Release 2022 R1 Highlights
Ansys Motion

Table of Contents

Ansys

Table of Contents

- **Ansys Mechanical Integration Enhancements:**
  - New Functions:
    - User Defined Mass
    - Contact Group Scoping
    - Function Previewer
    - Design Variable
    - Path Follower
  - Improvements:
    - Joint Friction
    - FMI 2.0
  - Postprocessor Improvements
    - Joint Probes
    - Custom Result
    - Nodal Averaged Stress and Strain

Ansys

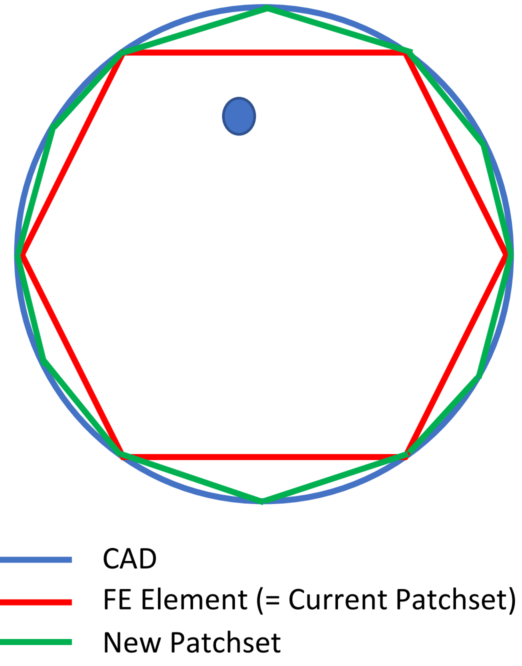
Ansys Motion Solver

Ansys Motion Solver

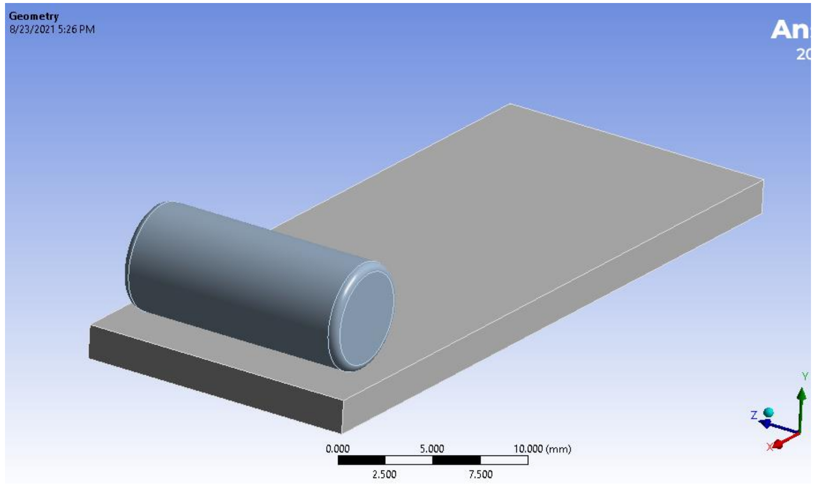
- New functions:
  - FE Body contact improvement (rigid body data)
  - Friction for Joints
- Performance improvements:
  - Solution improvement on FDM Jacobian
  - Super solver improvements
  - Hyperelastic material solution improvement
  - RBE3 memory improvement - (EasyFlex + FE) for eigenvalue
- Additional Improvements:
  - The possible location path - when solver can't find USUB DLL
  - Result name in simulation progress bar
  - CRG type 3D Road (F-TIRE Interface)
  - Frictional Heat --> USUB+Measure :: Solver BETA
  - External force entity Generalize

FE Body contact improvement (rigid body data)

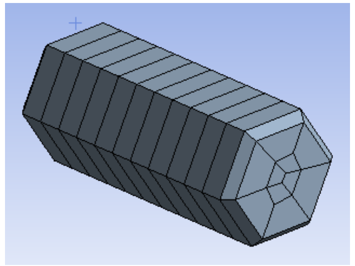
- Smoothest solution with Minimum patching
  - Coarse meshes (red) usually produce very coarse solutions with unwanted contact peak noise.
  - Contact geometry is decoupled from FE mesh data, reducing solution dependence on mesh quality.



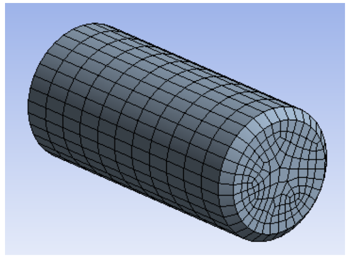
(Only Available in Workbench version)

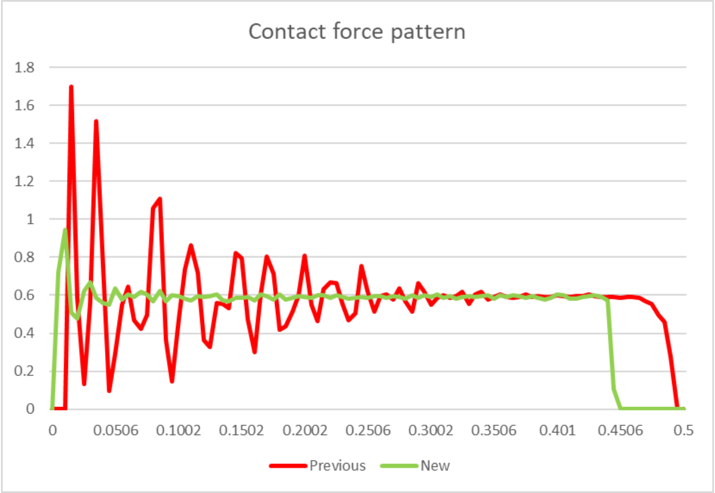


Real FE



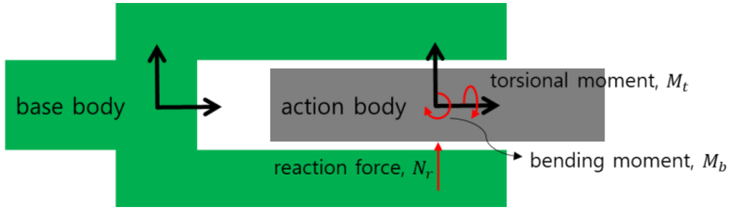
Only Patch

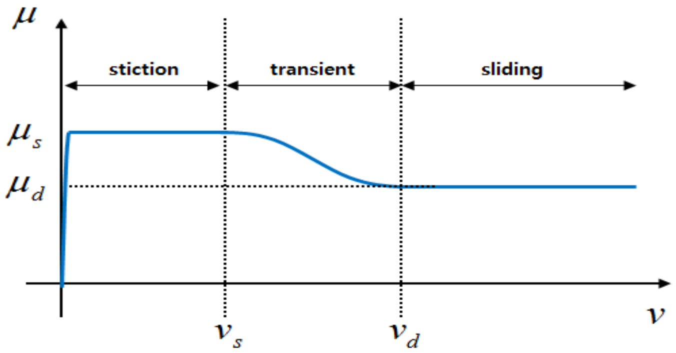




Friction for Joints

- Translational/Cylindrical joint friction
  - Joint Friction is mandatory and need to be expanded.
  - Basic Friction force calculation
    - Friction force : $f\_f=\mu \cdot N$
    - Friction coefficient (μ) : function of relative velocity
    - Normal force (N) : generated by reaction force and torque
  - Stiction characteristic also available

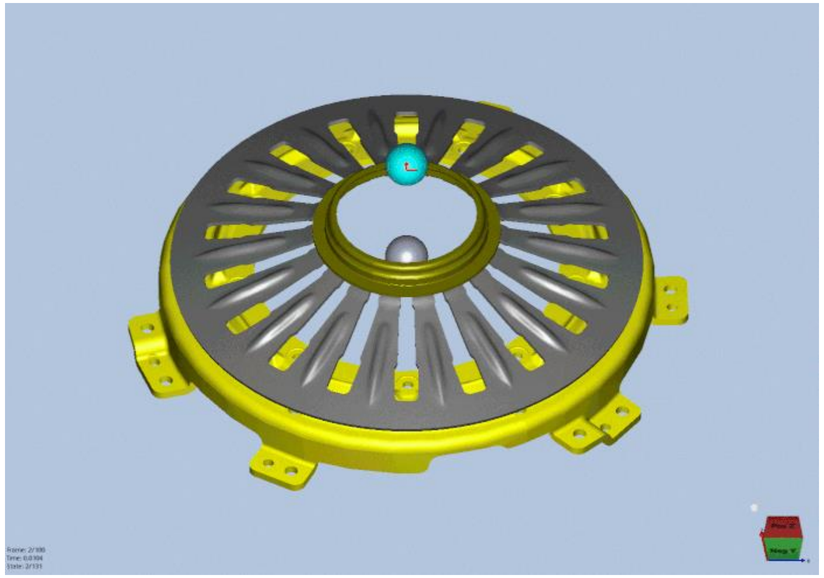


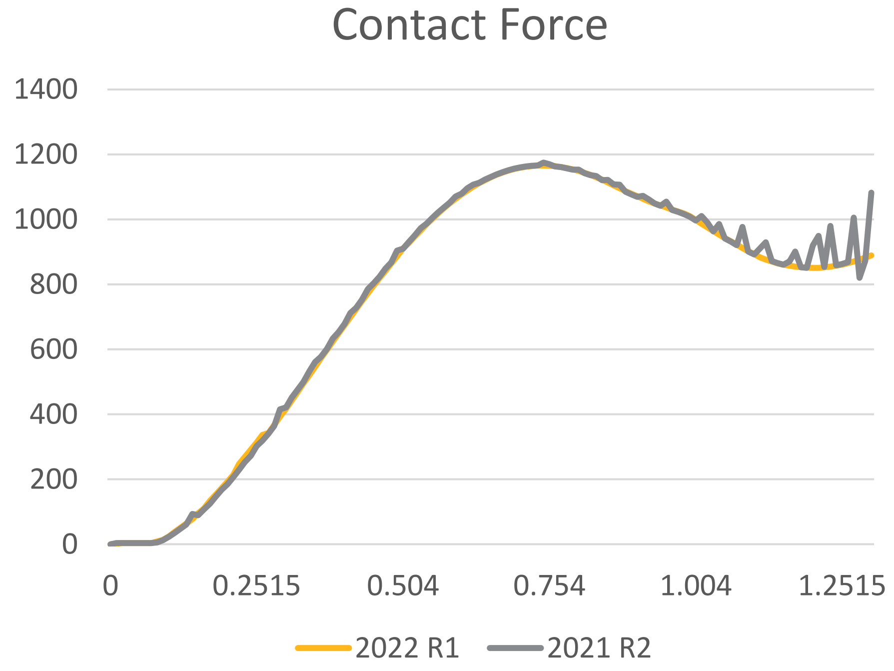


	Rev	Trans	Fixed	Ball	Cylind	Plane	Univ	Screw
~ 22R1	O	X	X	X	X	X	X	X
22 R1	O	O	X	X	O	X	X	X
22 R2	O	O	X	O	O	X	O	X

