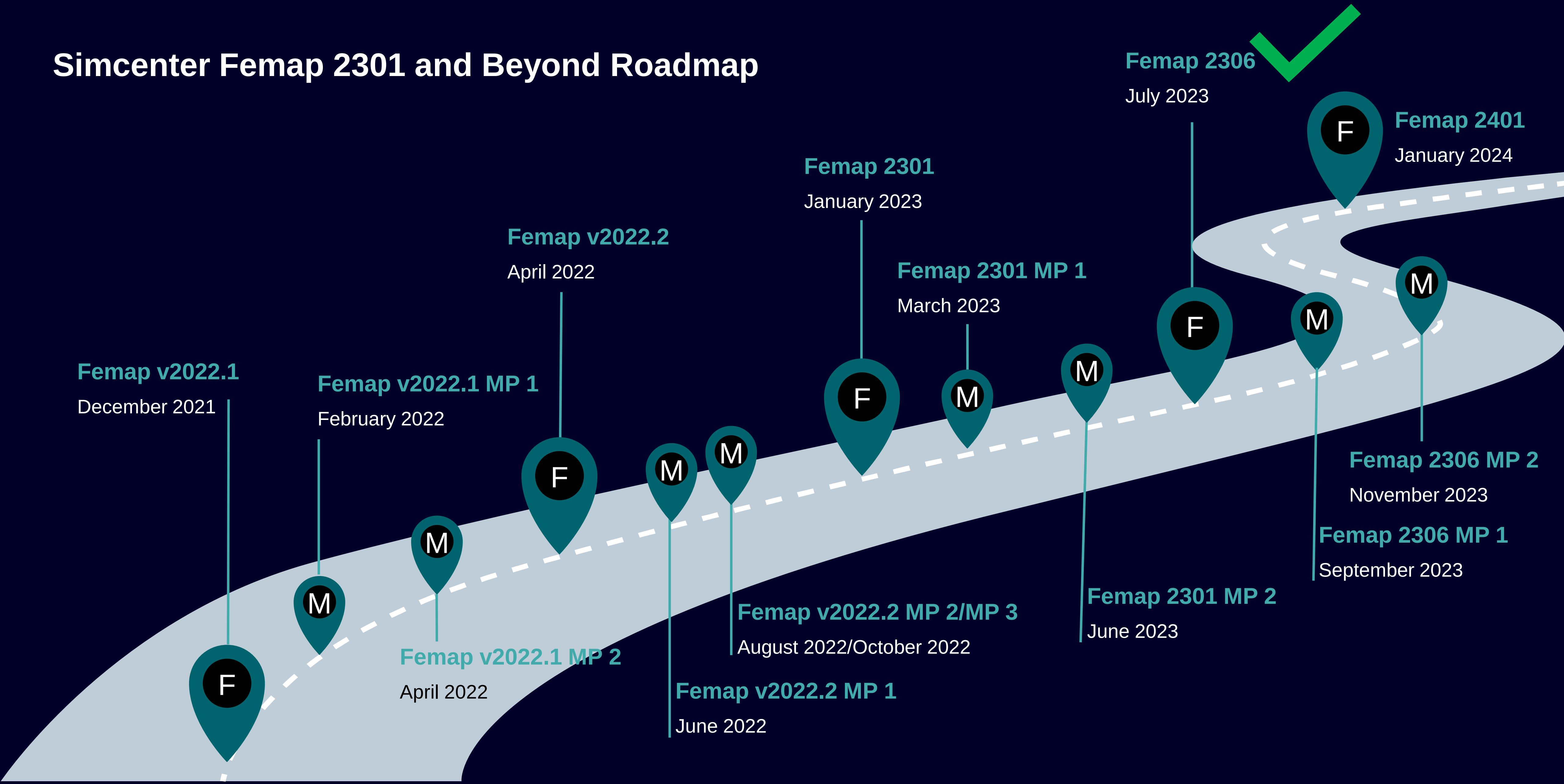
Simcenter Femap

What’s New in 2306

SIEMENS

Simcenter Femap 2301 and Beyond Roadmap



What’s new

Simcenter Femap 2306

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

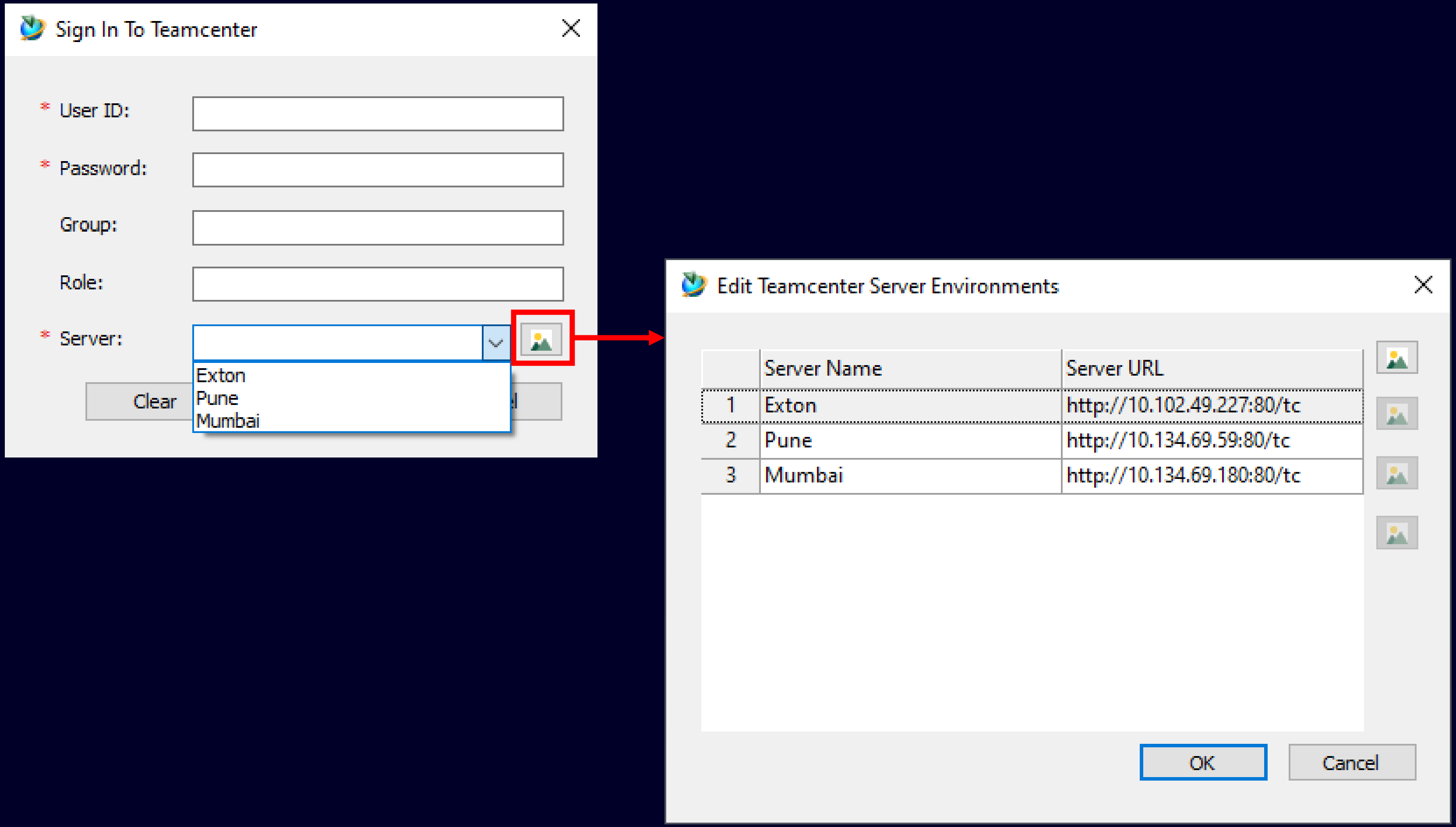
SIEMENS

Simcenter Femap 2306
Teamcenter Integration

Users can now specify multiple Teamcenter Server Environments via the PDM -> Teamcenter -> Sign In command

Server Definition items can be Added, Removed, Moved Up, and Moved Down in the Edit Teamcenter Server Environments dialog box

Once added, each server is then available in the drop-down



SIEMENS

What’s new

Simcenter Femap 2306

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

Simcenter Femap 2306

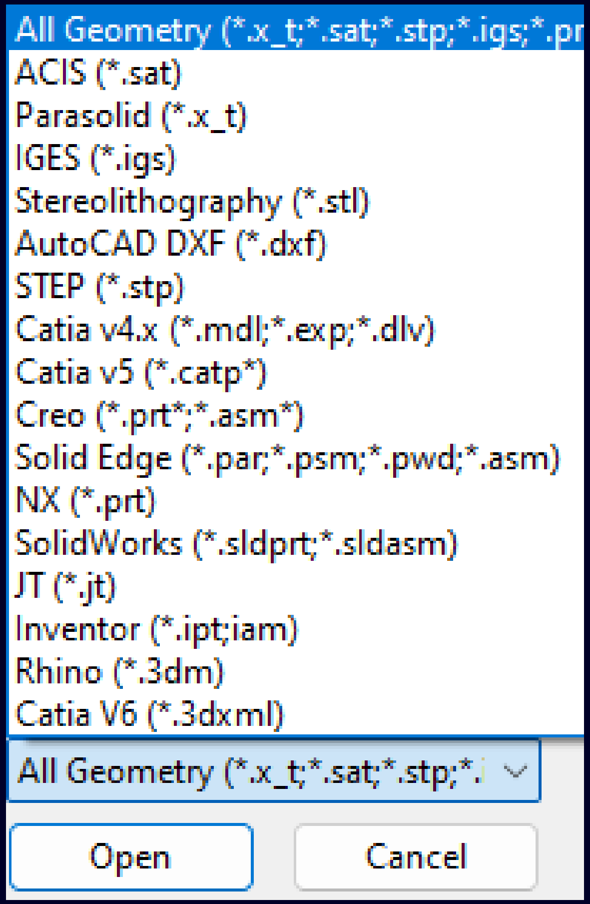
Geometry – CAD Data Translators

CAD Data Translators migrated to Siemens standard PLMXML components, which offers:

- New Catia v6 translator! – Supports .3dxml file
- New Inventor translator! – Supports .ipt and .iam files
- New Rhino translator! – Supports .3dm file
- Catia v5 translator now part of the base product!

Benefits

- Same User Interface for all translators
- Standard Siemens PLMXML Components are well maintained
- Easier for FEMAP development to keep CAD translators up to date



What’s new

Simcenter Femap 2306

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

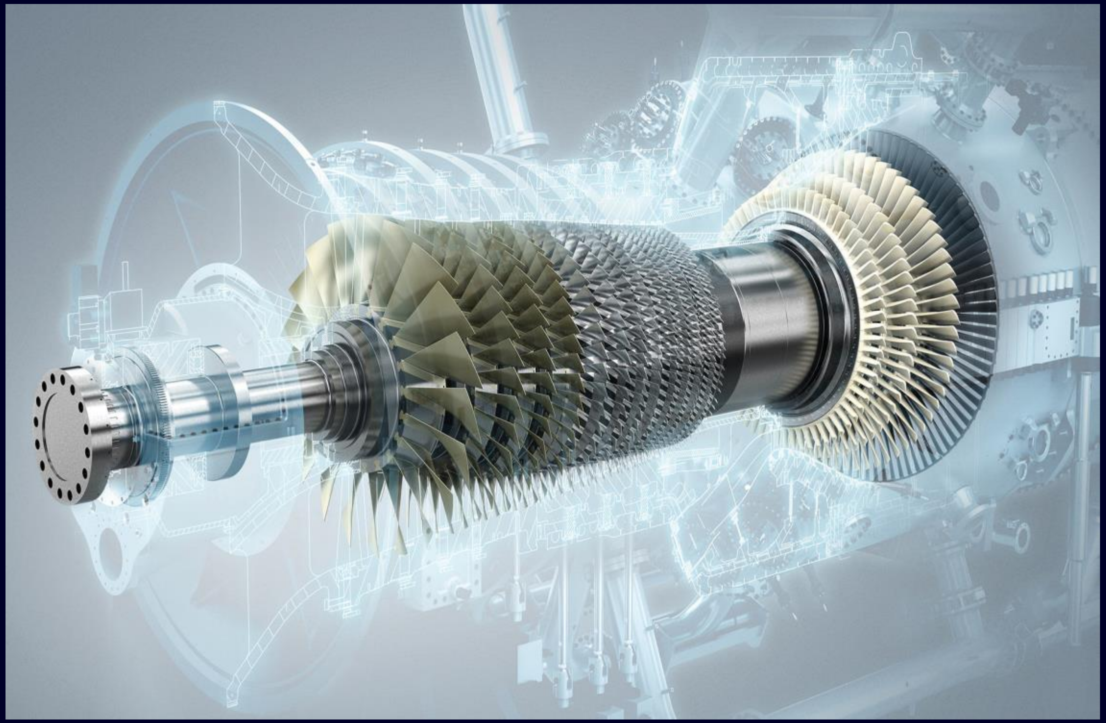
Postprocessing

Miscellaneous and API

Simcenter Femap 2306
Preprocessing – Rotor Dynamics

Simcenter Rotor Dynamics (SOL 414) is …
A dedicated solution to Rotor Dynamics analysis

