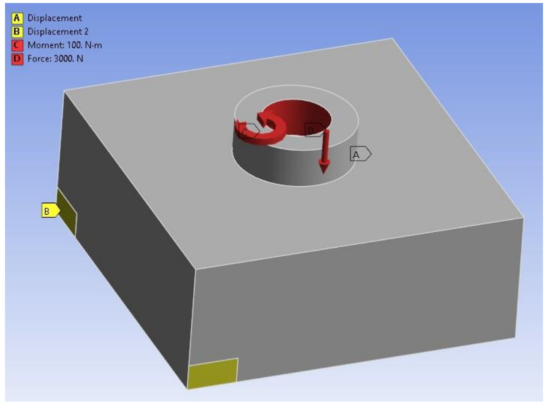
$$\begin{cases} \min\limits_{\Omega} complianc \\ vol < 40 \\ \boxed{+h} \end{cases}$$



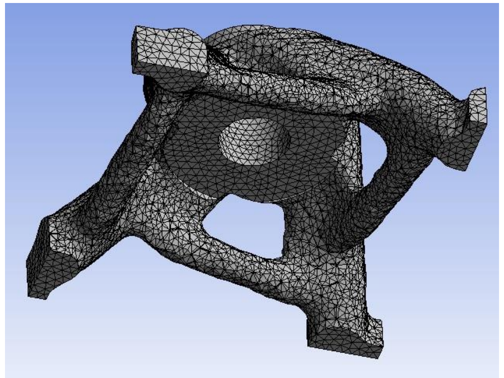
Surfaces to enclose

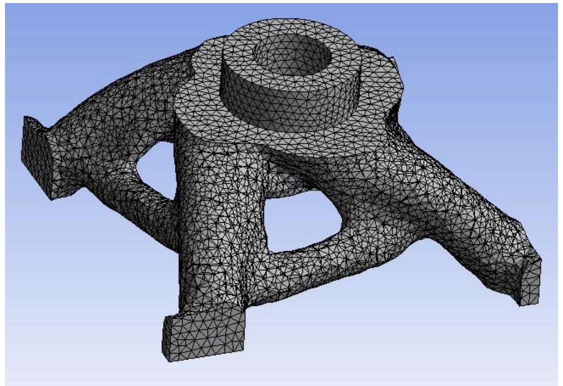


Topology Optimization - Housing

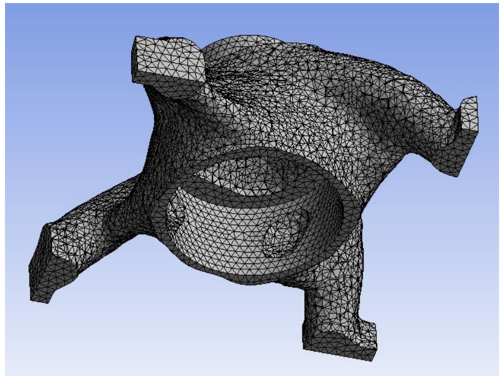


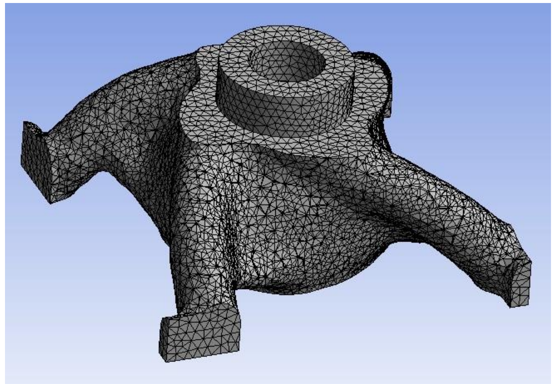
$$\begin{cases} \min\limits_{\Omega} compliance \\ vol < 30\% \end{cases}$$

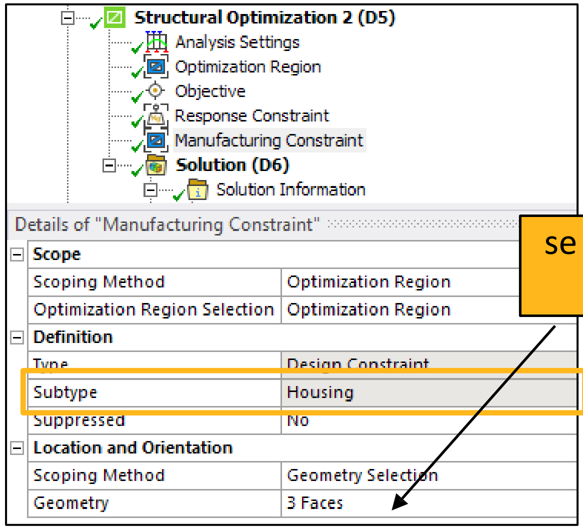




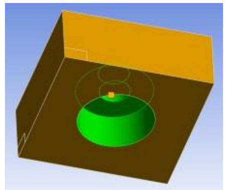
$$\begin{cases} \min\limits_{\Omega} compliance \\ vol < 30\% \\ +\text{housing (3 faces to enclose)} \end{cases}$$