Solving Performance – FDM Jocobian

- Simulation Issue
  - When the FDM Jacobian flag is turned on, flex to rigid, flex to flex contact solution get much smoother.
  - The analysis time is almost the same.
  - The best performance improvement happens if a system has a floating flexible body,

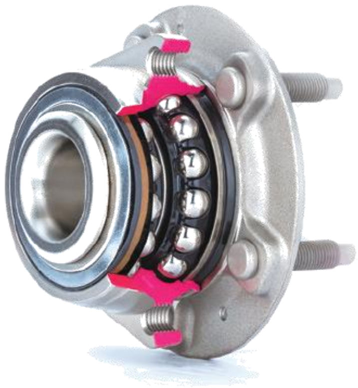




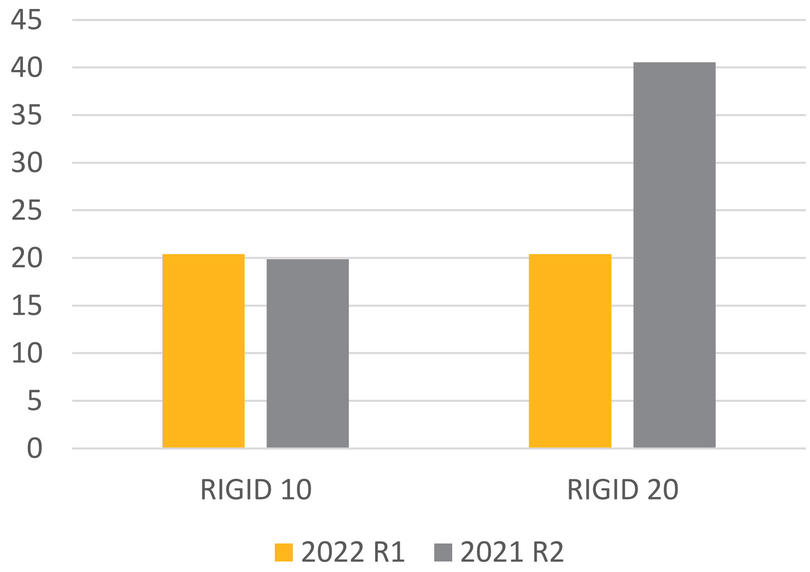
Solving Performance – Super solver symbolic improvement

- The number of factorization time.
  - In the Super solver, there was a problem that the symbolic process time increased a lot depending on the number of rigid bodies.
  - Improvement of graph analysis in symbolic process improves solving performance.

- No.DOF = about 702k
  - No.Node = 234,137
  - No.Rigid Body = 10 or 20

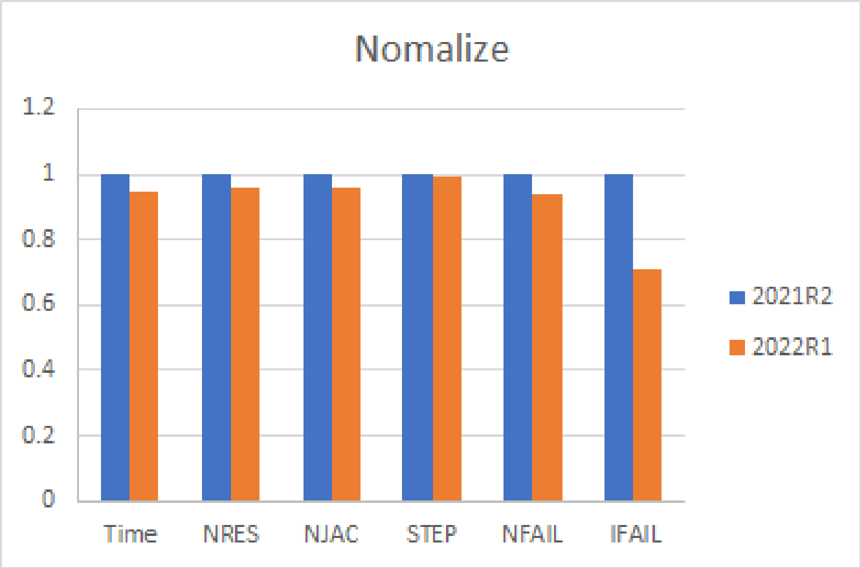


Courtesy of Iljin Global



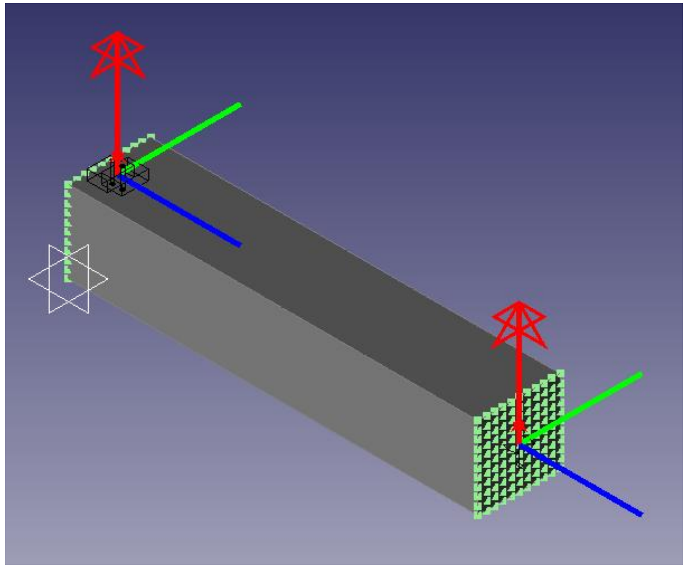
Improvement of step size control

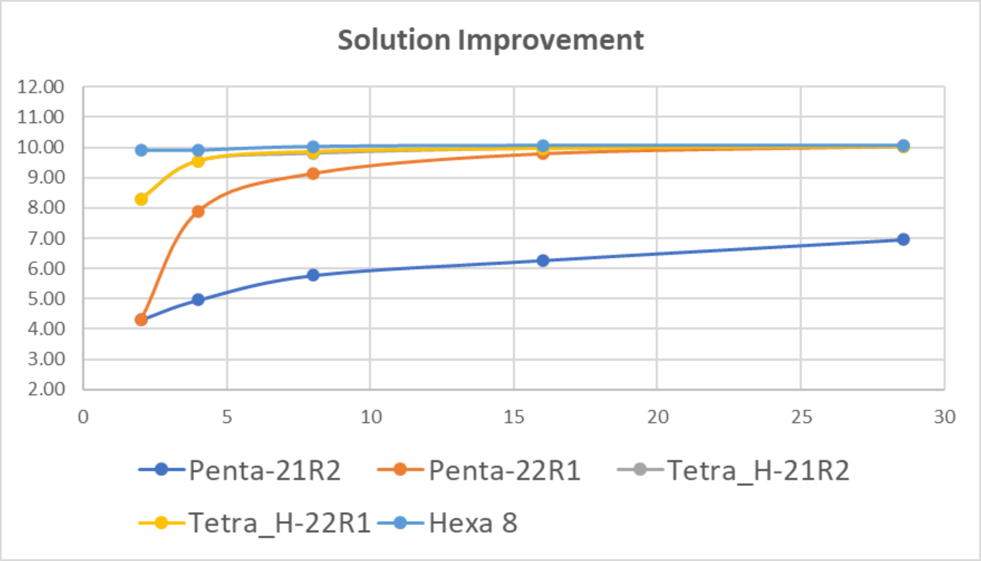
- Simulation time reduced
  - The time step size controller improved to not increase the value when the system frequently meet iteration failures.
  - The improvement saves an average of about 10% simulation time.
  - It only available for Super solver.