- An established, recognized rotor dynamics technology, SAMCEF (ROTORS)
- Best-in-class and complete rotor dynamics capabilities
- Uses Simcenter Nastran input format
- Flexible modelling & smart solutions for industry-sized problems
- Dedicated streamlined workflows and scenarios
- Ease-of-use & short learning curve
- Tokenable solution (Enterprise Nastran)



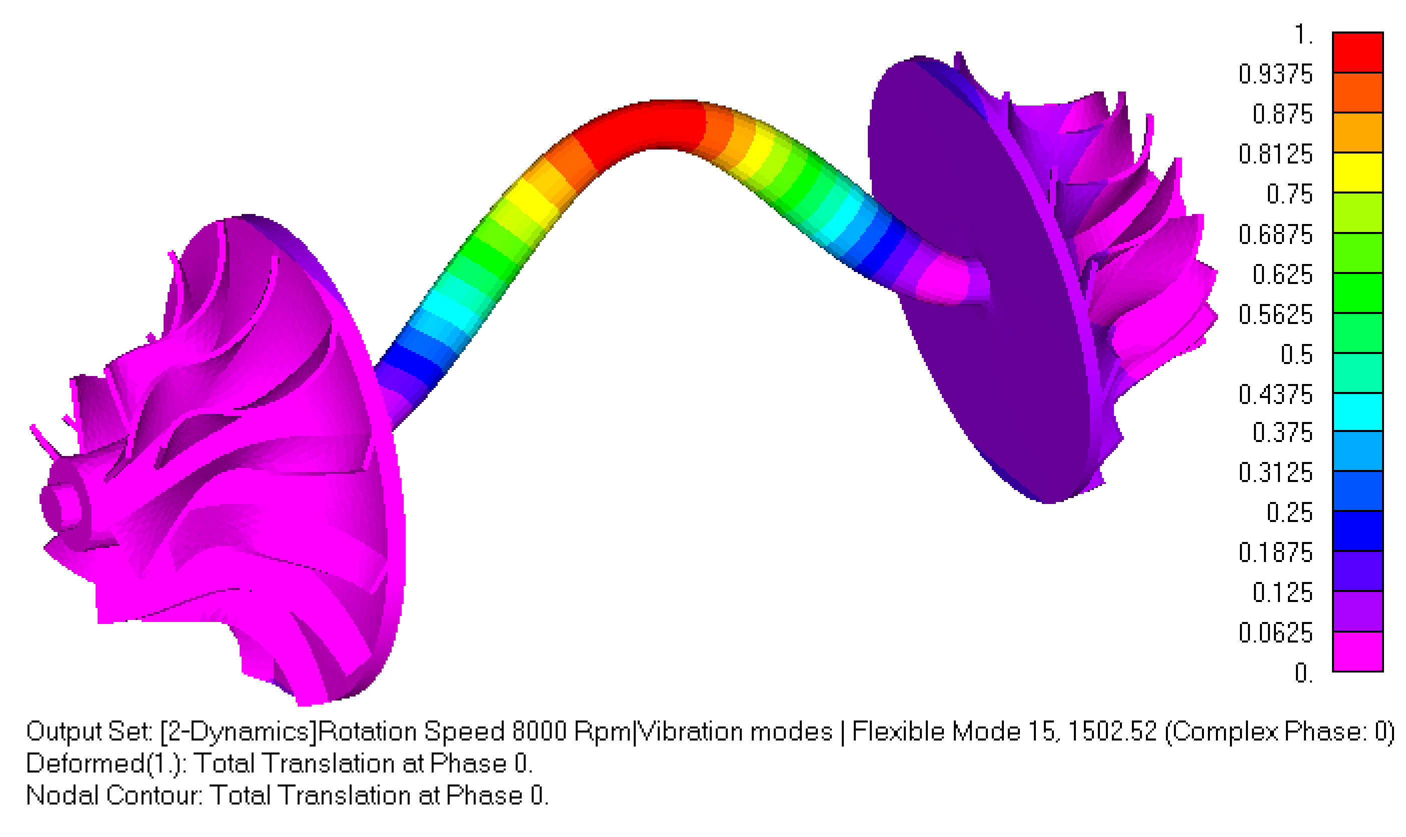
Simcenter Femap 2306
Preprocessing – Rotor Dynamics

Rotor Dynamics analysis in Simcenter Nastran (SOL 414) is now supported

New Element/Property Types
- Bearing
- Gear
- Fourier

New Rotor Dynamics Entity Types
- XY Plot Definitions
- Bearing Speeds
- Rotational Speeds

New Load Types
- Unbalance Mass
- Unbalance Moment



SIEMENS

Simcenter Femap 2306

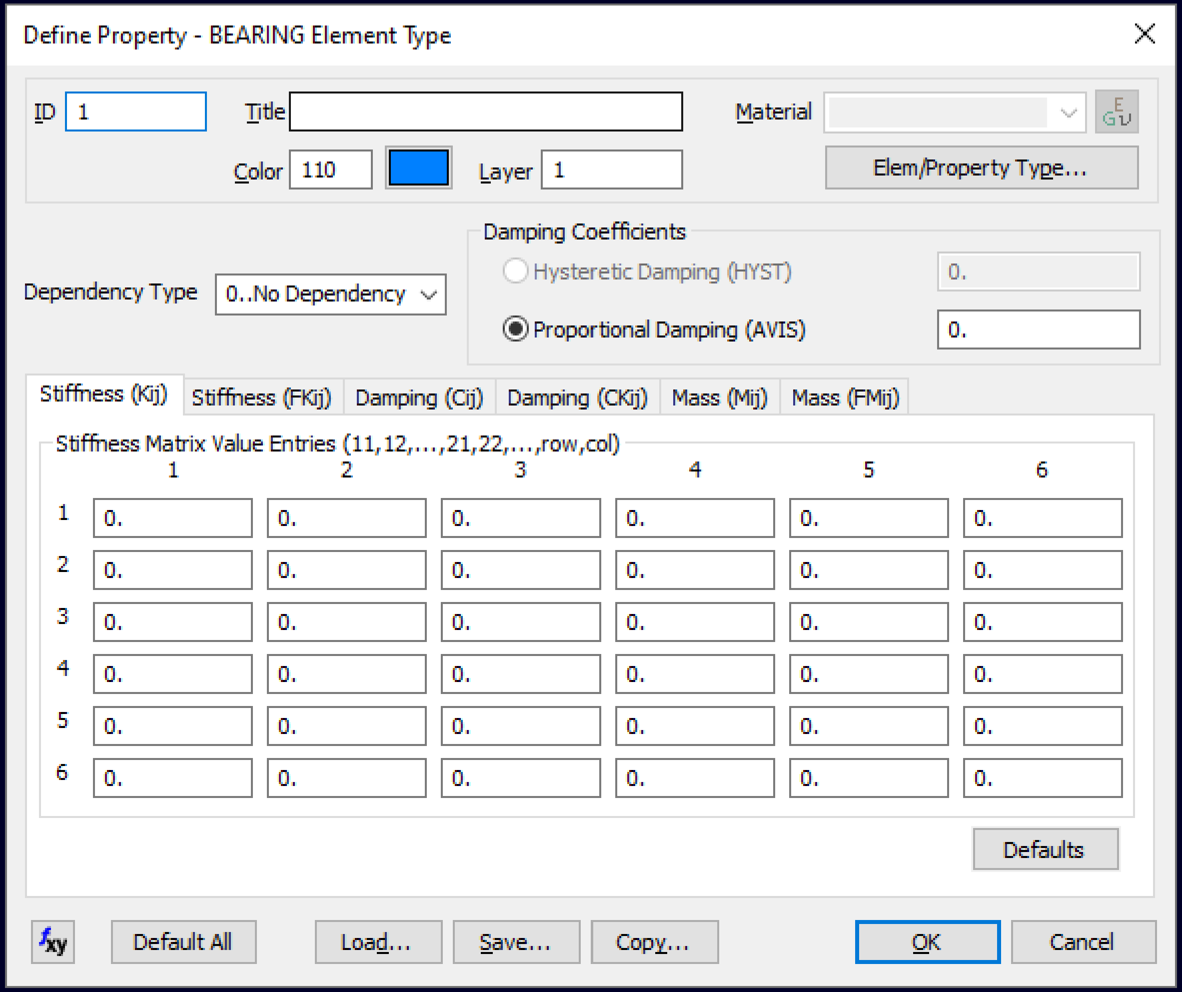
Preprocessing – Rotor Dynamics Element/Property Types

Bearing Element – writes CBEAR2 entry

- Defined by two nodes and a coordinate system

Bearing Property – writes PBEAR2 entry with TYPE field set to BEAR

- Constant stiffness, damping, and/or mass
- Option to have functionally-dependent stiffness, damping, and/or mass
  - Rotor Speed – writes ROTF flag
  - Frequency – writes FREQ flag
  - Time – writes TIME flag
- Hysteretic Damping – writes HYST and corresponding value
- Proportional Damping – writes AVIS and corresponding value



SIEMENS

Simcenter Femap 2306

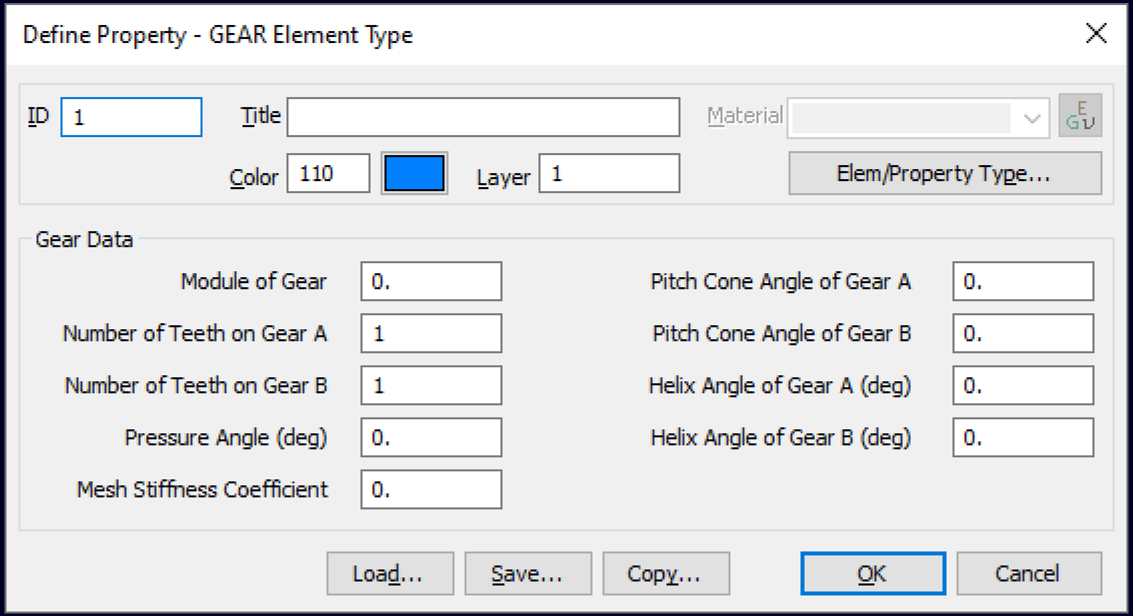
Preprocessing – Rotor Dynamics Element/Property Types

Gear Element – writes CGEAR entry

- Defined as two nodes at center of two gears (separate rotors) and two coordinate systems

Gear Property – writes PGEAR entry

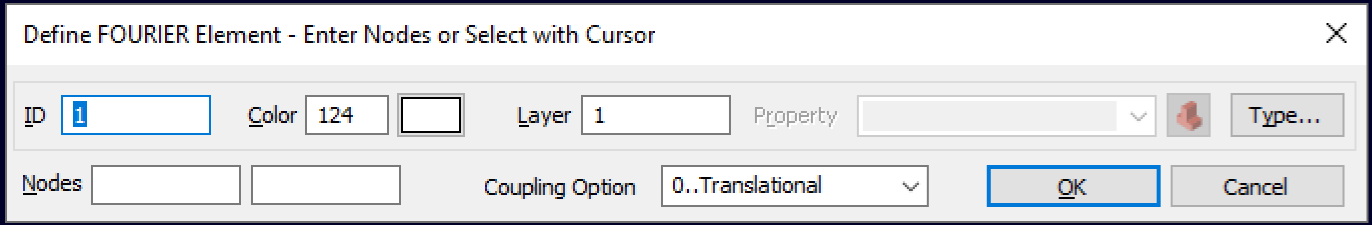
- Module of Gear – pitch diameter of gear divided by number of teeth
- Gear A/Gear B
  - Number of Teeth – must be > 0
  - Pitch Cone Angle – in degrees
  - Helix Angle (deg)
- Pressure Angle (deg)
- Mesh Stiffness Coefficient (optional) – if not specified, gears are considered rigid