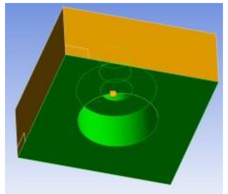


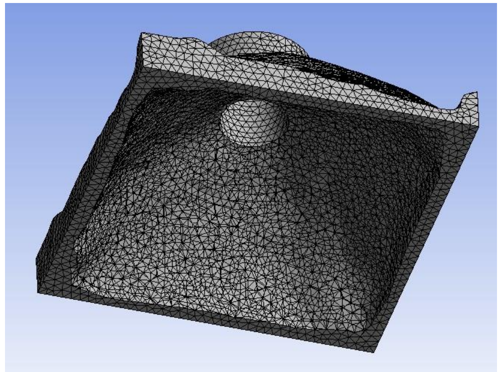


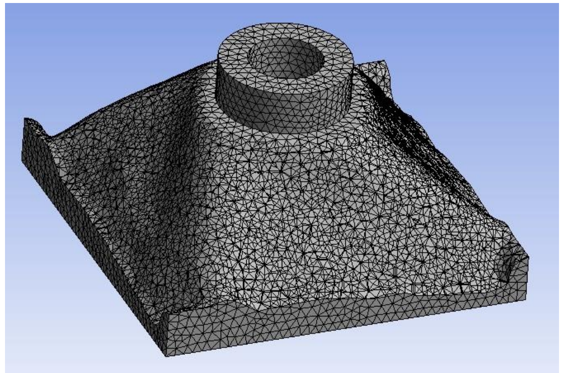
se ect the face(s) to enclose



$$\begin{cases} \min\limits_{\Omega} compliance \\ vol < 30\% \\ +\text{housing (5 faces to enclose)} \\ +\text{2-sided z-dir pullout} \end{cases}$$

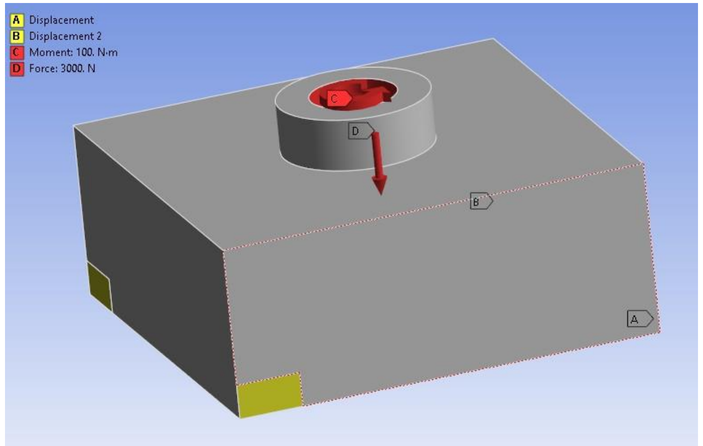






Stamping

- Given a working domain and a stamping direction, can topology optimization give any guidance to sketch plate-like design?

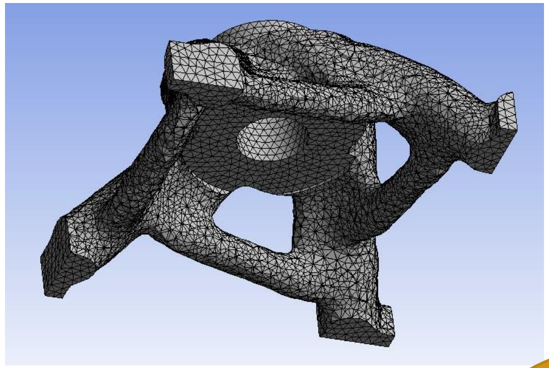


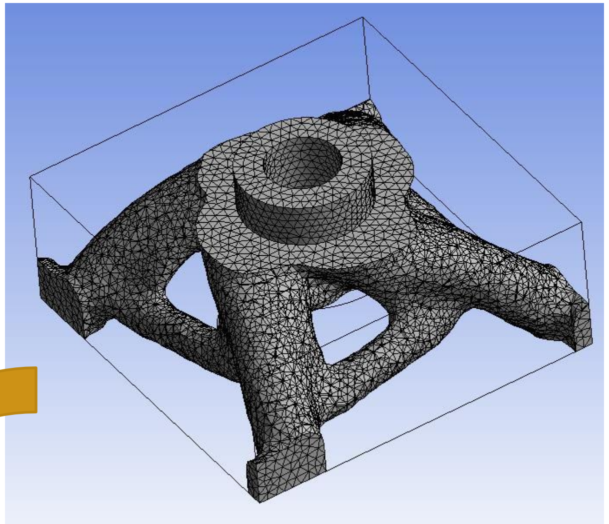
Details of "Manufacturing Constraint 2"	
Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Manufacturing Constraint
Subtype	Pull Out Direction
Suppressed	No
Pull Out Option	Stamping
Location and Orientation	
Coordinate System	Global Coordinate System
Axis	Z Axis
Direction	Both Directions