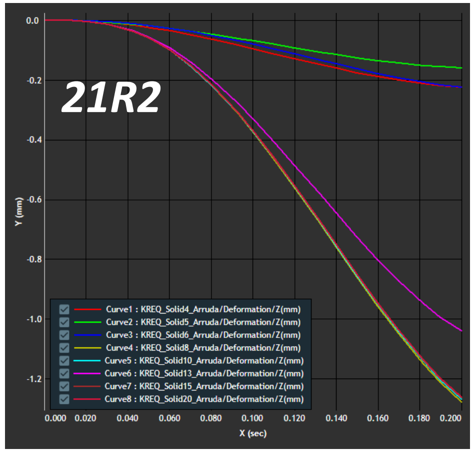


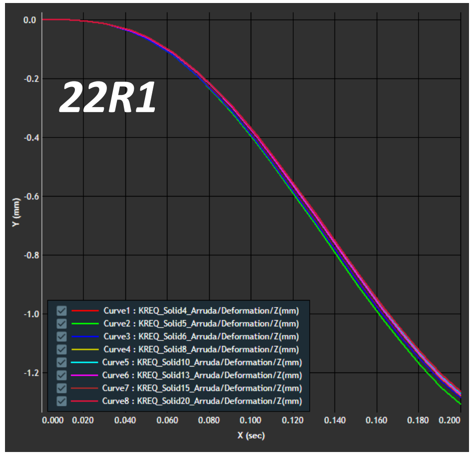
Hyperelastic material solution improvement

- Solution improvement
  - Solid4, solid5, solid6 and solid13 elements with hyperelastic material were too stiff and have been improved
  - $100 \times 20 \times 20\ mm^3$ / -0.05N in Z-direction









Ansys

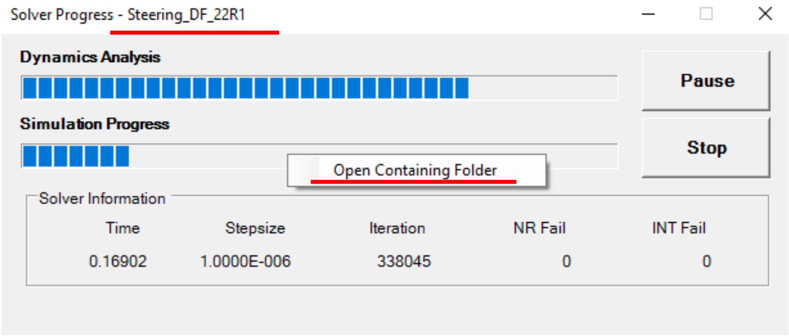
Additional improvements

- The message for the user-subroutine
  - Can't find a function/dll → More details given

```
* <ERROR> DFSOLVER:: CAN'T LOAD A DLL FILE IN USER-SUBROUTINE

* <ERROR> DFSOLVER:: FUNCTION USER SUBROUTINE ID = 0

* <ERROR> DFSOLVER:: dll_sample.dll

* <ERROR> DFSOLVER:: SIMTIME = 0.0000e+00
```

```
* <ERROR> DFSOLVER:: CAN'T LOAD A DLL FILE IN USER-SUBROUTINE
                     CHECK IF THE DLL EXISTS IN THE PATH BELOW
                     C:\Program Files\VirtualMotion, Inc\DAFUL 2022 R1\Solver\Windows\dll_sample.dll

* <ERROR> DFSOLVER:: FUNCTION USER SUBROUTINE ID = 0

* <ERROR> DFSOLVER:: SIMTIME = 0.0000e+00
```

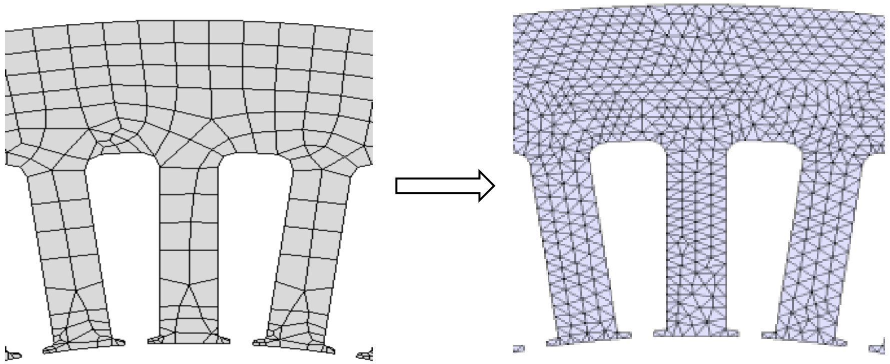
- Display result name and access button to result folder
  - There was no way to match the progress and simulation model.



Additional improvements

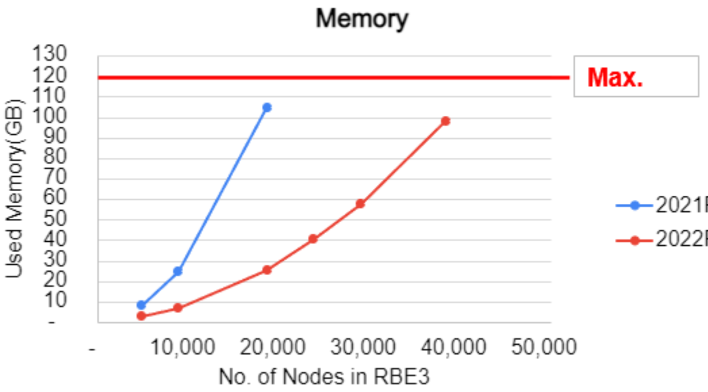
- FTire interface
  - The latest FTire interface is available.

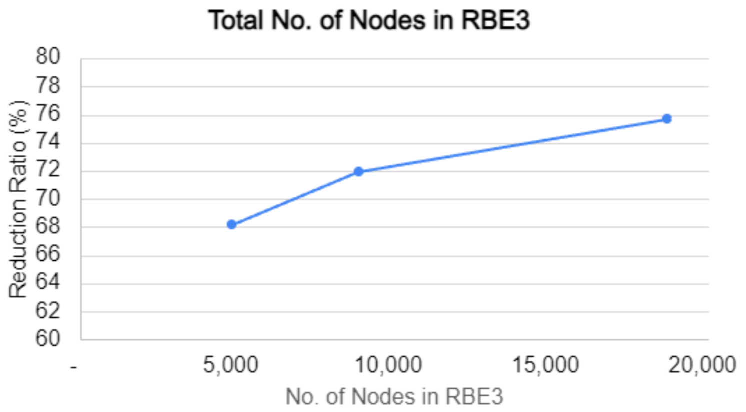
- External force
  - Open a way to use a nodal force calculated from different meshing configuration
  - Functions
    - EM force interface
    - EHD force
    - Durability (EasyFlex)



Additional improvements

- Memory usage improvement for RBE3
  - Penalty approach changed to constraint one and up to 40% memory has been less required for eigenvalue analysis.
  - It also gives more accurate solution and fast solving time.





Example data of Motor Housing

property	values
No, Node (RBE node)	0.7 (0.2) million
No. of normal modes	50

	~ 2021R2	2022R1
Memory usage (MB)	55957	38181 (32.8% ↓)
Solving time (s)	1738	1591 (8.5% ↓)
Solution comparison	Eigenvalue (Natural Frequency) -9.9348e+00 (0.0000e+00) -3.5816e+00 (0.0000e+00) -3.2749e-01 (0.0000e+00) -5.7474e-02 (0.0000e+00) 6.1180e+00 (3.9366e-01) 7.7119e+00 (4.4198e-01) 2.3558e+05 (7.7248e+01) 2.4932e+05 (7.9469e+01) 6.8299e+05 (1.3153e+02) 8.1770e+05 (1.4392e+02) 9.0305e+05 (1.5124e+02) 1.0054e+06 (1.5958e+02) 1.7934e+06 (2.1313e+02) 5.0072e+06 (3.5614e+02) 5.3235e+06 (3.6721e+02) 6.0059e+06 (3.9004e+02) 6.9118e+06 (4.1842e+02) 8.8540e+06 (4.7358e+02) 9.4867e+06 (4.9020e+02)	Eigenvalue (Natural Frequency) -3.0096e-01 (0.0000e+00) -1.1097e-01 (0.0000e+00) -2.3536e-02 (0.0000e+00) -1.7305e-02 (0.0000e+00) 2.5158e-03 (7.9829e-03) 3.4351e-03 (9.3281e-03) 2.3559e+05 (7.7249e+01) 2.4933e+05 (7.9470e+01) 6.8299e+05 (1.3153e+02) 8.1770e+05 (1.4392e+02) 9.0305e+05 (1.5124e+02) 1.0054e+06 (1.5958e+02) 1.7934e+06 (2.1313e+02) 5.0072e+06 (3.5614e+02) 5.3235e+06 (3.6721e+02) 6.0059e+06 (3.9004e+02) 6.9118e+06 (4.1842e+02) 8.8540e+06 (4.7358e+02) 9.4867e+06 (4.9020e+02) Difference: less than 0.1%

Ansys

Standalone

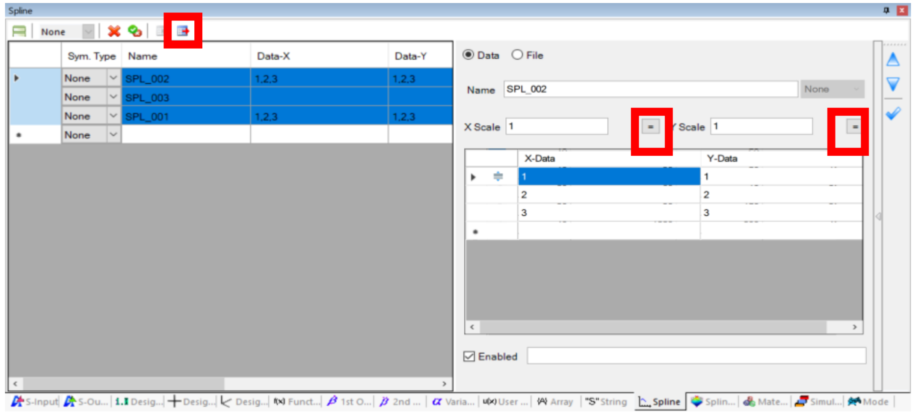
Ansys

Standalone Enhancements

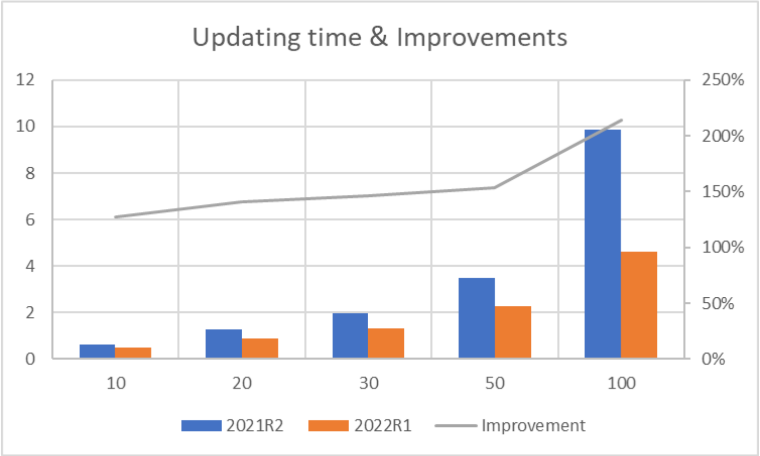
- Pre/Postprocessor Improvement:
  - Spline Upgrade(Scale Parameterization, File Export)
  - Subsystem navigator performance
  - Postprocessor Script Environment
  - Sound Pressure with DBA, DBB, DBC, DBD
  - Min/Max Display in Contour
  - Export Ensight
  - Event Driven Item Selection in Action Panel
  - Deformation Scaling
  - Default Scale for Mode shape Animation
  - F-Tire interface
- Drivetrain:
  - DT Auto-Modeler Upgrade
  - Motor Designer Upgrade
  - Tooth stiffness calculation – automation
  - Default value for the "Number of slices“
  - Bearing component improvement – stiffness
  - DOE Support of Drivetrain DV
- Links:
  - New Link Assembly (Open loop, Rail) - SDI