Simcenter Femap 2306
Preprocessing – Rotor Dynamics Element/Property Types

Fourier Element – writes FOU3 entry

- Connects a 3-D node with six DOF (Node A) to a 2-D node of a Fourier multi-harmonic (axisymmetric) element (Node B)
- Coupling Option – writes appropriate value to OPT field on FOU3 entry
  - Translational (OPT = 1) – Nodes A and B must be coincident, translational DOFs are coupled
  - Rigid Translational (OPT = 2) – Node A must be a symmetry axis and Nodes A and B must have the same axial coordinate which creates a rigid connection between the translational DOF of the 3D node and the Fourier node
  - All DOFs (OPT = 5) – Similar to Rigid Translational, except all six DOFs are coupled, which allows the Fourier disk to deform (i.e., connection is not rigid)
- Can also be used in SOL 402



Fourier Property – none

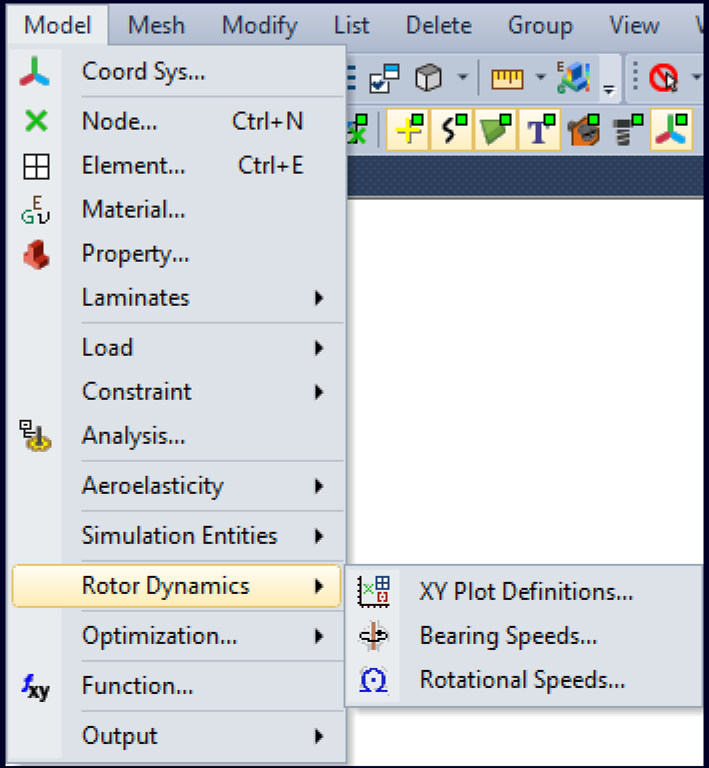
Simcenter Femap 2306

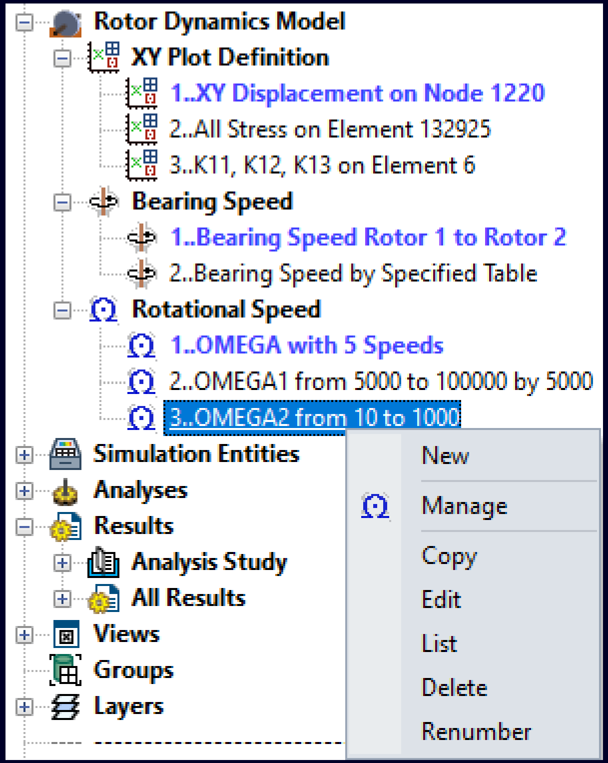
Preprocessing – Rotor Dynamics Entity Types

The new Rotor Dynamics Entity Types can be access in two ways:

- Model -> Rotor Dynamics… menu
  - In all cases, these commands display the dedicated Manager dialog box for the entity type

- Model Info tree – Context-sensitive menus for each entity type offer ability to:
  - Create a New Entity
  - Display Manager for Entity Type
  - Copy Selected Entities
  - Edit Selected Entities
  - List Selected Entities
  - Delete Selected Entities
  - Renumber Selected Entities





SIEMENS

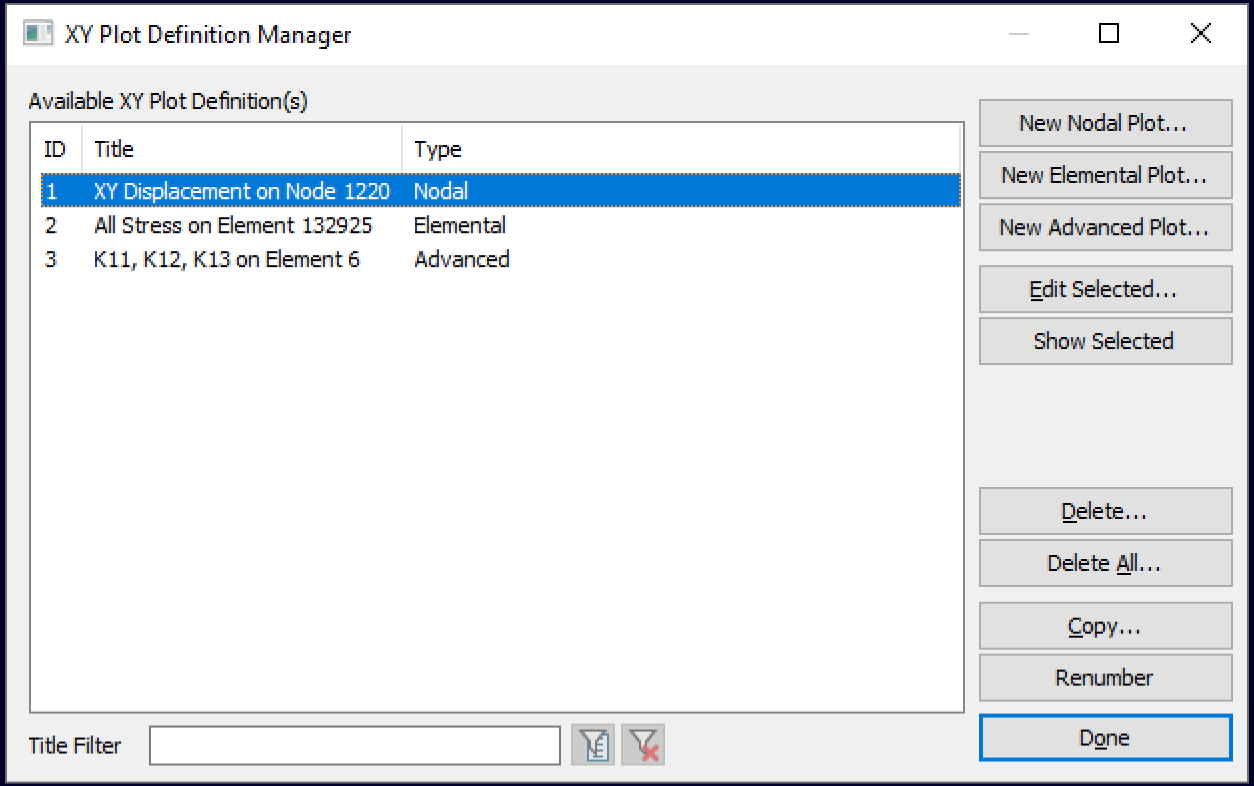
Simcenter Femap 2306
Preprocessing – Rotor Dynamics Entity Types

XY Plot Definition – Defines an XY plot request for Nodal, Elemental, or Advanced output

- Writes both OUTMGT Case Control and corresponding OUTMGT Bulk Data entries

XY Plot Definition Manager

- *New Nodal Plot*
- *New Elemental Plot*
- *New Advanced Plot*
- *Edit Selected*
- *Show Selected*
- *Delete and Delete All*
- *Copy*
- *Renumber*



SIEMENS

Simcenter Femap 2306

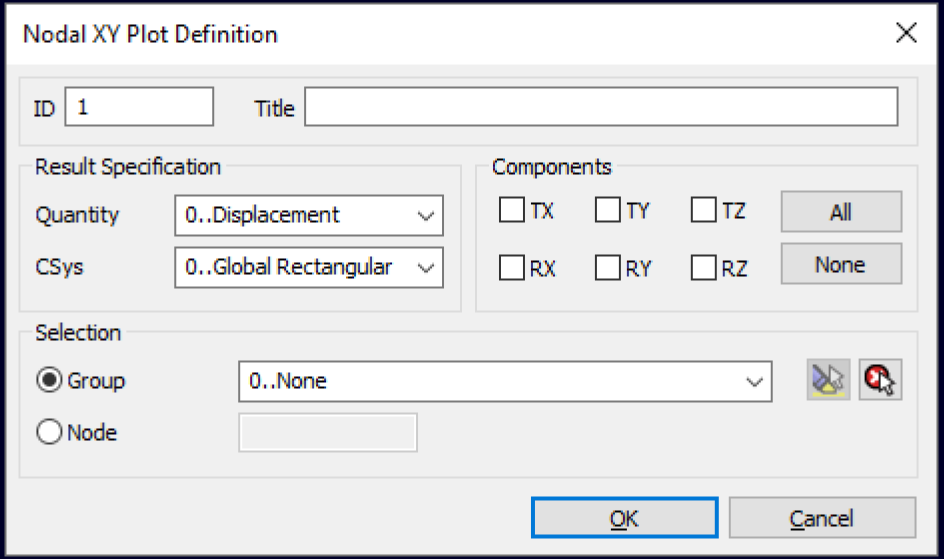
Preprocessing – Rotor Dynamics Entity Types

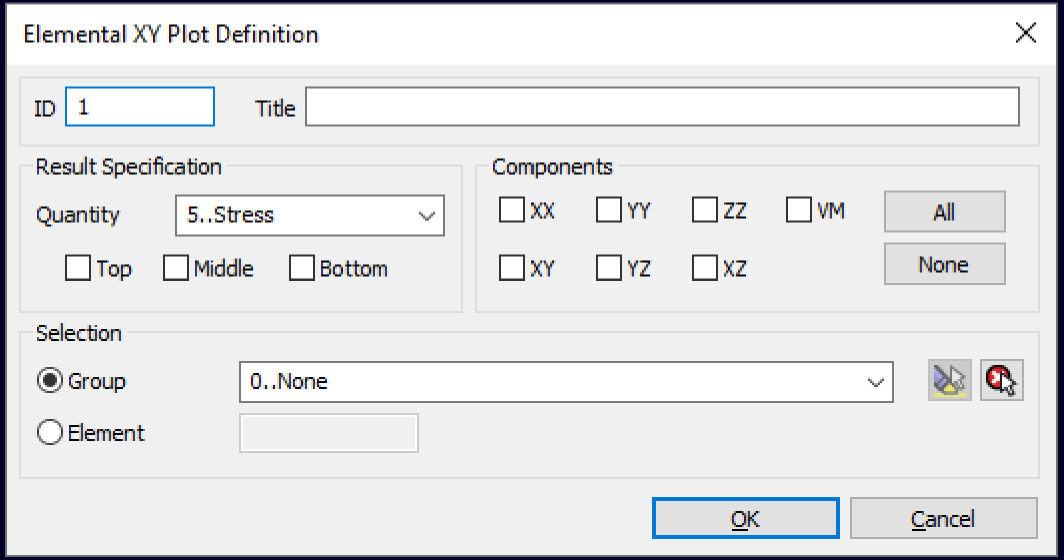
Nodal XY Plot – writes NODE or NODGRP to TYPE field of OUTMGT Bulk Data entries along with corresponding values for other fields based on selected Components, Coordinate System, and Quantity