About the set-up
1)create a pull-out manuf. constraint
2) select the « stamping » option
3)define the pullout direction corresponding to your stamping direction

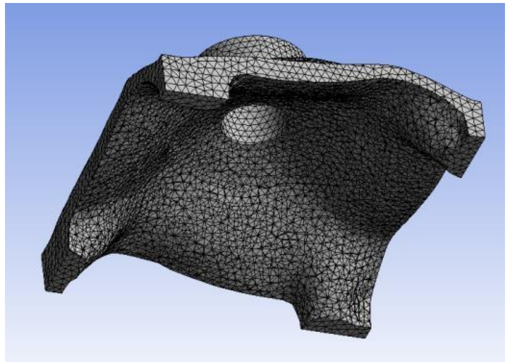
$$\begin{cases} \min_{\Omega} compliance \\ vol < 40\% \end{cases}$$

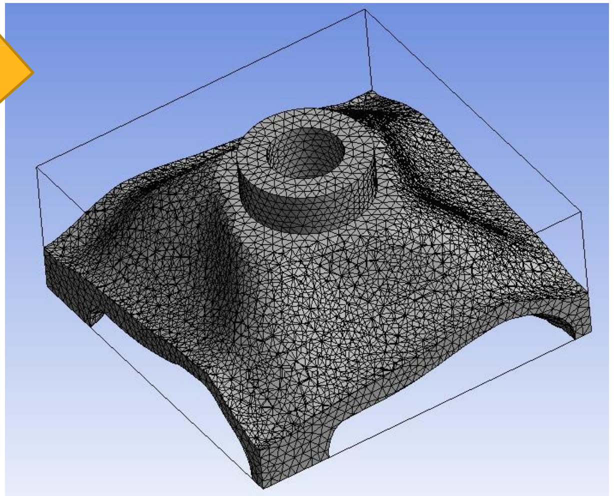
By nature, Topology Optimization is not capable to deliver plate design, it rather favors massive design.



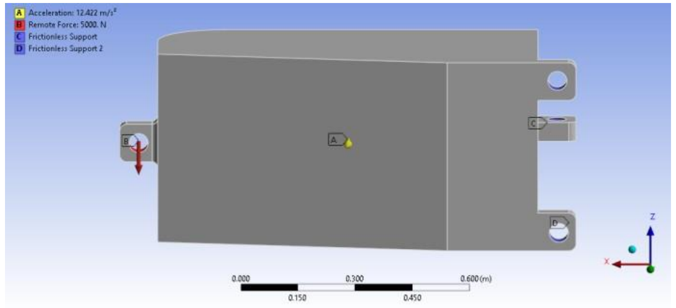


$$\begin{cases} \min_{\Omega} compliance \\ vol < 40\% \\ \boxed{+\text{stamping}} \end{cases}$$





No-hole (iso-topology)



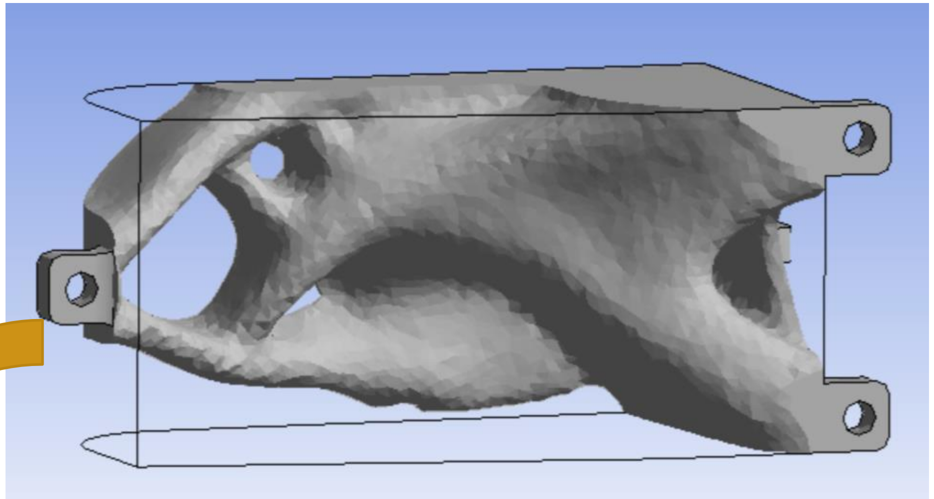
- In casting process it may be requested to have the simplest design as possible in a bid to ease the filling process. Any perforation or hole is somehow an obstacle that makes the filling stage more delicate and energy-consuming.
- The so-called « no-hole » feature aims to fulfill this manufacturing constraint.