Preprocessor Improvement

- Spline Upgrade
  - The scale factor for each axis value of spline(2D/3D) can be parameterized with DV(design variable)
  - A group of splines can be exported to one file. One of the spline can be selected once it imported with File option.

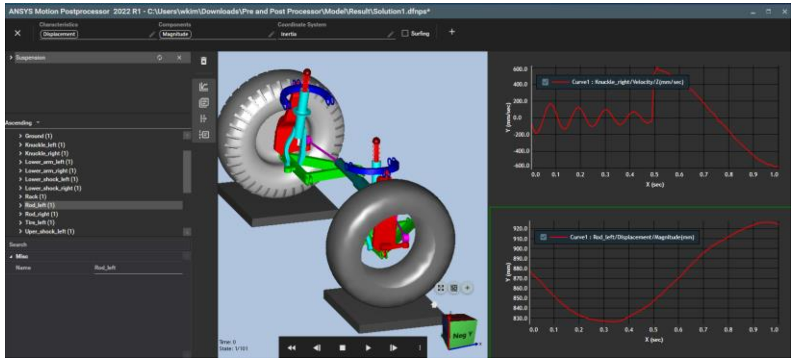


- Subsystem navigator performance
  - The performance of updating subsystem navigator data has been improved.
  - The performance is also available on API with flag.

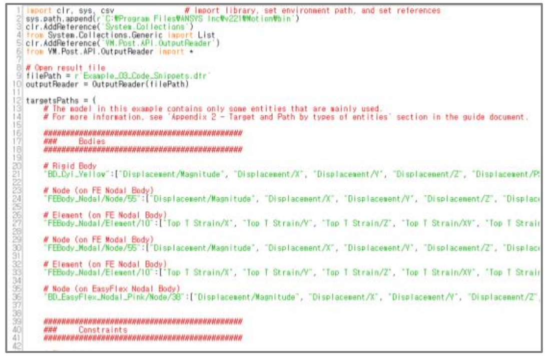


Postprocessor Script Environment

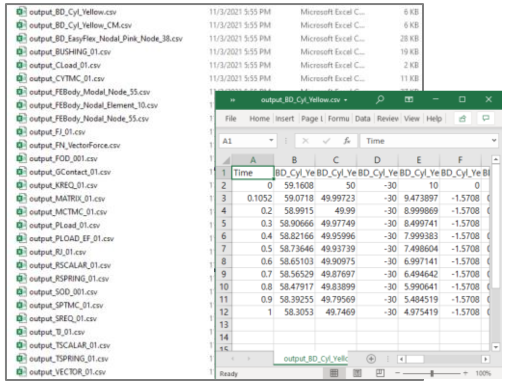
- Postprocessor API (Python Script)
  - Postprocessor API allows to extract plot data from the result files of Ansys Motion.
  - Postprocessor API can be used for projects such as
    - Build automated processes
    - Integration with optimal design tools
- Requirements
  - Operation System : Windows
  - Python version : 2.7.x, 3.5.x, 3.6.x, 3.7.x and 3.8.x



Ansys Motion Result



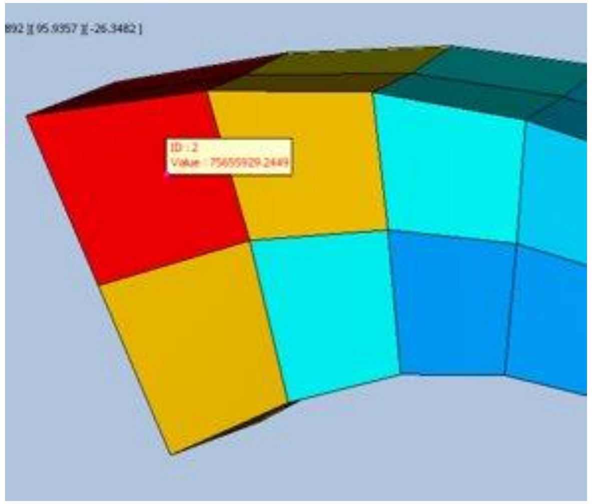
Postprocessor API (Python Script)

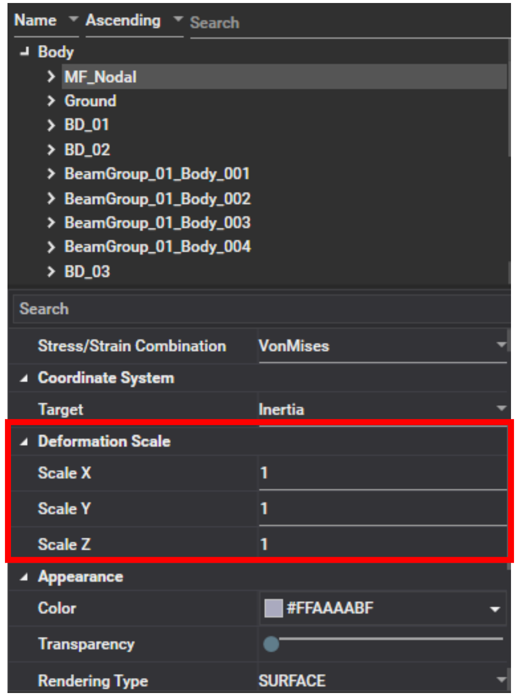


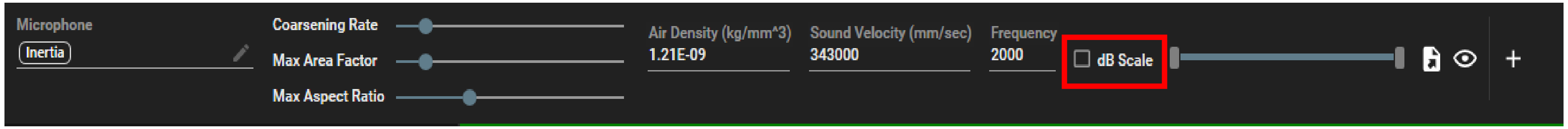
Postprocessor API Result

Postprocessor Improvements

- Min/Max display in Contour
- Export to EnSight Format
- Item Selection for Charting Improvements
  - Preserving items conditionally as updating selected items on Action Panel
  - Surfing
- Deformation Scaling
- Default Scale for Modeshape Animation
- Decibel Scale on Sound Pressure Analysis

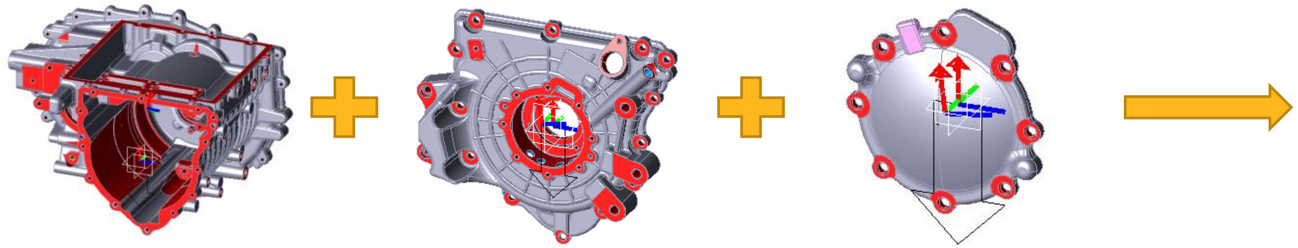






Multiple solid bodies for DT modeler

- Convert multiple geometries at once
  - Converting well developed CAD geometry automatically into matching DT entity is highly useful but it only allows single geometry to be converted.
  - The most of models are composed of multiple solid
  - New version allows converting multiple geometries to one DT entity for Shaft, Housing

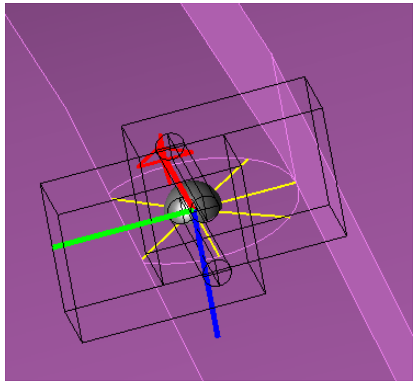


One Entity
(part file)