- Displacement (DISP)
- Velocity (VELOC)
- Acceleration (ACCEL)
- Reaction Force/Moment (REAC)
- Elliptic Orbit Parameters (ORBITP)

Elemental XY Plot – writes ELEM or ELEMGRP to TYPE field of OUTMGT Bulk Data entries along with corresponding values for other fields based on selected Components and Quantity

- Stress (STRESS – Top, Middle, and/or Bottom)
- Force (FORCE – Top, Middle, and/or Bottom)





Simcenter Femap 2306

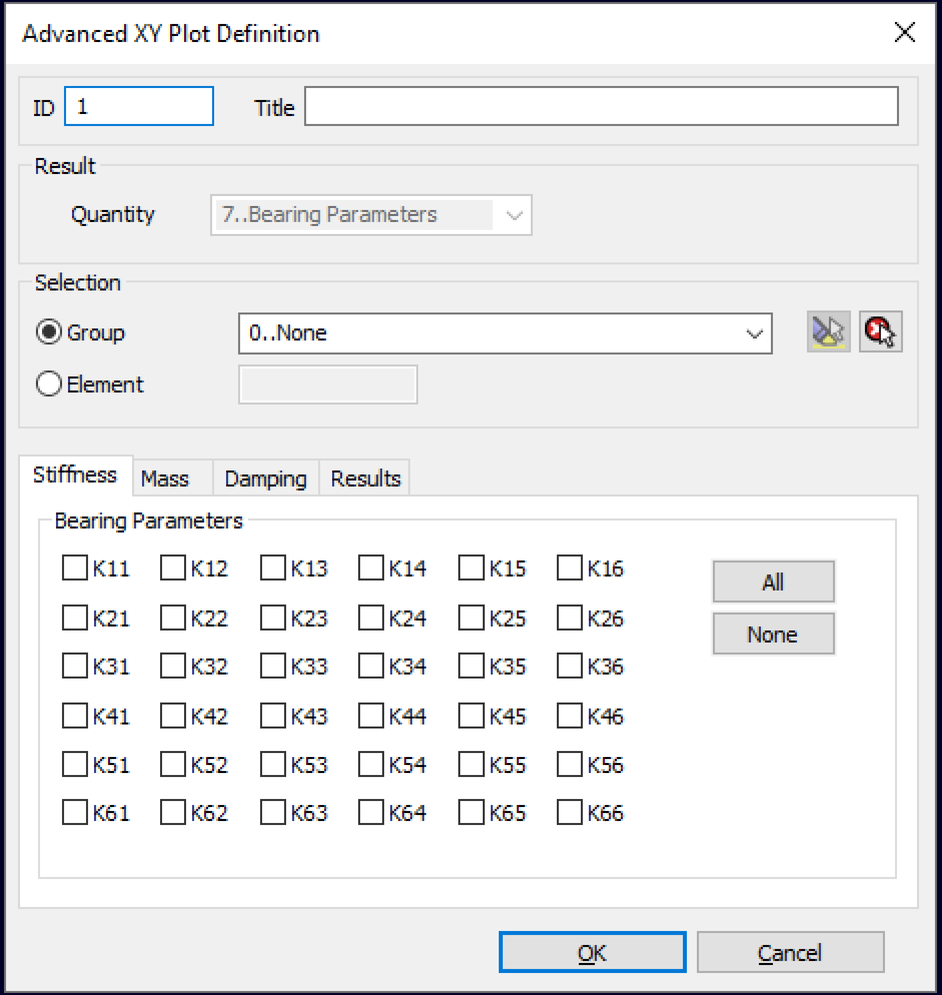
Preprocessing – Rotor Dynamics Entity Types

Advanced XY Plot – writes ELEM or ELEMGRP to TYPE field of OUTMGT Bulk Data entries along with corresponding values for other fields based on selected items on Stiffness, Mass, Damping, and/or Results tabs in lower portion of dialog box

Selected in Analysis Set Manager for available Rotor Dynamics Analysis Types

- SOL 414,103 – Eigenvalue Analysis (Only when there is a transient response subcase)
- SOL 414,111 – Harmonic Analysis
- SOL 414,129 – Transient Analysis

```
$ Femap XY Plot Definition 1 : XY Displacement on Node 1220
OUTMGT        100     NODE     1220     DISP          X
$ Femap XY Plot Definition 1 : XY Displacement on Node 1220
OUTMGT        100     NODE     1220     DISP          Y
$ Femap XY Plot Definition 2 : All Stress on Element 132925
OUTMGT        100     ELEM   132925   STRESS        ALL
$ Femap XY Plot Definition 3 : K11, K12, K13 on Element 6
OUTMGT        100     ELEM        6   CBEAR2                                   +
+             K11        1       K12        1       K13        1
```

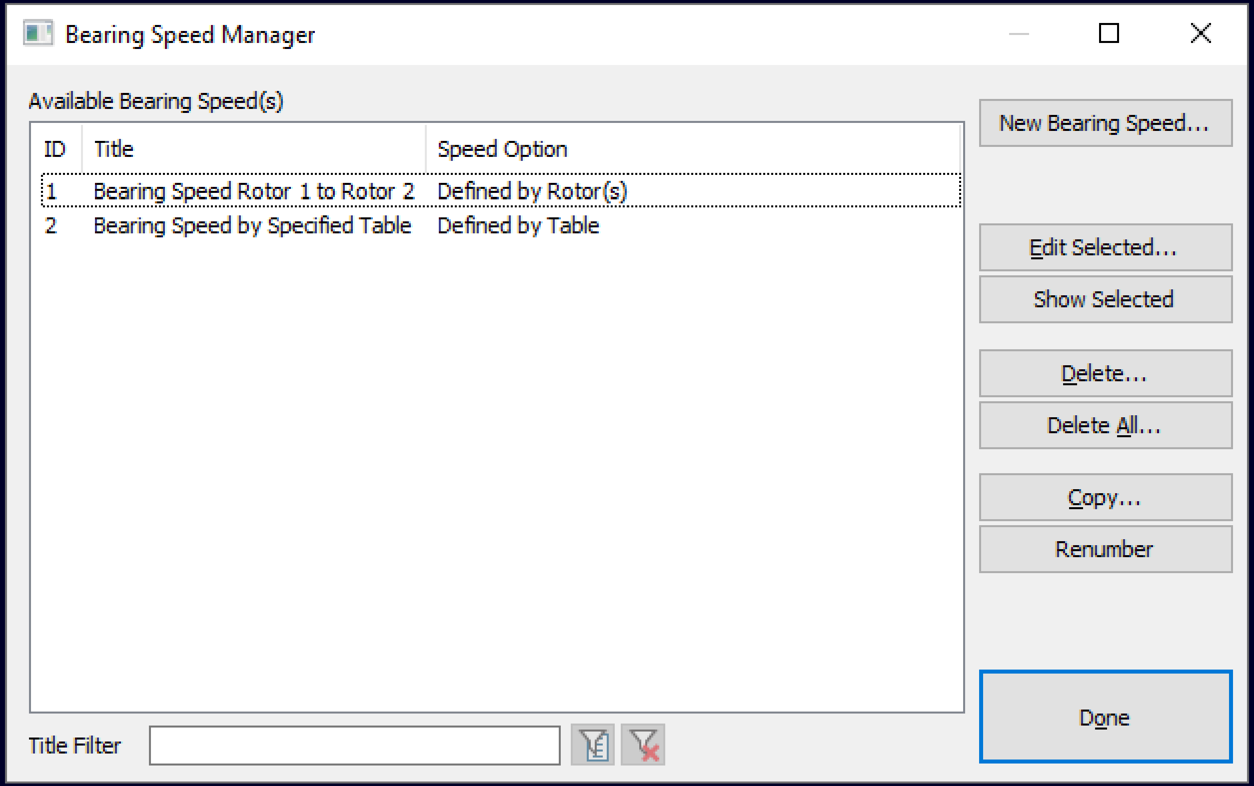


Simcenter Femap 2306

Preprocessing – Rotor Dynamics Entity Types

Bearing Speed – Connect Rotor Regions to Bearings when Bearing Properties are speed-dependent (Dependency Type = “1..Rotor Speed”)

- Writes SPEBE2 Bulk Data entries which can be used with CBEAR2 which have the TYPE field on PBEAR2 set to BEAR or USER (not currently supported in Femap)



Bearing Speed Manager

- *New Bearing Speed*
- *Edit Selected*
- *Show Selected*
- *Delete* and *Delete All*
- *Copy*
- *Renumber*

Simcenter Femap 2306
Preprocessing – Rotor Dynamics Entity Types

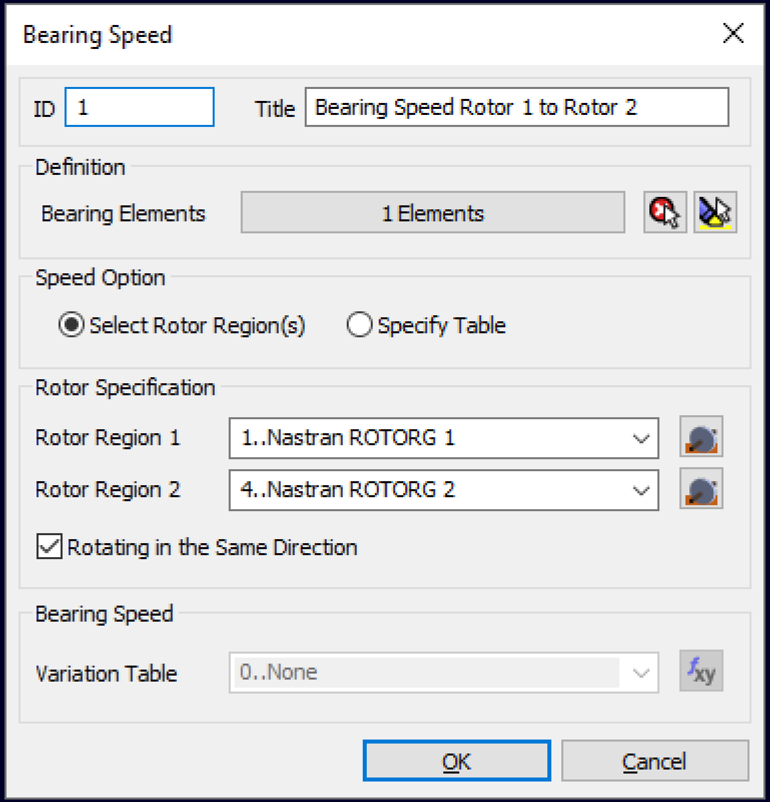
Bearing Speed – individual SPEBE2 entry written for each Bearing element that is selected in *Definition* section

Speed Option section defines possible user inputs
- *Select Rotor Region(s)* – Select 1 or 2 rotors from the *Rotor Specification* section
  - *Rotor Region 1* – writes ID to RID1 field
  - *Rotor Region 2* – writes ID to RID2 field
  - *Rotating in the Same Direction* option
    - When on, writes CORO to DIREC field
    - When off, writes COUN to DIREC field

Note: If only single rotor is selected, DIREC field is ignored

- *Specify Table* – choose function in *Bearing Speed* section
  - Writes ID of function to SPEED field

Note: DIREC field to be ignored, as bearing speeds are explicitly defined by function