Details of "Manufacturing Constraint"

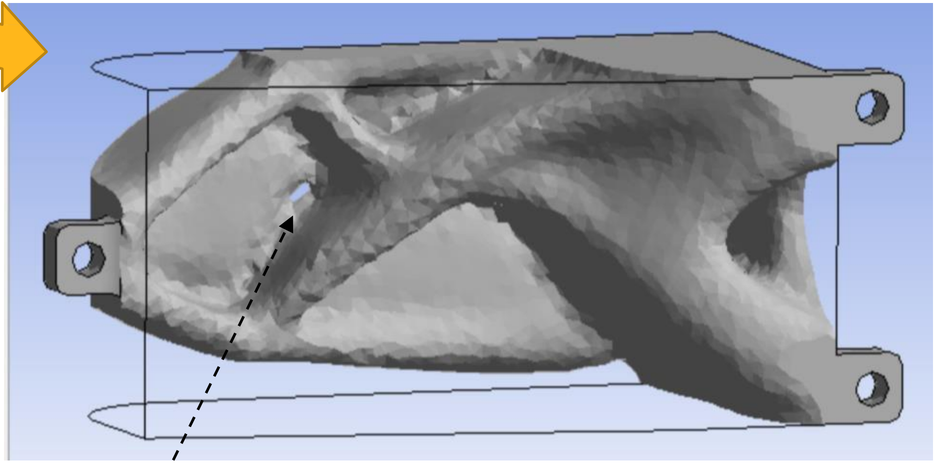
Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Manufacturing Constraint
Subtype	Pull Out Direction
Suppressed	No
Pull Out Option	No-Hole
Location and Orientation	
Coordinate System	Global Coordinate System
Axis	Y Axis
Direction	Both Directions

About the set-up
1)create a pull-out manuf. constraint
2)select the « no-hole » option

$$\begin{cases} \min\limits_{\Omega} compliance \\ mass < 50\% \\ \text{y-dir one-sided pullout} \end{cases}$$



$$\begin{cases} \min\limits_{\Omega} compliance \\ mass < 50\% \\ \text{y-dir one-sided pullout} \\ \boxed{\text{+no-hole}} \end{cases}$$



Known limitation: The no-hole constraint can be sometimes slighlty violated.

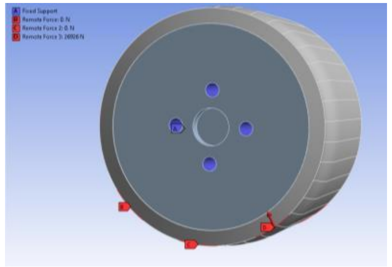
CAD-IT CONSULTANTS

Topology Optimization

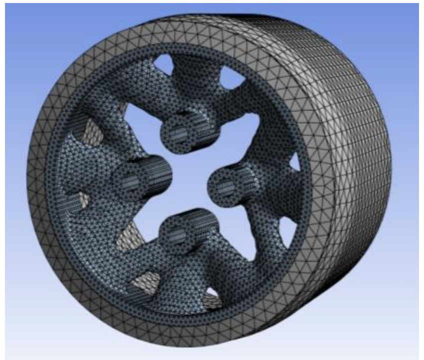
Design constraint with Level-Set

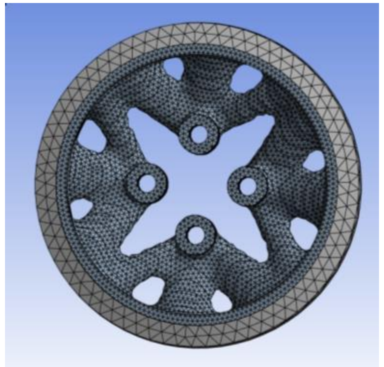
- Design constraints are now available in Level-set based Topology Optimization:
  - cyclic symetry,
  - plane symetry
  - and pattern repetition.

Mechanical setup
with 3 loadsteps

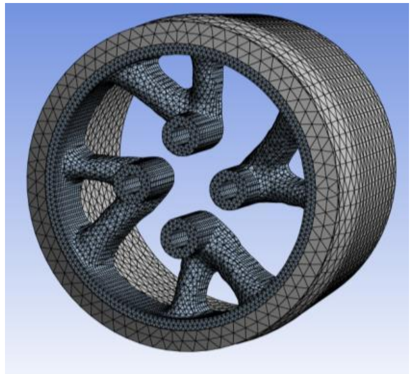


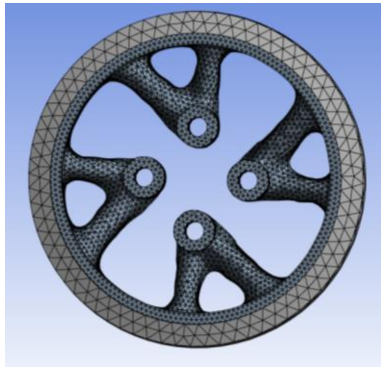
$$\begin{cases} \min\limits_{\Omega} \sum\limits_{k} complianc \\ mass \leq 35 \\ \text{+4th} \end{cases}$$