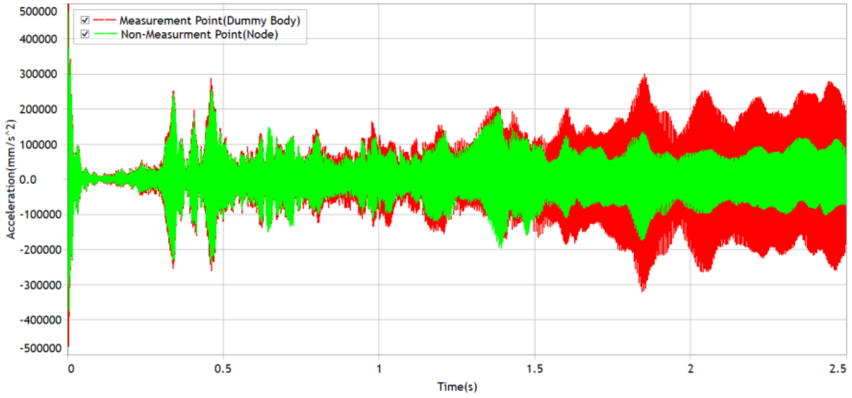
	2021 R2	2022 R1
EF Housing	“Housing Type”__”Housing Name”	“Housing Type”__”Housing Name”__”Solid Name”
EF Shaft	“Shaft Type”__”Housing Name”	“Shaft Type”__”Housing Name”__”Solid Name”

Measurement points for Motor designer

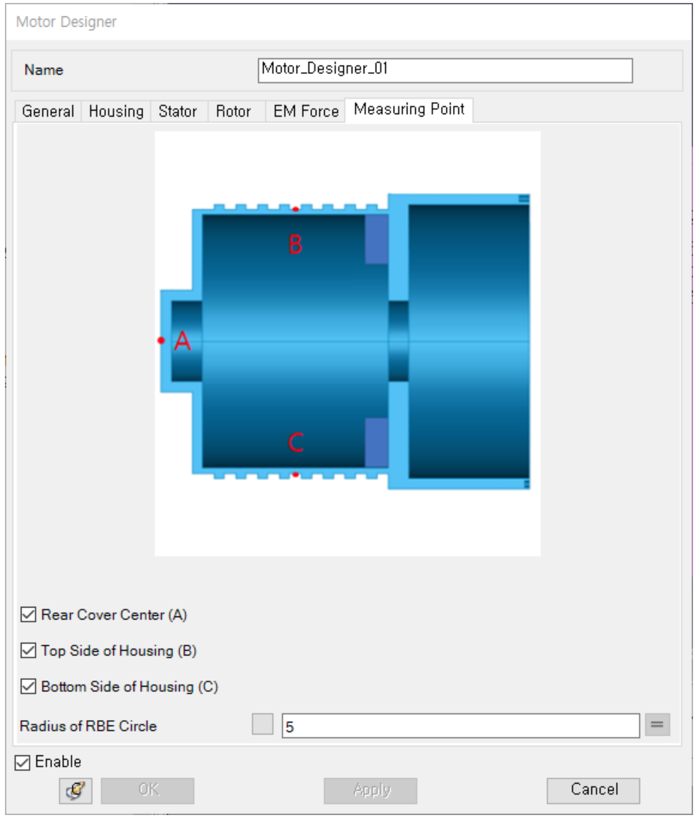
- Report points for Motor
  - Three report options are added for convenient monitoring
  - Once the point selected, collection mode calculation carried out automatically in order to increase solution accuracy.
  - It helps you to get result more easily with high accuracy.



<Fix joint (RBE-Dummy)>

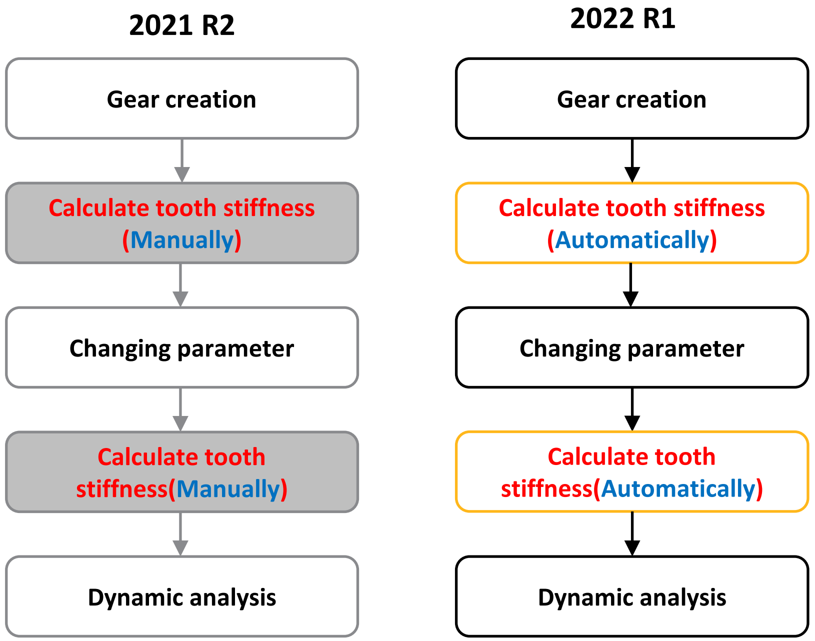


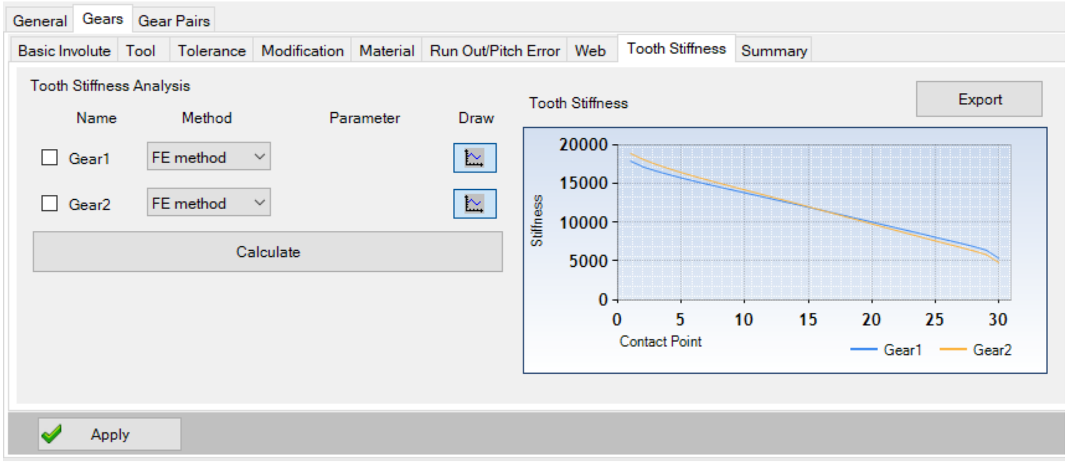
<Acceleration (Additional RBE vs Non additional RBE)>



Tooth stiffness calculation

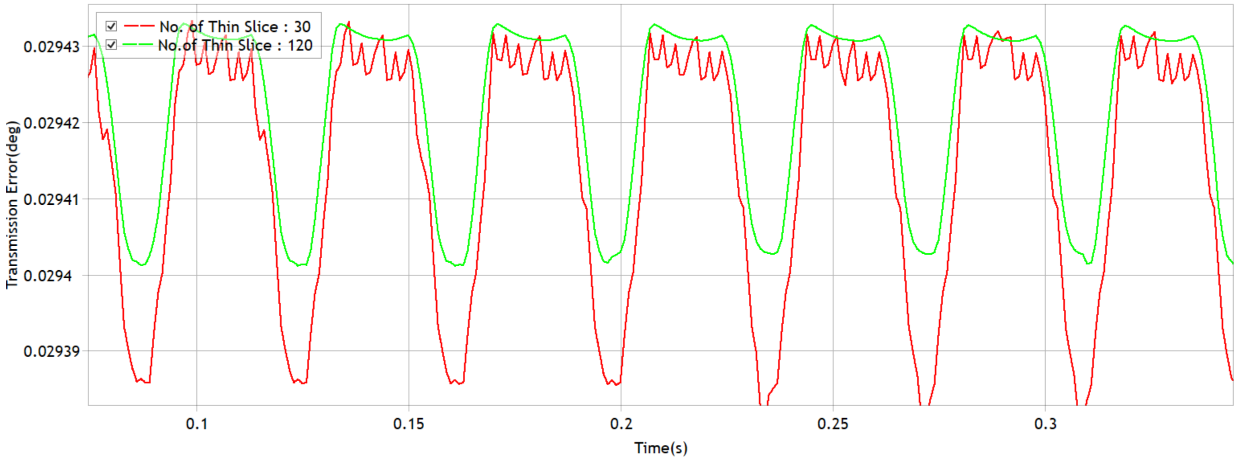
- Tooth stiffness calculation automatically
  - The tooth stiffness calculation must be recalculated when a gear specification that has highly contributed is changed.
  - New version updates the value when it detect changes of major specification.



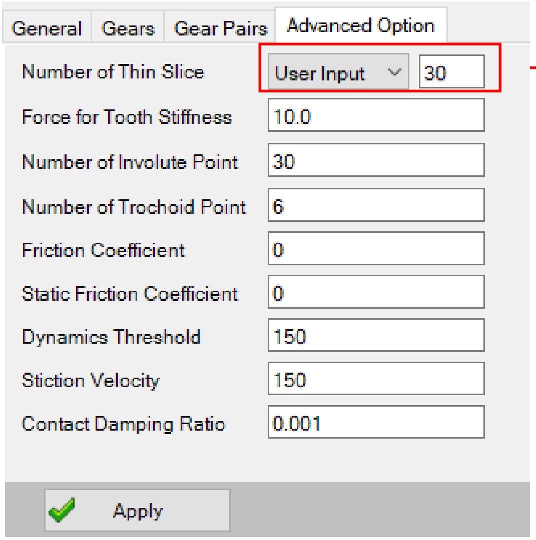


Changing default value of “No. of thin slice”

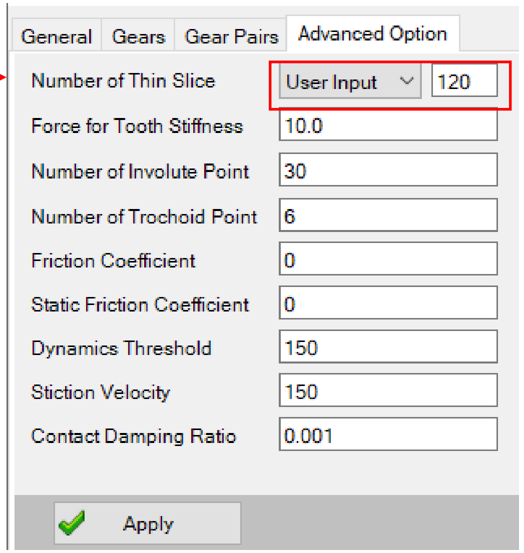
- The number of thin slices
  - The default value is changed from 30 to 120 to decrease gear contact noise.
  - Without the value, helical type gear can’t provide reasonable solutions.