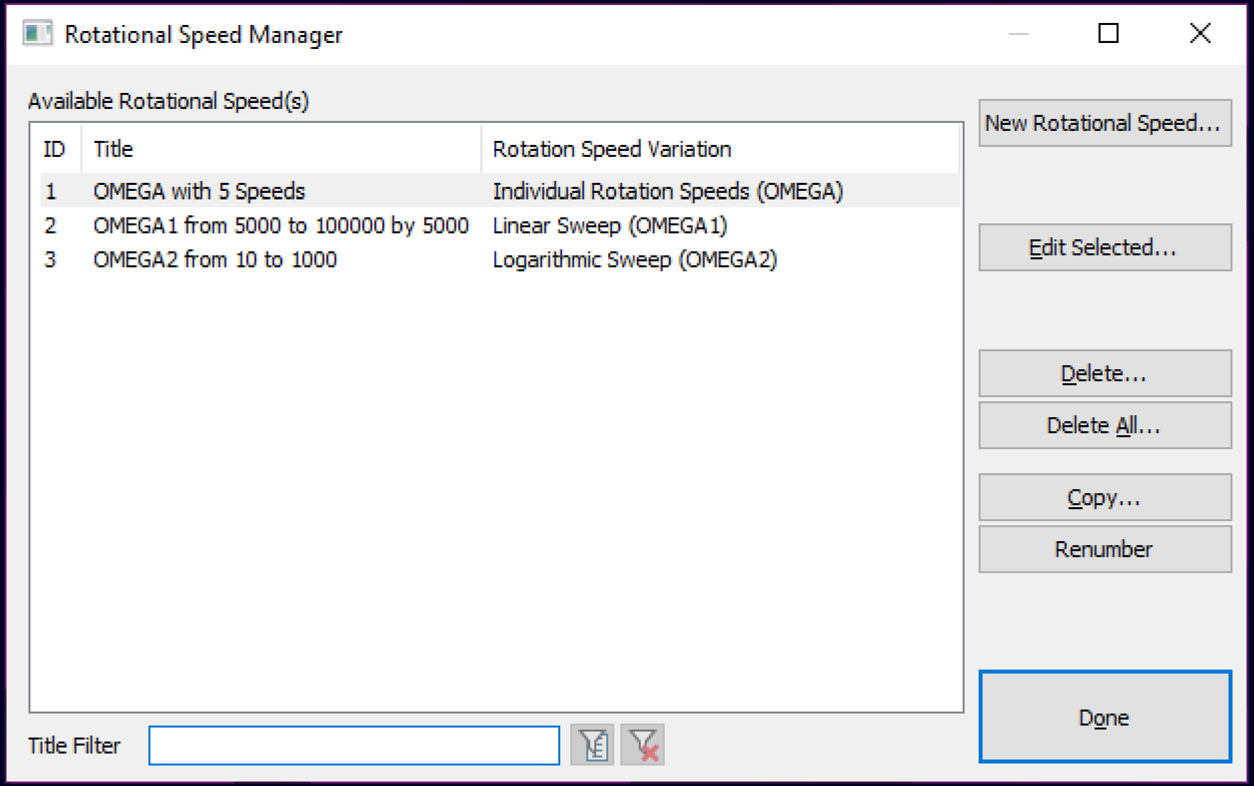
Simcenter Femap 2306
Preprocessing – Rotor Dynamics Entity Types

Rotational Speed – Allows definition of rotation speeds in 3 different forms for use in Complex Modal analysis (SOL 414,110)

- Writes OMEGA, OMEGA1, or OMEGA2 entries with appropriate values
- Rotational Speeds are selected via *Select Rotational Speed(s)* dialog box available only for subcases with Analysis Type set to “2..Rotor Complex Modal”

Rotational Speed Manager

- *New Rotational Speed*
- *Edit Selected*
- *Show Selected*
- *Delete* and *Delete All*
- *Copy*
- *Renumber*



SIEMENS

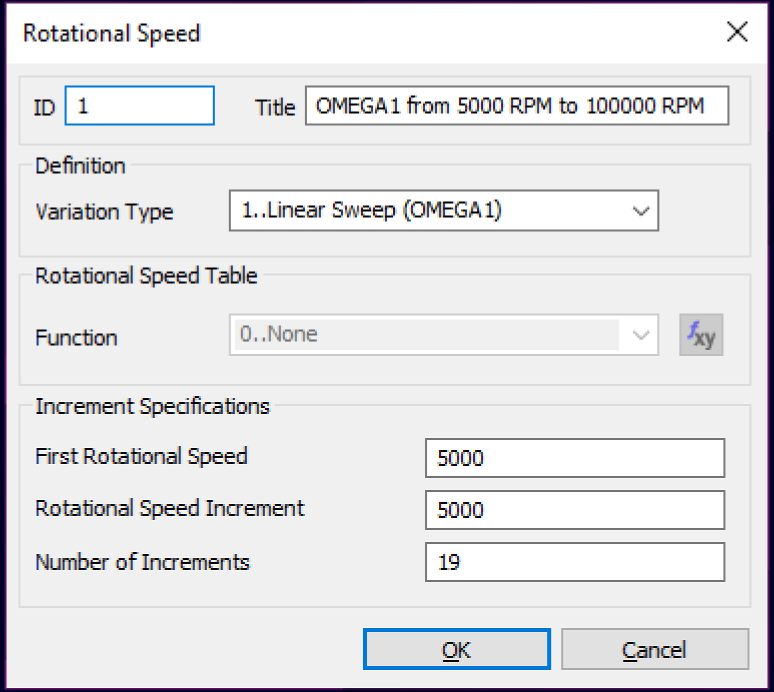
Simcenter Femap 2306
Preprocessing – Rotor Dynamics Entity Types

Rotational Speed – entry written to input file depends on option selected for *Variation Type* in *Definition* section

- *Individual Rotation Speeds (OMEGA)* – writes OMEGA entry
  - Values in Y column of *Function* selected in *Rotational Speed Table* section are written to Wi fields

- *Linear Sweep (OMEGA1)* – writes OMEGA1 entry
  - *First Rotational Speed* – value written to W1 field
  - *Rotational Speed Increment* – value written to DW field
  - *Number of Increments* – value written to NDW field

- *Logarithmic Sweep (OMEGA2)* – writes OMEGA2 entry
  - *First Rotational Speed* – value written to W1 field
  - *Last Rotational Speed*– value written to W2 field
  - *Number of Increments* – value written to NDW field



SIEMENS

Simcenter Femap 2306

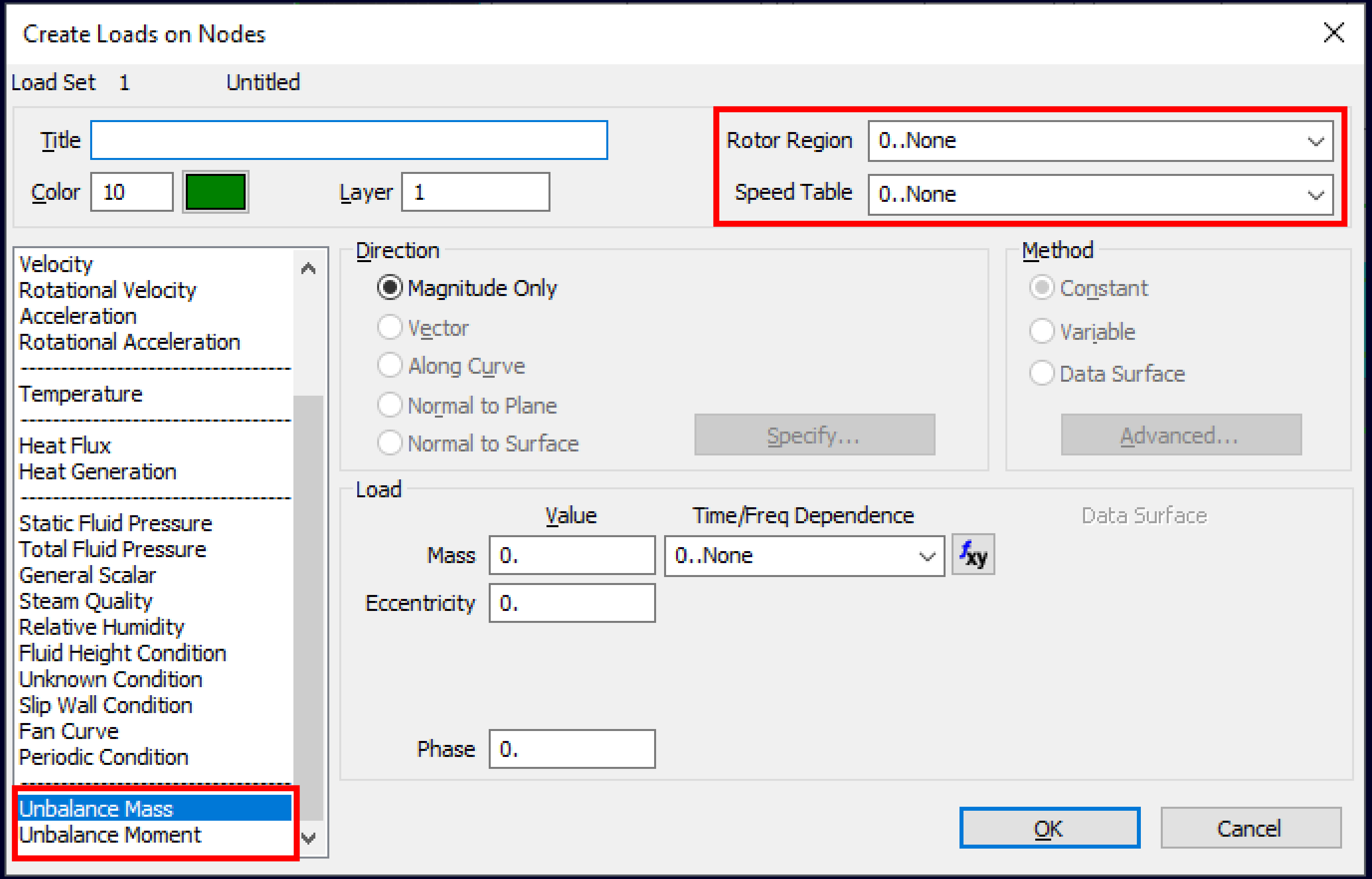
Preprocessing – Rotor Dynamics Load Types

The Model -> Load -> Nodal command can be used to create two new load types specific to Rotor Dynamics:

- Unbalance Mass
- Unbalance Moment

Both types:

- Require a *Rotor Region* be selected
- Allow a function to be specified as a *Speed Table* (option)
- Can only be used for SOL 414,111 (Subcase only) or SOL 414,129 by selecting them via *Boundary Conditions*
- Are written to UNBA Bulk Data entries referenced by ULOAD Case Control entry in specified case



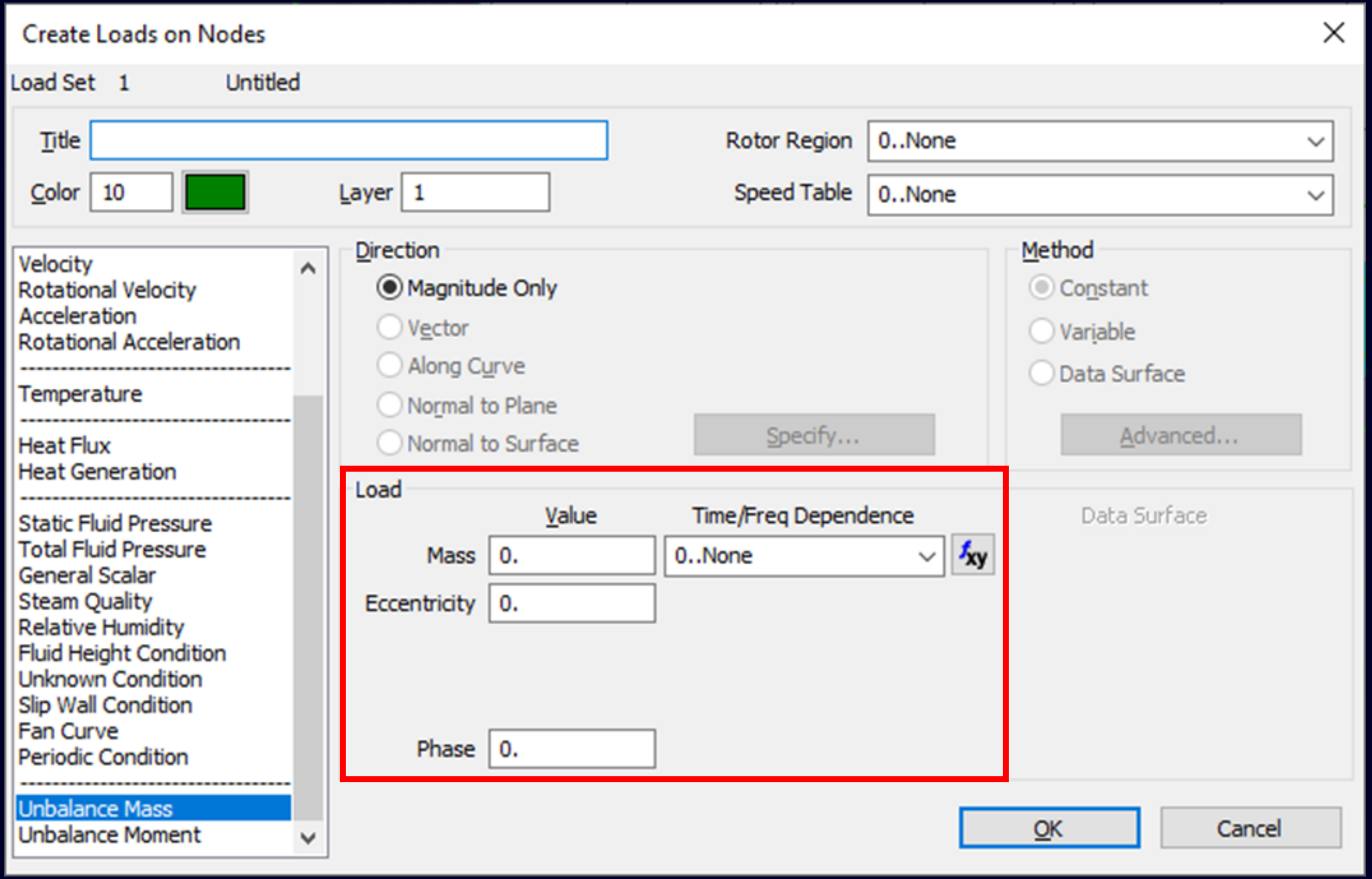
SIEMENS

Simcenter Femap 2306
Preprocessing – Rotor Dynamics Load Types

Unbalance Mass

- Rotor Region – writes ID to RID field
- Speed Table – writes ID of function to SPEED field
- Mass – writes value to MASS field
- Eccentricity – writes value to ECC field
- Phase – writes value to PHASEF field
- Time/Freq Dependence – writes ID of vs Time function to VBAL field