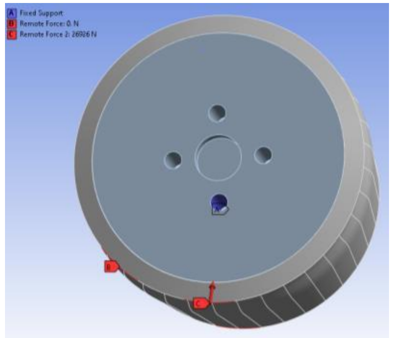


$$\begin{cases} \min\limits_{\Omega} \sum\limits_{k} compliance_k \\ mass \leq 35\% \\ \text{+4th y-cyclic} \\ \text{+1-sided y-pullout} \end{cases}$$

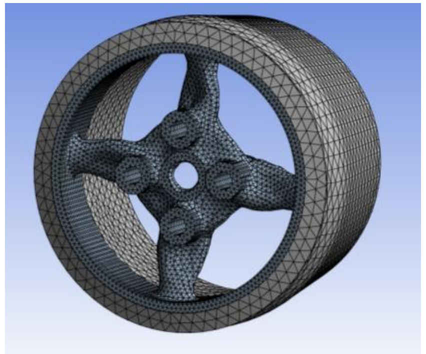


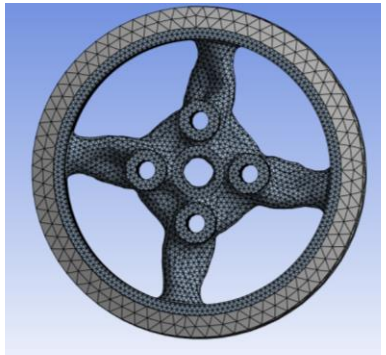


Mechanical setup
with 2 loadsteps



$$\begin{cases} \min\limits_{\Omega} \sum\limits_{k} compliance_k \\ mass \leq 35\% \\ \text{+4th y-cyclic} \end{cases}$$

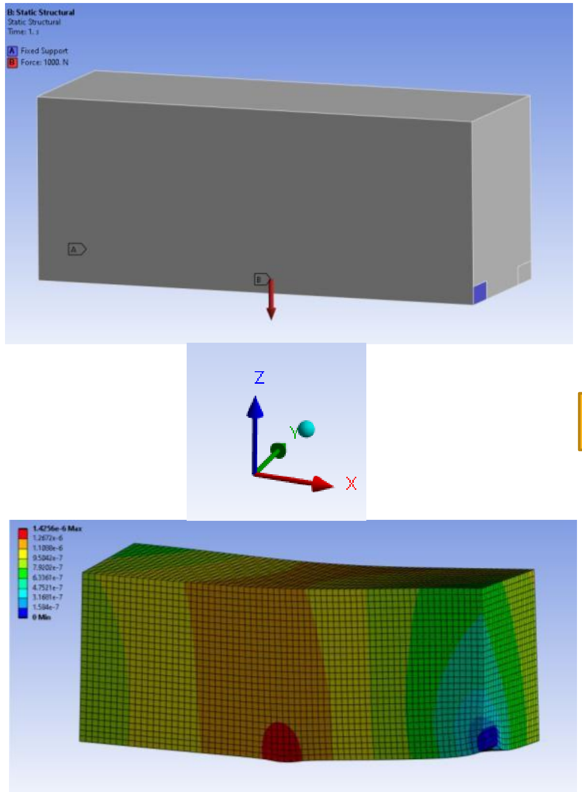




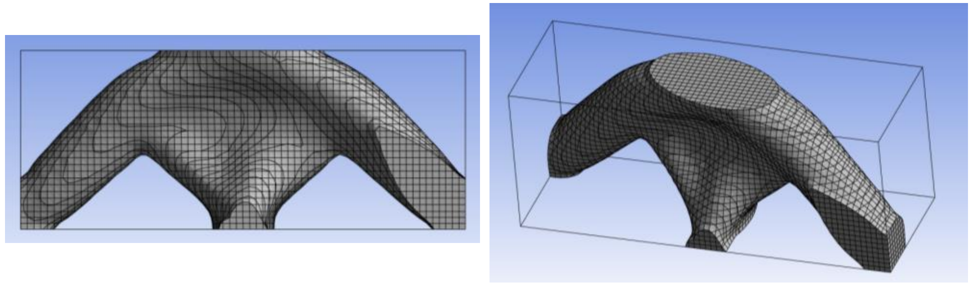
Details of "Design Constraint"	
Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Design Constraint
Subtype	Cyclic Repetition
Suppressed	No
Location and Orientation	
Number of Sectors	4
Coordinate System	Global Coordinate System
Axis	X Axis