<Transmission Error>



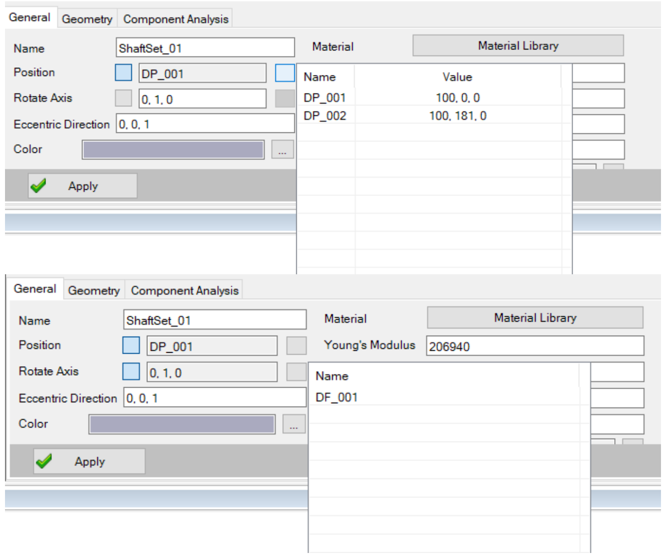
<2021 R2>



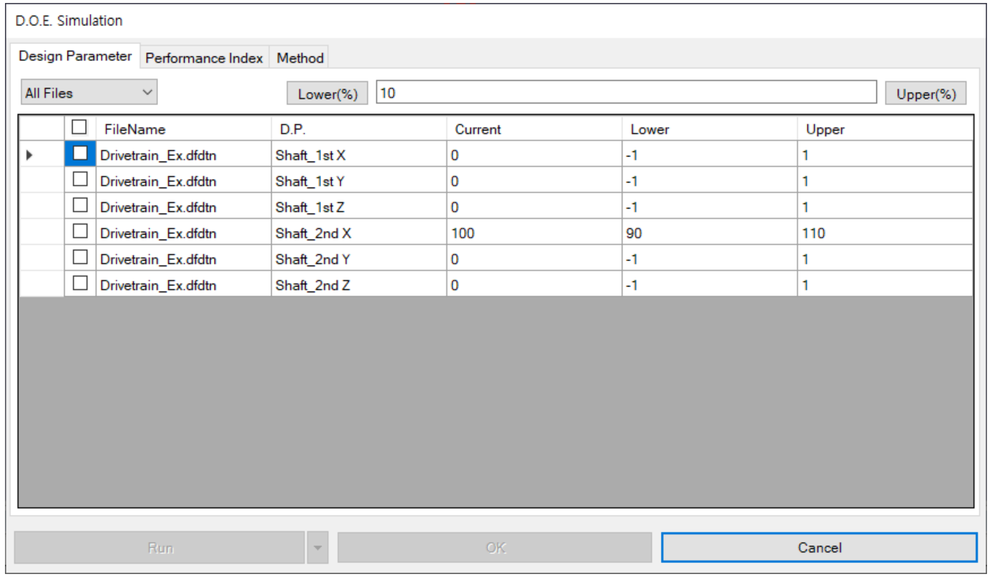
<2022 R1>

D.O.E support of Drivetrain DV

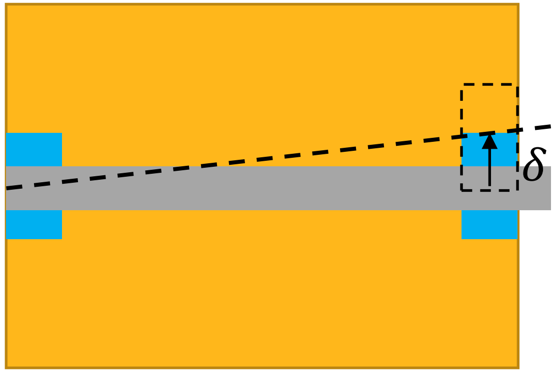
- DOE Target value is added
  - The starting position of shaft and its direction can be parametrized. Design variable and frame can be connected to them
  - These parameters are shown in D.O.E simulation dialog and it enables you to do design study with offset, mis-alignment conditions of shaft.



<Set design parameters in shaft set designer>



<Configuration of D.O.E>

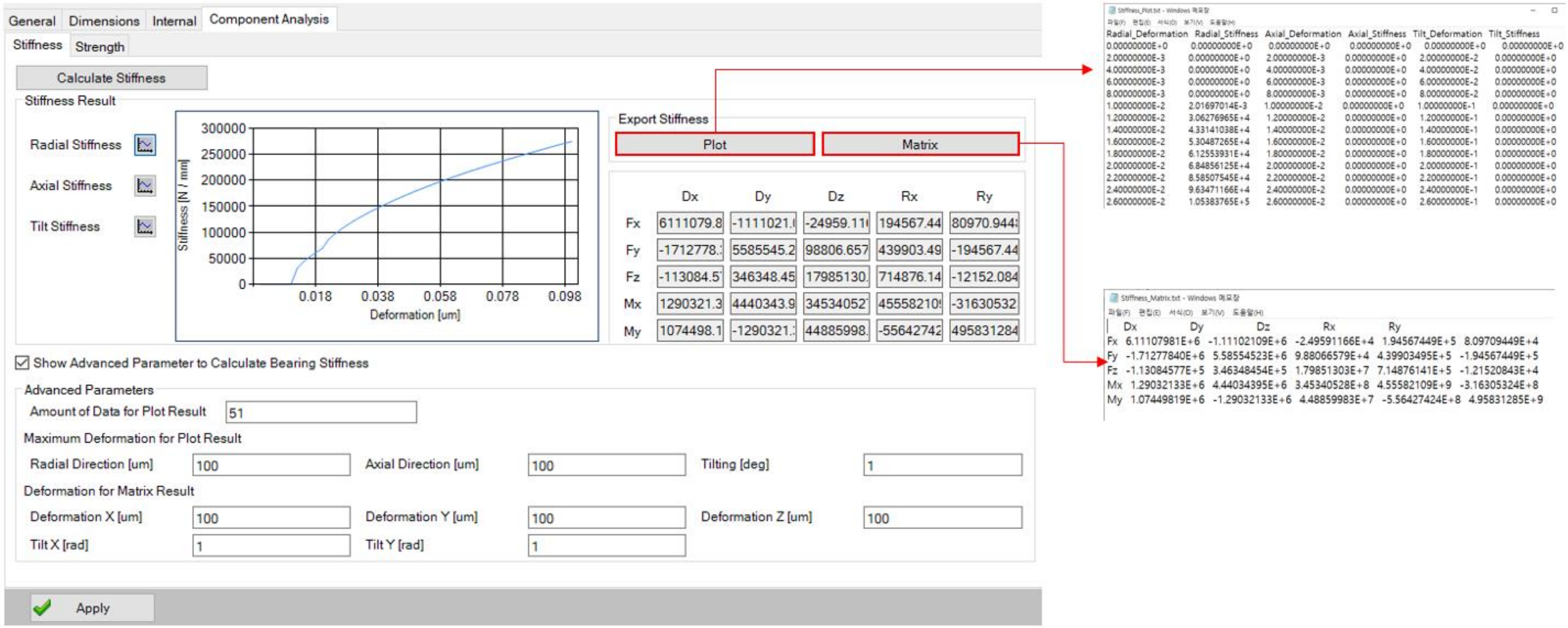


<Mis-Alignment>

Ansys

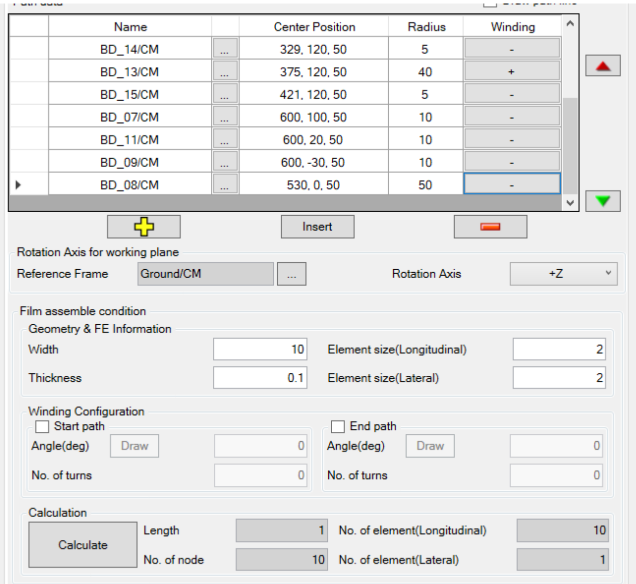
Bearing Stiffness (Component Analysis)