Note: If values for MASS, ECC, and VBAL are all specified, MASS and ECC are ignored by solver



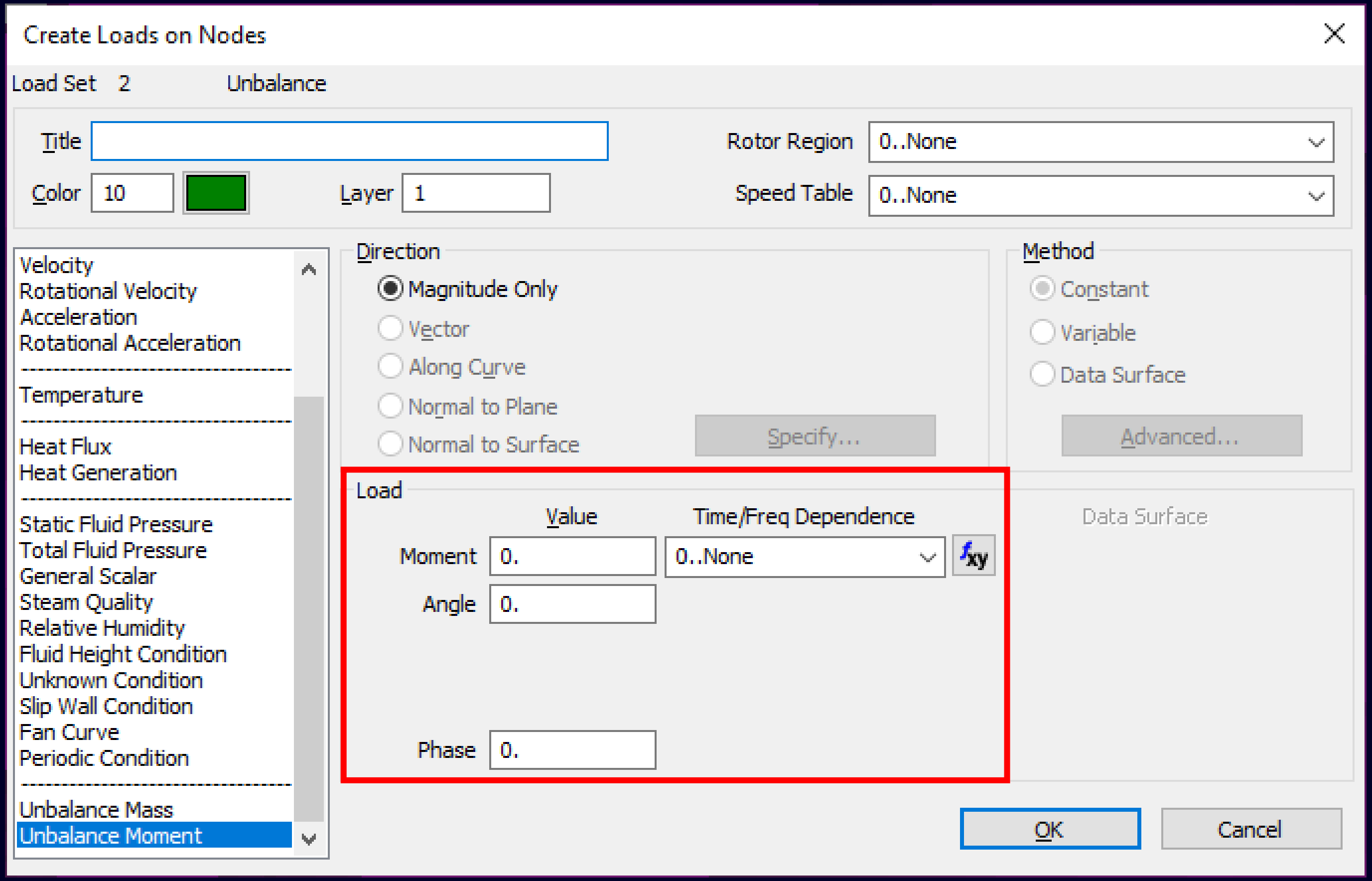
SIEMENS

Simcenter Femap 2306
Preprocessing – Rotor Dynamics Load Types

Unbalance Moment

- Rotor Region – writes ID to RID field
- Speed Table – writes ID of function to SPEED field
- Moment – writes value to MOMENT field
- Angle – writes value to THETA field
- Phase – value written to PHASEM field
- Time/Freq Dependence – writes ID of vs Time function to MBAL field

Note: If values for MOMENT, THETA, and VBAL are all specified, MOMENT and THETA are ignored by solver



SIEMENS

What’s new

Simcenter Femap 2306

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

Simcenter Femap 2306

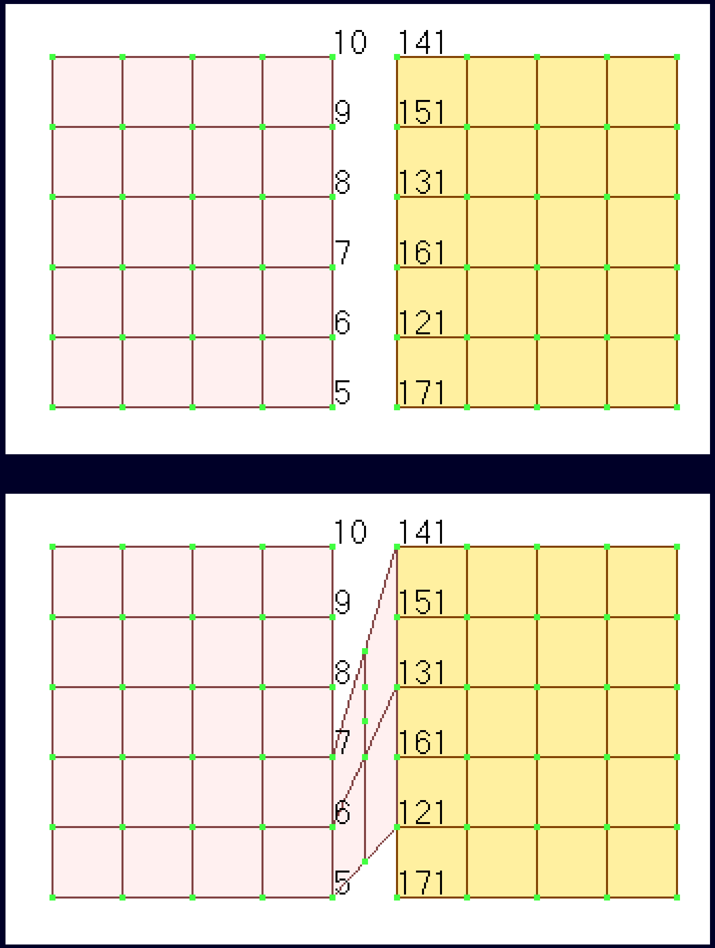
Meshing – Mesh -> Region Enhancement

The Mesh -> Region command has been in Femap for a very long time, but was enhanced to a large extent for 2306

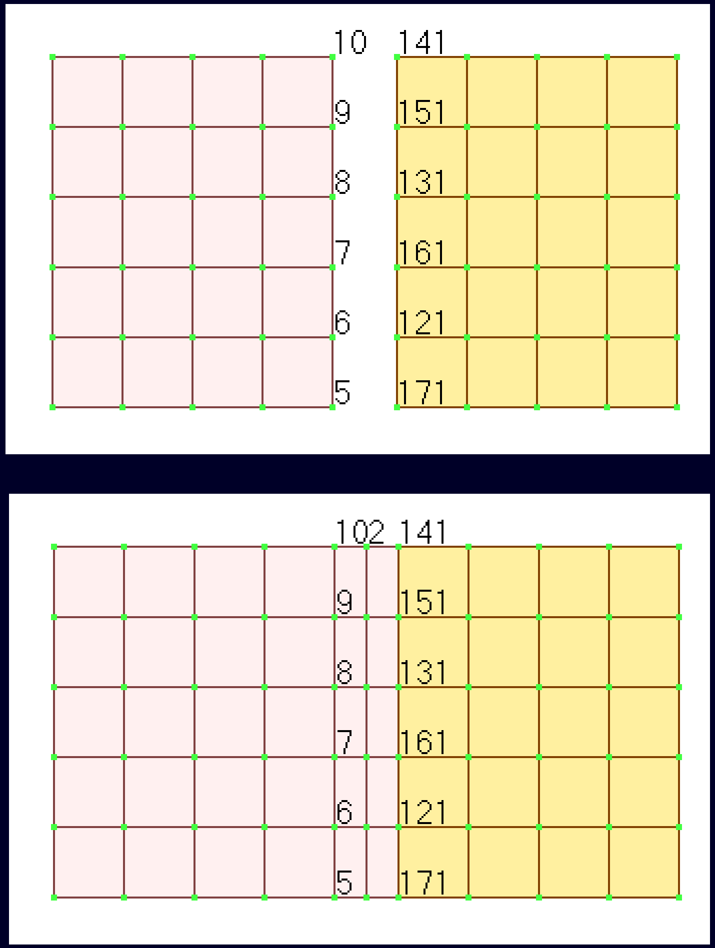
In previous versions, the node numbering needed to be in a very specific order for the command to create appropriate elements between the two sets of selected nodes (“Region 1” and “Region 2”)

In 2306, the only requirement that remains is that the same number of nodes be selected for “Region 1” and “Region 2”.