CAD-IT CONSULTANTS

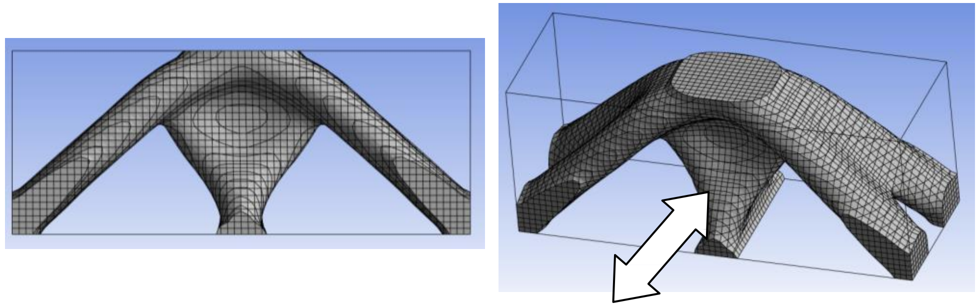
Academic test-case



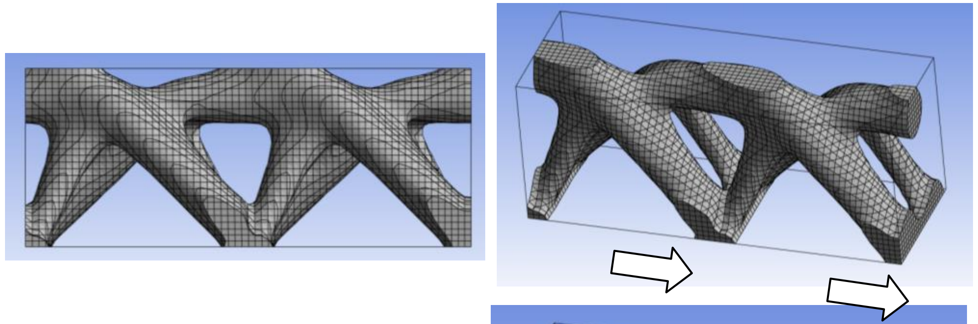
$$\begin{cases} \min\limits_{\Omega} compliance \\ mass \leq 25\% \end{cases}$$



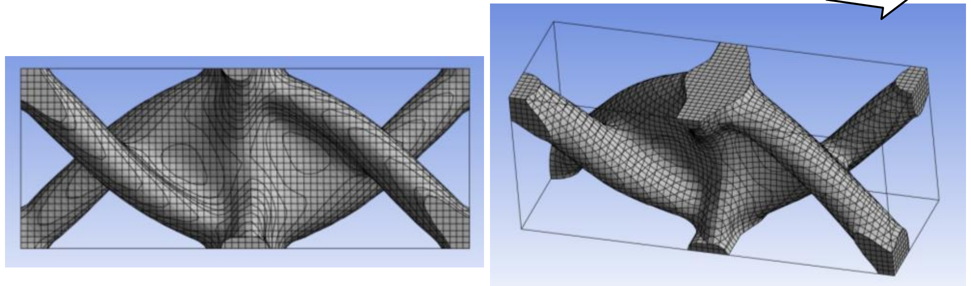
$$\begin{cases} \min\limits_{\Omega} compliance \\ mass \leq 25\% \\ \text{xz-plane symetry} \end{cases}$$



$$\begin{cases} \min\limits_{\Omega} compliance \\ mass \leq 25\% \\ \text{2nd x-pattern repetition} \end{cases}$$



$$\begin{cases} \min\limits_{\Omega} compliance \\ mass \leq 25\% \\ \text{2nd x-cyclic symetry} \end{cases}$$



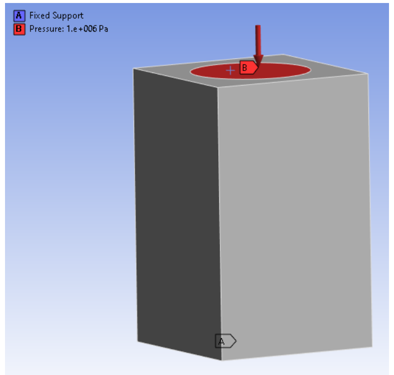
Min Gap

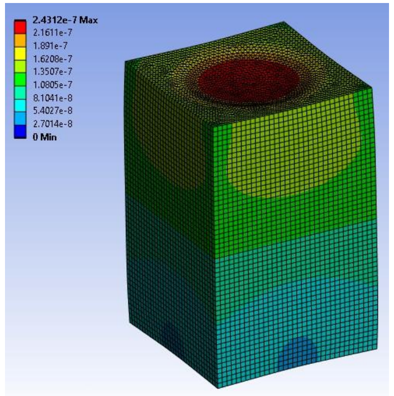
- This new manufacturing constraint aims to keep a minimum distance between features. It is motivated by the casting process to increase the life cycle of the mold.
- This constraint is qualitative. The formulation is based on an approximation that aims to limit the amount of material within multiple test regions. This capability should be used in combination with max-thickness.