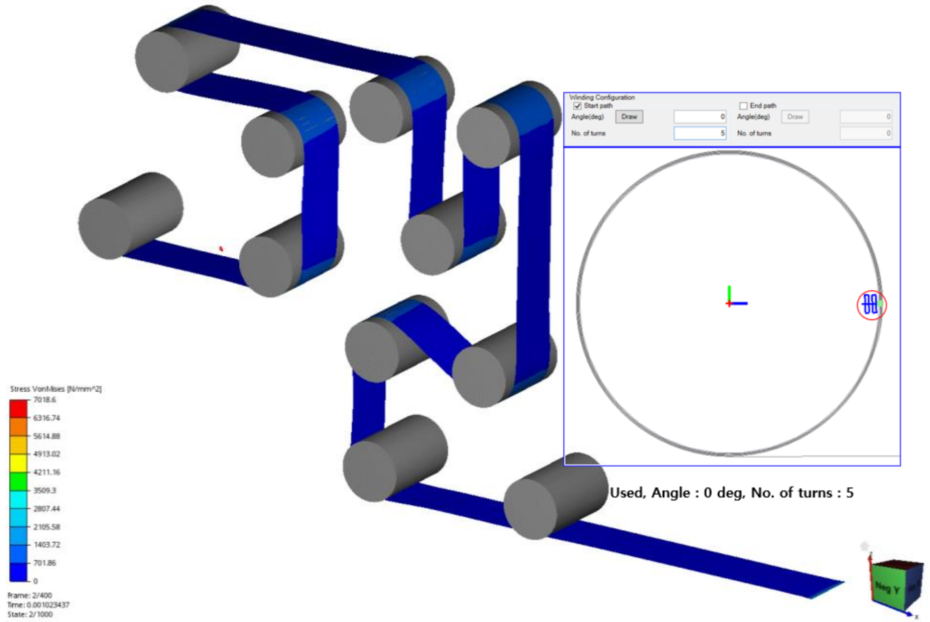
- Bearing Stiffness Calculation
  - The calculation method of bearing stiffness changed more accurately.
  - The stiffness matrix can be extracted.



New Links assembly – Thin Film

- Roll simulation with thin film
  - Making an initial assembly considering of deformed shape for a flexible body usually requires pre-simulation to derive it.
  - Also requires long simulation time when the model has a complicate assemble path.
  - FE Film assembly enables you to do that quite easily by removing the deformation derive simulation
  - Selecting rollers and setting some specifications are all you need to do. Followings are automation part
    - Mesh for thin plate
    - Deformed initial position
    - Self-contact between thin plate
    - Contacts between rollers and thin plate





Ansys/Motion – FTire Interface

- FTire is a highly accurate physics-based, 3D nonlinear tire simulation model for all kinds of vehicles. It is the leading tire model for handling, ride, durability and NVH analysis.
- Ansys Motion uses Cosin's tire interface(CTI), a C/C++ API. CTI provides a time-discrete generalized interface. All interfaces are designed to run an arbitrarily large number of tire instances simultaneously.
- The coupling to the vehicle or suspension model of the calling program is done by rigid body state variables of the rim, that is:
  - position of the rim center in the global frame
  - translational velocity vector of the rim center
  - angular orientation of the rim, defined by the transformation matrix from the rim fixed frame to global frame
  - angular velocity vector of the rim

  FTire return forces and torques acting on the rim center, represented in the global coordinate system.

- When calling FTire from within Ansys/Motion, there is no need to know the meaning of these variables. They are automatically extracted from, or transferred to, the vehicle model by Ansys Motion.

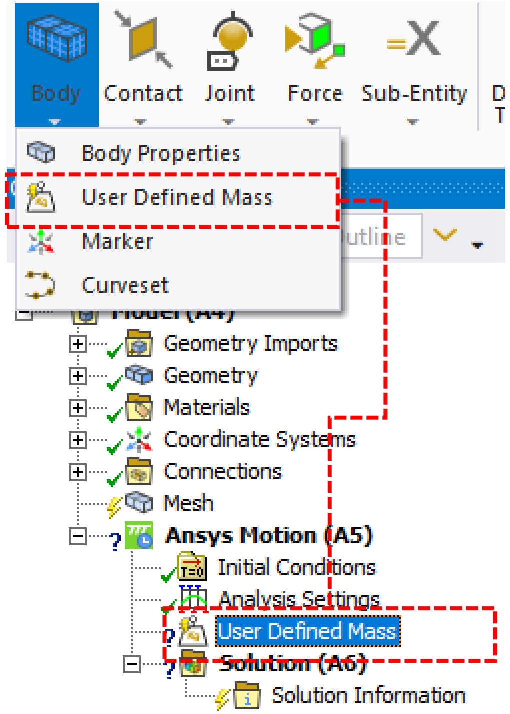
Ansys Mechanical Integration Enhancements

Ansys Mechanical Integration Enhancements

- New Functions:
  - User Defined Mass
  - Contact Group Scoping
  - Function Previewer
  - Design Variable
  - Path Follower
- Improvements:
  - Joint Friction
  - FMI 2.0
- Postprocessor Improvements
  - Joint Probes
  - Custom Result
  - Nodal Averaged Stress and Strain

User Defined Mass

- Change Mass/Inertia property
  - A common practice in rigid dynamics analysis is to replace a complex geometry by a simpler one, but it is important to use the right geometry properties, thus User Defined Mass object allows to modify the Mass, Moments of Inertia and centroid of selected rigid bodies.

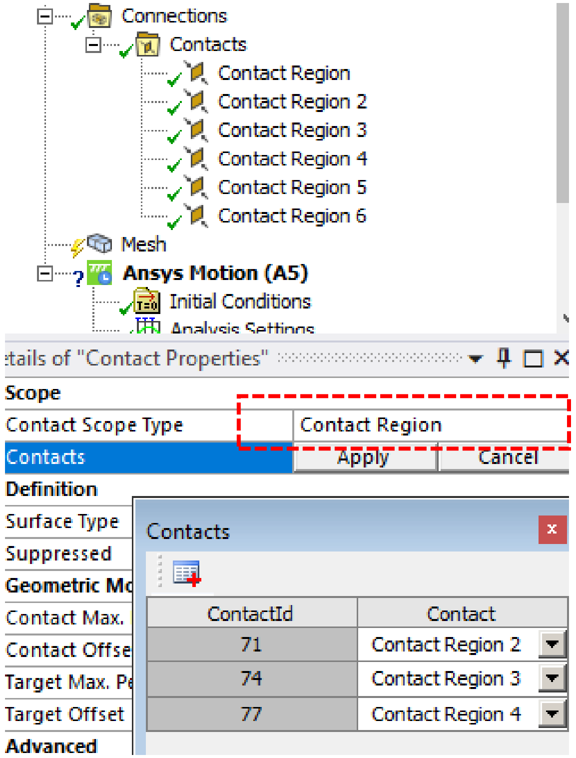


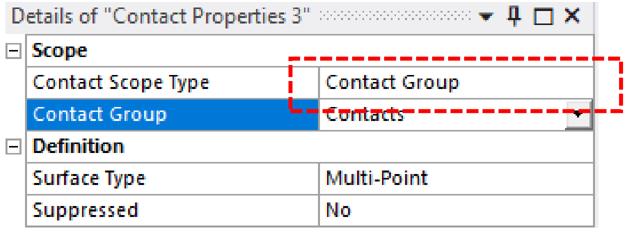
Details of "User Defined Mass"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Mass	9.8125 kg
Moment of Inertia Ip1	0.408854117927411 kg·m²
Moment of Inertia Ip2	0.408854069188203 kg·m²
Moment of Inertia Ip3	0.408854069188204 kg·m²
Centroid	Unchanged

Contact Group Scoping

- Improvement in selection Method
  - Contact Properties, Contact Friction Properties and Tie Properties objects are used to modify contact settings. For important number of contact regions, it was cumbersome to select all desired contacts to modify, thus now it is possible to modify all contacts of a given Connection Group improve a lot user friendliness