2301 – Box Pick Regions

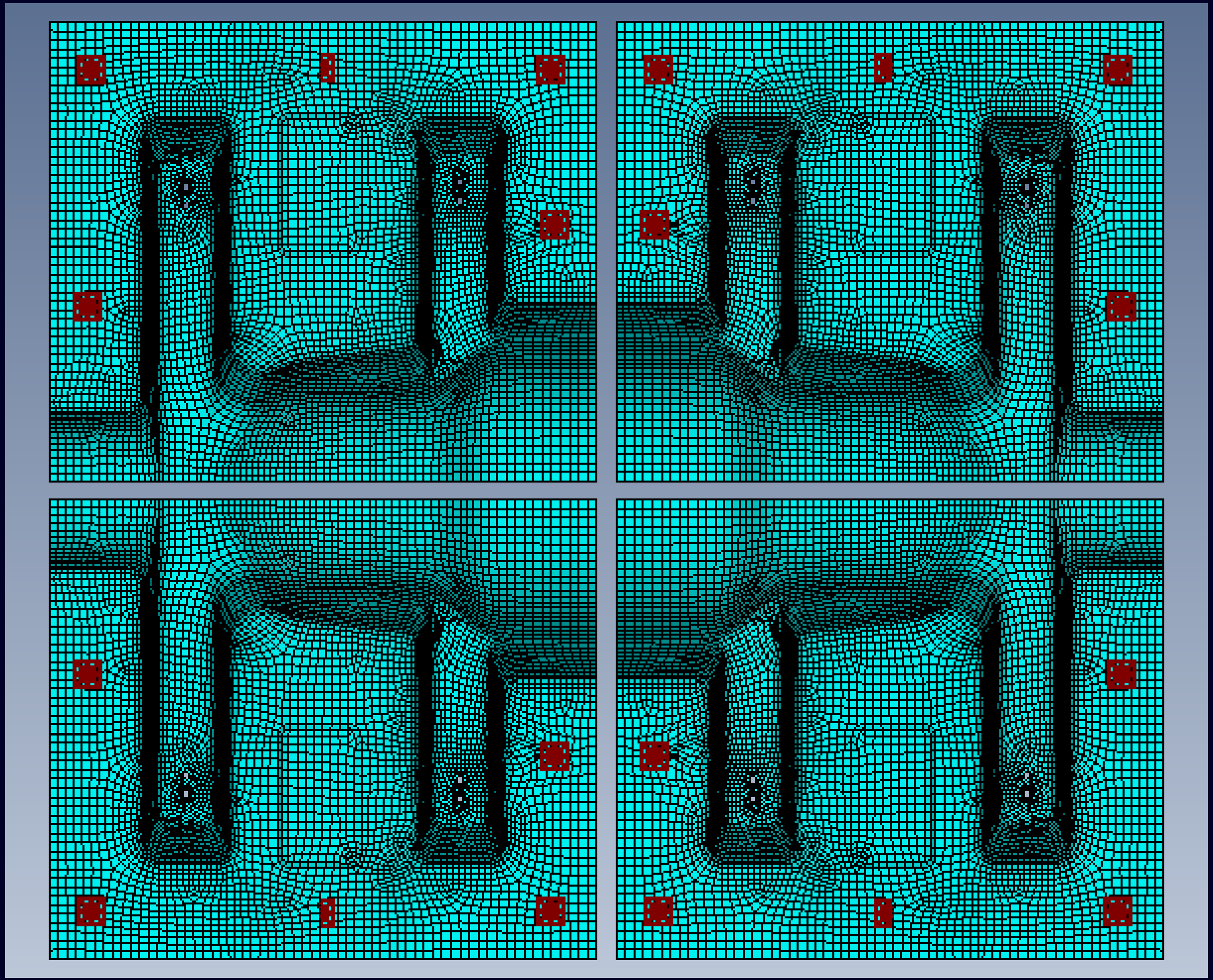


2306 – Box Pick Regions

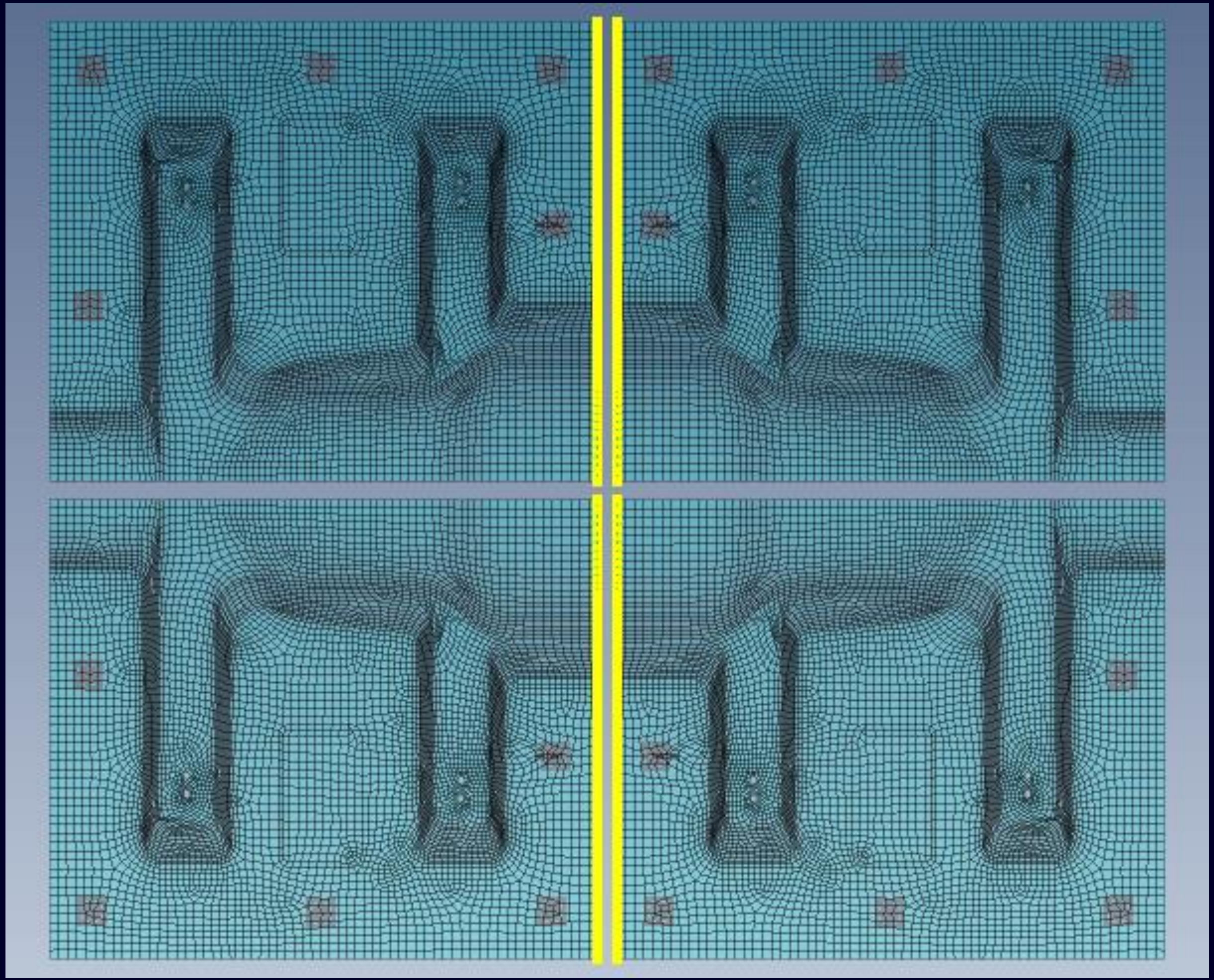


Simcenter Femap 2306
Meshing – Mesh -> Region Enhancement

Starting Model – No Geometry

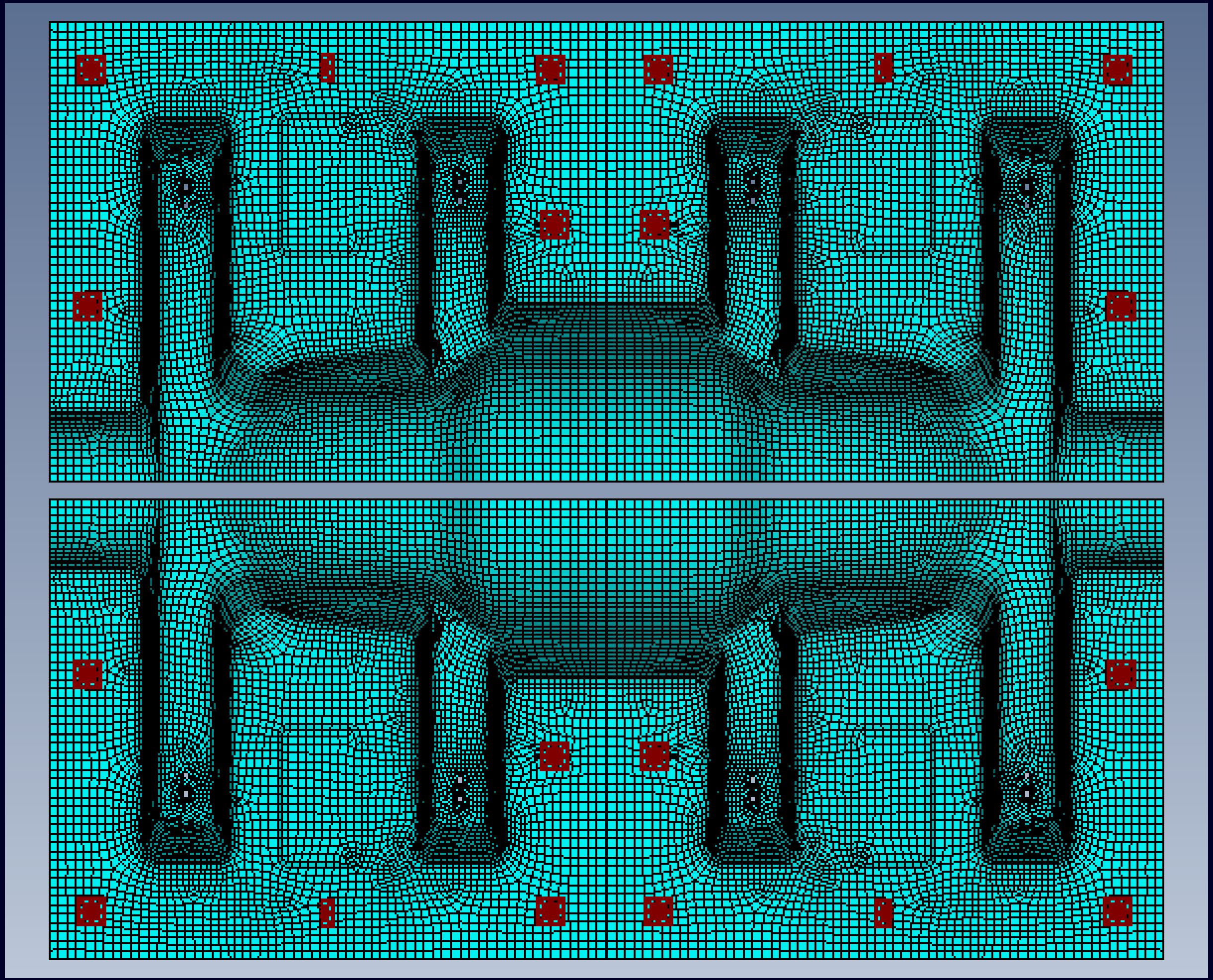


Regions Selected Vertical Middle Area

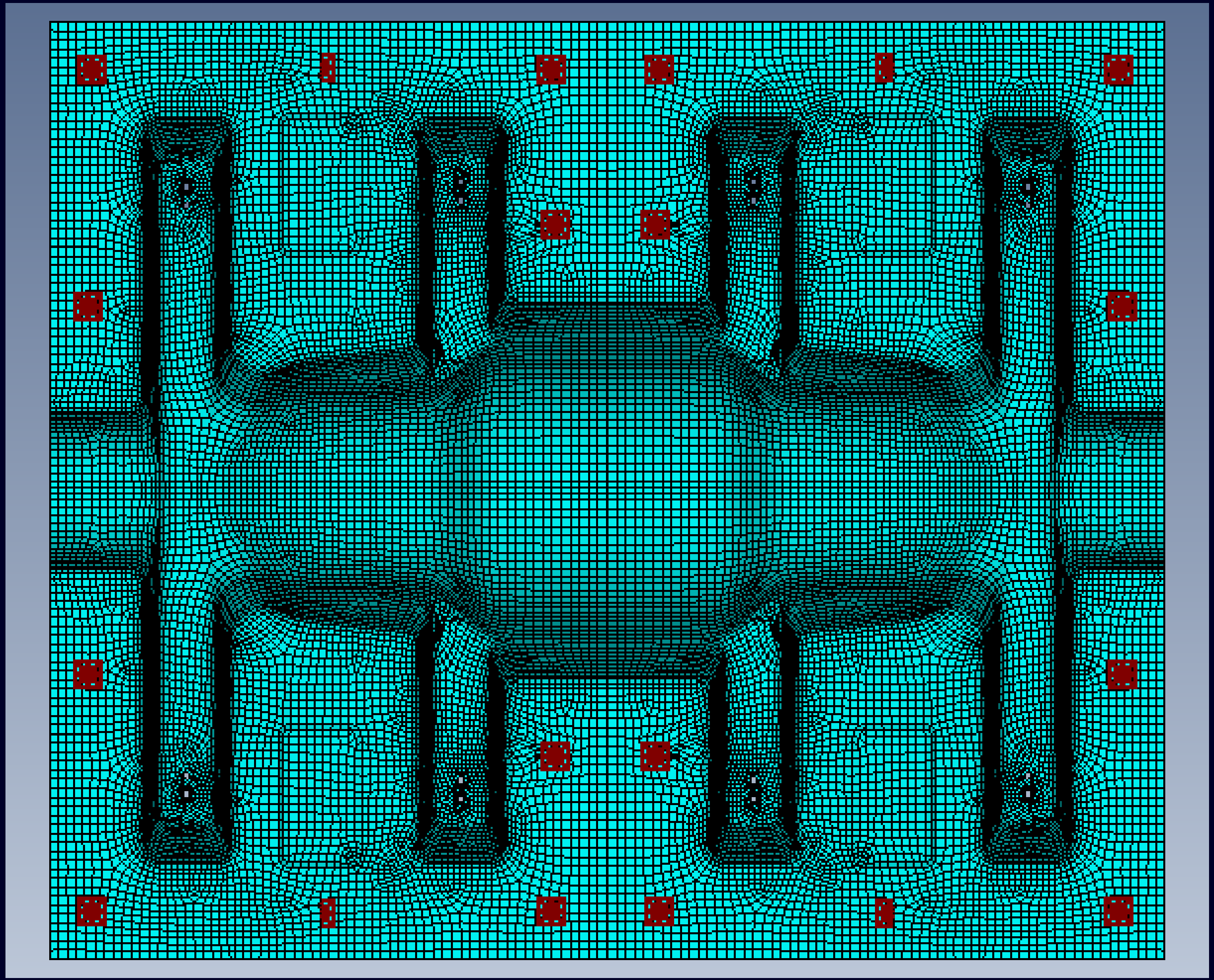


Simcenter Femap 2306
Meshing – Mesh -> Region Enhancement

Horizontal Gaps in Mesh Skipped



Mesh Fully Connected by using Mesh -> Region Again



SIEMENS

What’s new

Simcenter Femap 2306

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

Simcenter Femap 2306
Performance Improvements – Graphics

Data Structures were defined and modified to allow Unified Graphics Architecture (UGA), first introduced in 2301, to be able to handle Nodes and Elements in an upcoming version