Recommandation : $\text{GapSize} \leq 2.\text{maxThick}$

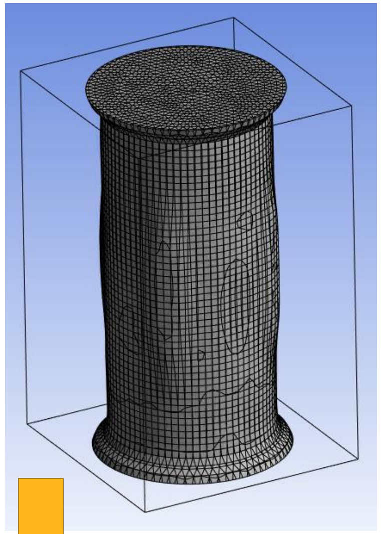
Details of "Manufacturing Constraint"

Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Manufacturing Constraint
Subtype	Member Size
Suppressed	No
Member Size	
Minimum	Free
Maximum	Manual
--Max Size	1.e-002 m
Gap Size	Manual
--Value	1.e-002 m

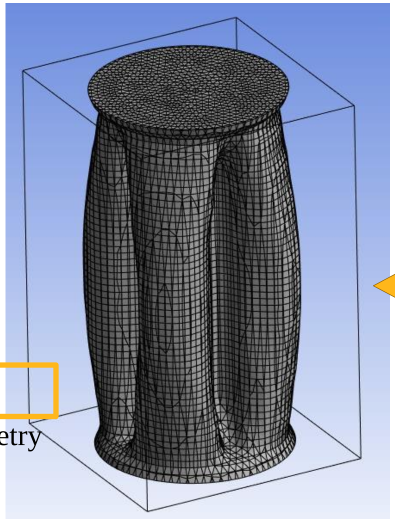




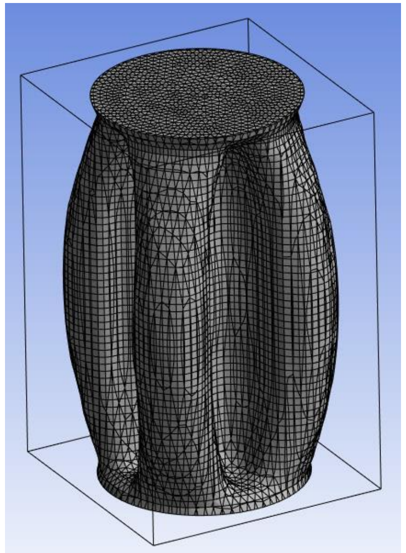
$$\begin{cases} \min_{\Omega} compliance \\ mass \leq 40\% \end{cases}$$



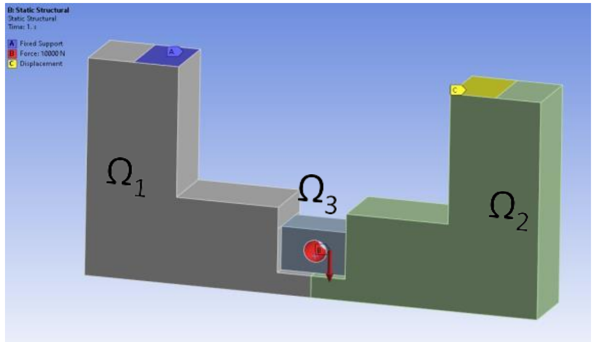
$$\begin{cases} \min_{\Omega} compliance \\ mass \leq 40\% \\ \boxed{\max\ T \leq 0.015m} \\ +4\text{th z-cyclic symetry} \end{cases}$$

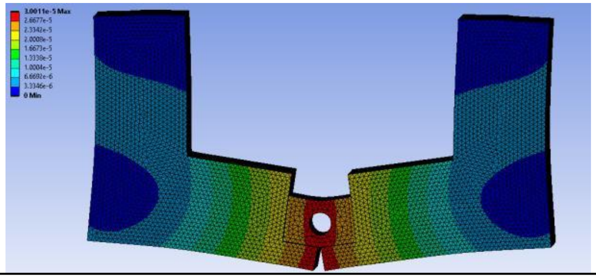


$$\begin{cases} \min_{\Omega} compliance \\ mass \leq 40\% \\ \max\ T \leq 0.015m \\ \boxed{\min\ Gap \geq 0.015m} \\ +4\text{th z-cyclic symetry} \end{cases}$$



Local Strain Energy



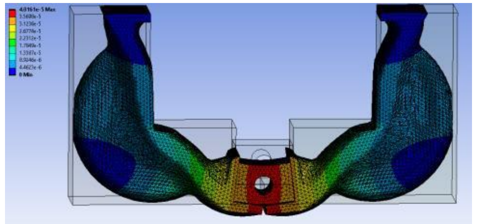


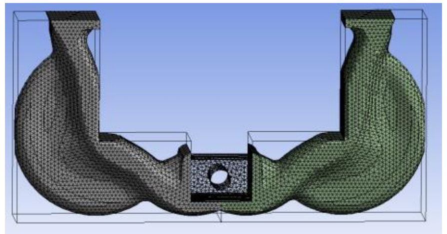
- LSE is exposed as a new stress-norm alongside vonMises or Maximum Principal Stress.
- In some context LSE is equivalent to compliance.
- LSE permits to control the strain-energy of the body of interest

Details of "Response Constraint 2"

Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region 2
Definition	
Type	Response Constraint
Response	Global Stress
Stress Type	Local Strain Energy
Maximum	15.
Environment Selection	All Static Structural
Suppressed	No

$$\begin{cases} \min\limits_{(\Omega_1,\Omega_2,\Omega_3)} \; complia \\ vol < \end{cases}$$