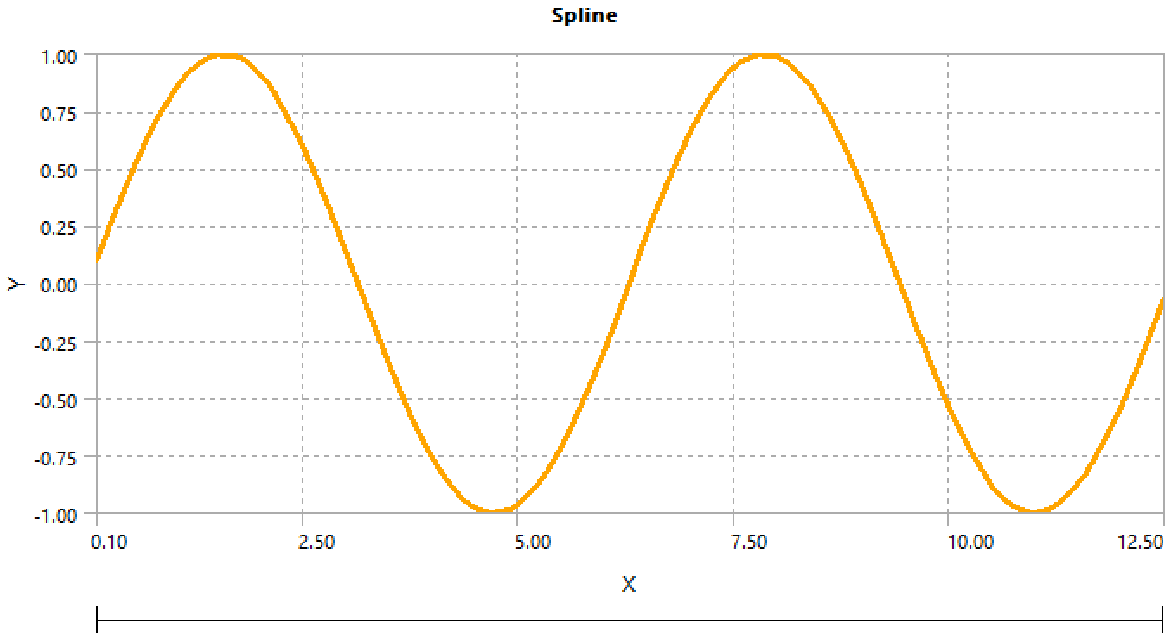


New – Contact group

Previous - Contact Region with tabular data

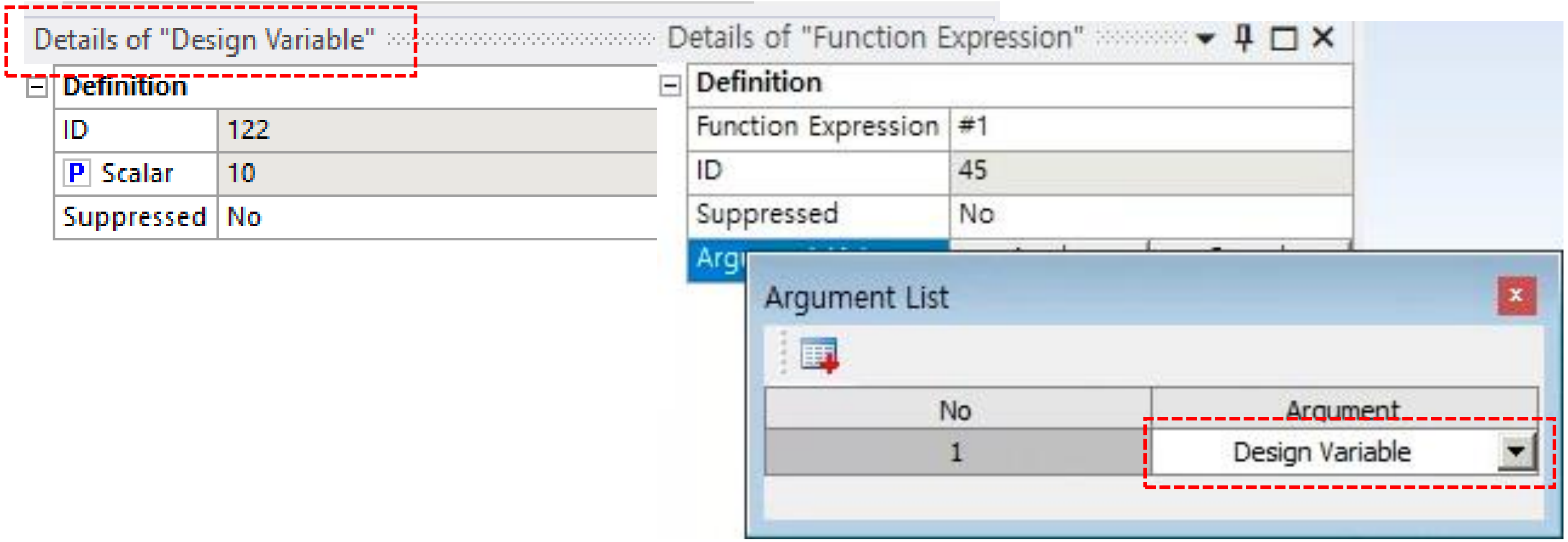
Function Previewer

- Preview function expression
  - Evaluation of function expression expressed by Mathematical(sin, cos, tan and so on), Logical (If) and several constant value are available.
  - Integration variables are not available and will be extended for STEP, LININT in next version.



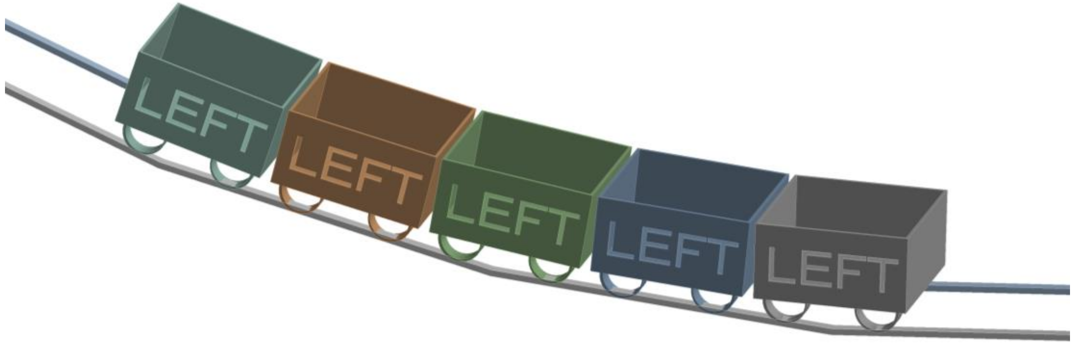
Design Variable

- Parametric study with design variable
  - A Design Variable(DV) object can be assigned to “Parameter Set”.
  - The DV can be used within Function Expression, and it enables to perform design experiments with complex loading conditions.



Links - Path Follower

- A new way to assemble segment
  - Links toolkit enables you to assemble segment along with imaginary line passing “Path”. On the other hands, A Path Follower function enables you to place segments on the predefined Curvesets.
  - It can be useful to build a roller coaster or train applications for placing cabin on the rails.



Additional Improvements

- Extended Joint Friction
  - Joint Friction Properties has been extended to Translational and Cylindrical joints.
  - Transition Velocity coefficient is now available.

Details of "Joint Friction Properties 2"

Scope	
Joint	Joint 2 [Translational]
Definition	
Static Friction Coefficient	0.5
Dynamic Friction Coefficient	0.3
Stiction Transition Velocity	0.0001 m/s
Transition Velocity Coefficient	1.5
Max Stiction Deformation	1E-05 m
Reaction Force	Use
Bending Moment	Use
Torsional Moment	Use
Friction Effect	Sliding And Stiction
Reaction Arm	0.001 m
Overlap Option	Constant
Initial Overlap	0.001 m
Pre Force	0 N

- FMI 2.0
  - FMI 2.0 is available. Interface Time Step and Model description are available.

Details of "Co-Simulator"

Definition	
Interface Type	FMI
FMI Version	2.0
Interface Time Step	0.01 s
Model Description	pendulum
Signal Input	Tabular Data
Signal Output	Tabular Data
Suppressed	No

Postprocessing Improvements

- Custom Result
  - Custom Result object allows you to evaluate output of Function expression and User Subroutine
  - Result values can also be used as output parameter.

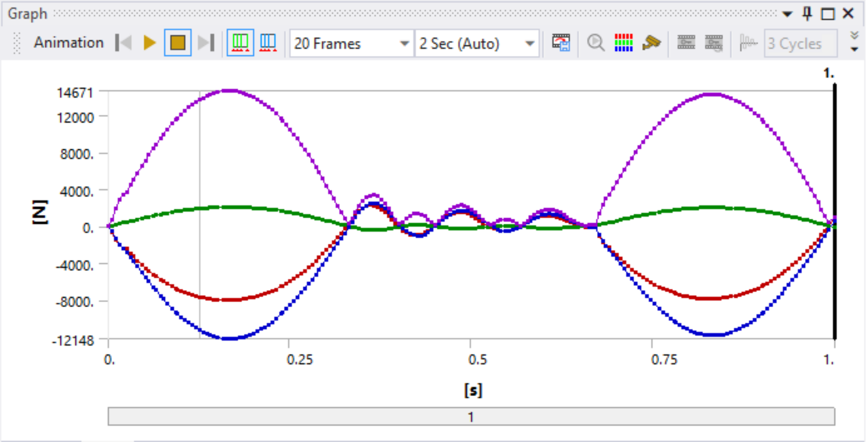
Details of "Custom Result"

Definition	
Type	Function Expression
Function Expression	output_Knuckle_right_dz
Unit Name	Length
Suppressed	No
Results	
Result	0.150153694 m
Maximum Value Over Time	
Result	0.242967413 m
Minimum Value Over Time	
Result	0.142779276 m

- Joint Probes
  - Joint Probe results can now be used to analyze joint results.

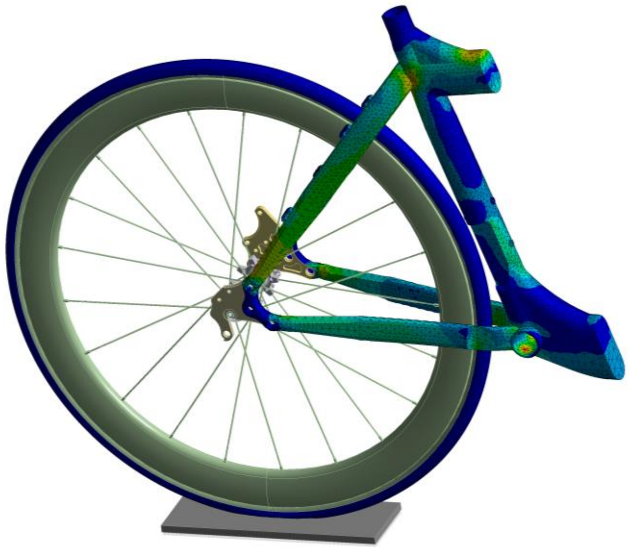
Details of "Joint Probe"

Definition	
Type	Joint Probe
Boundary Condition	BUSHING_05
Orientation Method	Joint Reference System
Suppressed	No
Options	
Result Type	Total Force
Result Selection	All
Display Time	End Time
Results	
Maximum Value Over Time	
X Axis	2295.6 N
Y Axis	2091.3 N
Z Axis	2485.4 N
Total	14671 N
Minimum Value Over Time	
X Axis	-7961.4 N
Y Axis	-393.5 N
Z Axis	-12148 N
Total	0.11785 N



Postprocessing Improvements

- Nodal Averaged Stress & Strain
  - Stresses and strains are now evaluated either at the nodes or at the element
  - Thus, now Mechanical postprocessing is consistent with standalone Postprocessor



Ansys