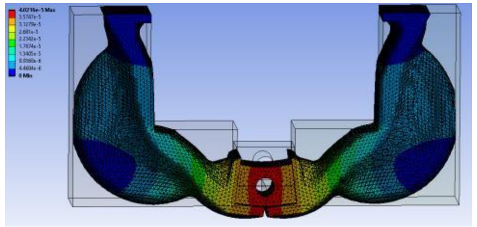


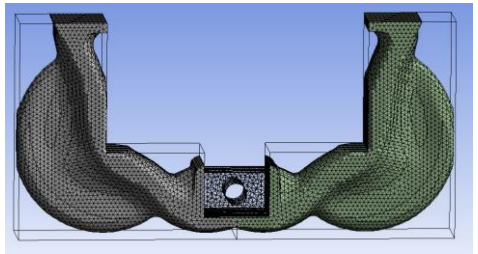


For some context, the compliance is the strain energy of the whole model …

=

$$\begin{cases} \min\limits_{(\Omega_1,\Omega_2,\Omega_3)} \; LSE \\ vol < 50\% \end{cases}$$

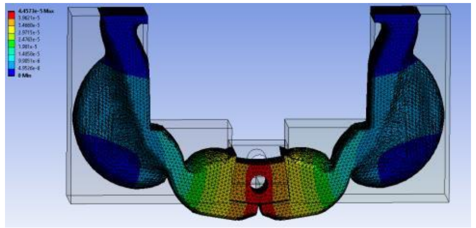


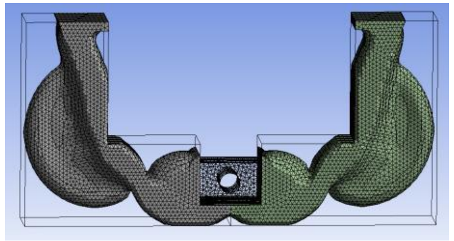


… so minimizing the total LSE (ie sum over all bodies) sometimes gives similar result to compliance.

≠

$$\begin{cases} \min\limits_{(\Omega_1,\Omega_2)} \; LSE \\ vol < 50\% \end{cases}$$





By contrast, minimizing partial LSE (eg sum over 2 out of 3 bodies) can return non-intuitive design.

Improvement & Corrections

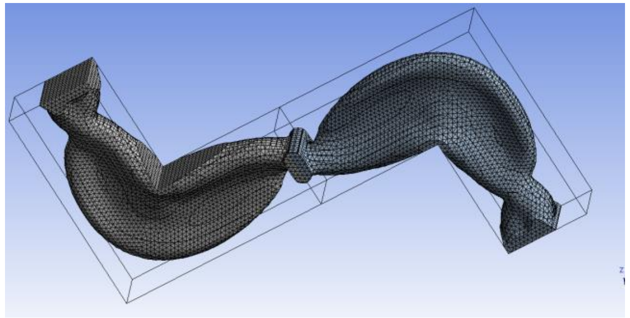
- Stress criterion
  - All elements lying in the Exclusion-Region are also excluded from the stress-control
  - A more accurate computation of the shape derivative has been implemented
  - Those changes may slightly affect the results with Level-set and Shape Optimization

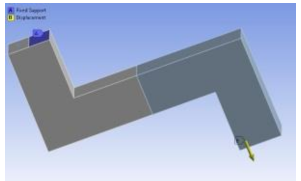
- UDC modal: « Robust Frequency »
  - Consolidation of the feature in order to better manage the context of multiple modes

- Max member-size
  - A new numerical scheme has been devised permiting to better access the maximum thickness in the context of Shape optimization and in Discovery Live

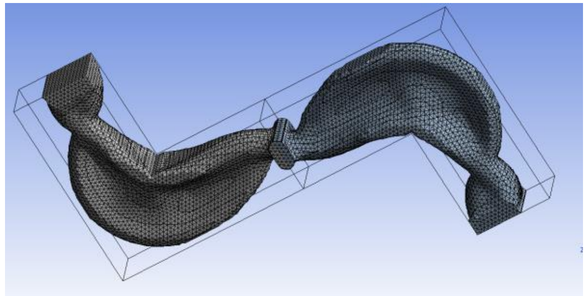
Stress in prescribed-disp context

$$\begin{cases} \min_{\Omega} vol \\ RF \geq 30kN \end{cases}$$

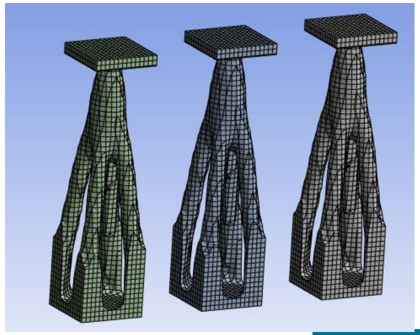




$$\begin{cases} \min_{\Omega} vol \\ RF \geq 30 \\ von \end{cases}$$



“Robust frequency” capability to manage multiple modes context



新科益工程仿真中心

咨询邮箱：ansyssupport@cadit.com.cn

公司网址：http://www.cadit.com.cn