Improved performance of dynamic rotation in larger models which essentially had a property defined for each element in the model

- One example was a model that had 2 Million 10-noded tetrahedral elements and 2 million properties which went from rotating at 5 frames/second to between 30-40 frames/second on the same hardware (6X-8X improvement)

SIEMENS

What’s new

Simcenter Femap 2306

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

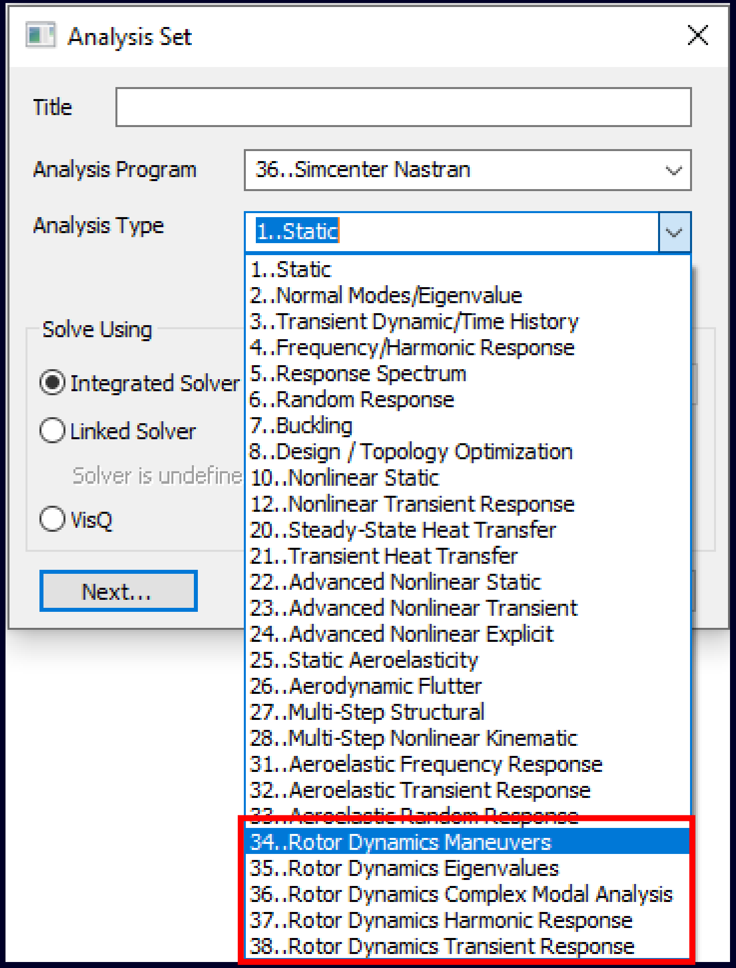
SIEMENS

Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

Five new analysis types for Rotor Dynamics have been added for Simcenter Nastran

- “34..Rotor Dynamics Maneuvers” (SOL 414,101) – Linear static analysis with rotational forces
- “35..Rotor Dynamics Eigenvalue Analysis” (SOL 414,103) – Compute modes of the rotating system at rest
- “36..Rotor Dynamics Complex Modal Analysis” (SOL 414,110) – Find critical speeds and stability of rotating system
- “37..Rotor Dynamics Harmonic Response” (SOL 414,111) – Model rotating system under synchronous/asynchronous frequency-dependent excitation
- “38..Rotor Dynamics Transient Response” (SOL 414,129) – Direct solution to model rotating system under time-dependent excitation



Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

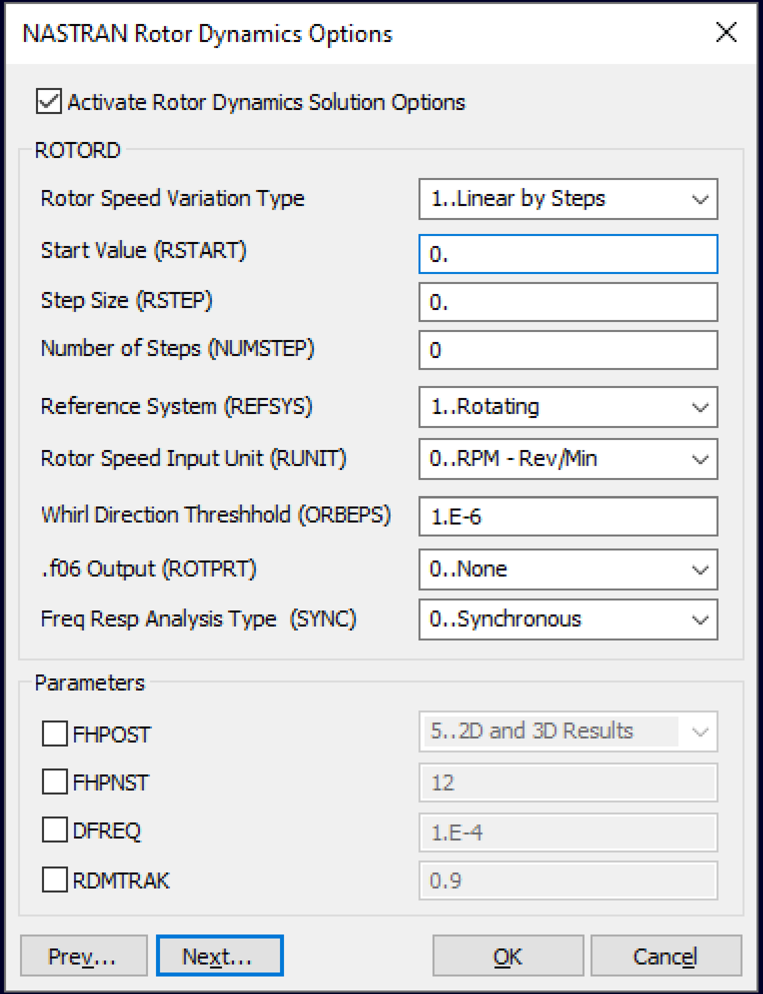
Much like other types of analysis, each type of Rotor Dynamics analysis requires specific inputs

New and updated dialog boxes available for all SOL 414 analysis types include:

- *NASTRAN Rotor Dynamics Options* – New!
- *Rotor Modeling Assembly Selection* – New!
- *Boundary Conditions* – Unique for Rotor Dynamics

Other new dialog boxes that may be used for specific type of SOL 414 analysis include:

- *Select XY Plot Definition(s)*
- *Select Rotational Speed(s)*



SIEMENS

Simcenter Femap 2306

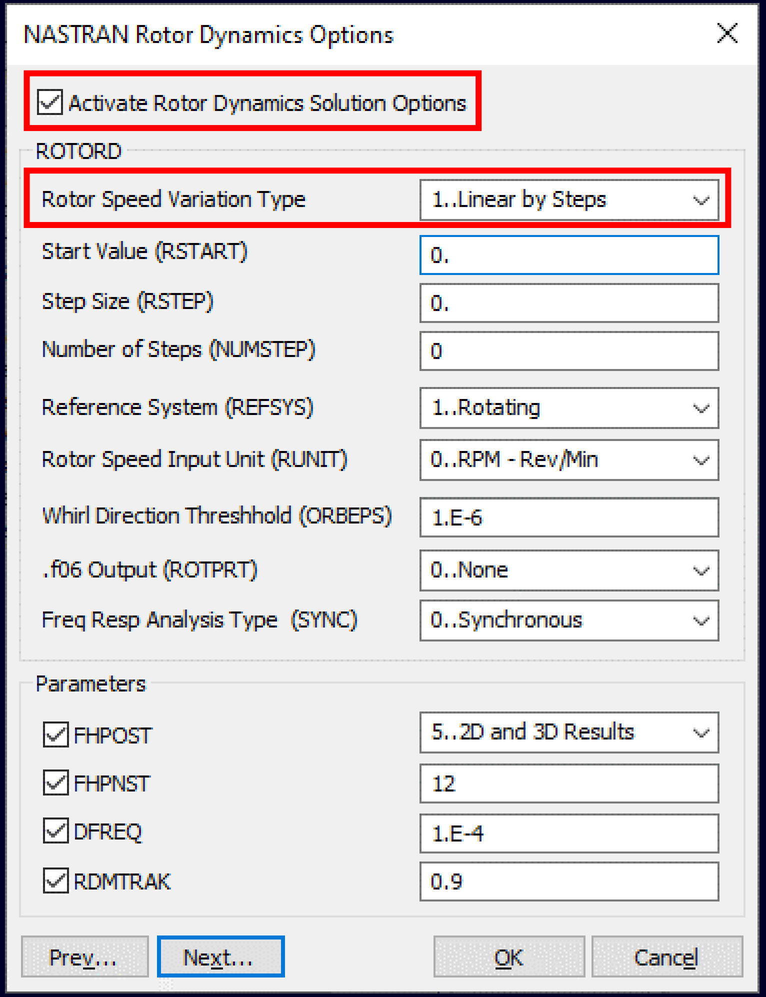
Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

The *NASTRAN Rotor Dynamics Options* dialog box is used to specify the Rotor Dynamics Solution Options as well as Parameters used for certain types of analysis

Activate Rotor Dynamics Solution Options – when on, entries in specified in ROTORD section are written to input file

ROTORD section – writes first two lines of ROTORD entry

- *Rotor Speed Variation Type* – Used to limit available inputs for reference rotor speed
  - “0..Constant” – reference rotor speed remains constant, thus only Start Value (RSTART) can be entered
  - “1..Linear by Steps” (Default) – Enter values for *Start Value (RSTART), Step Size (RSTEP),* and *Number of Steps (NUMSTEP)* to define varying rotor reference speed
  - “2..Function of a Sweeping Parameter” – Rotor speed defined in another manner (OMEGAi, FREQi, or TSTEPi)

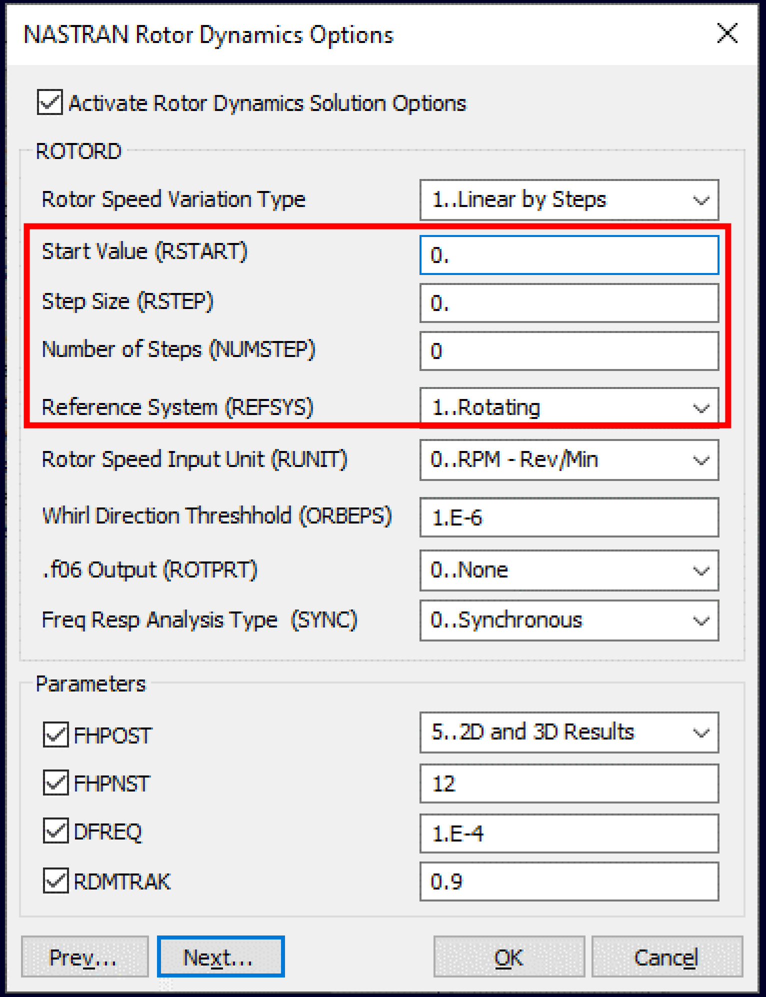


SIEMENS

Simcenter Femap 2306
Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

ROTORD section – writes first two lines of ROTORD entry

- *Start Value (RSTART)* – value written to RSTART field
- *Step Size (RSTEP)* – value written to RSTEP field
- *Number of Steps (NUMSTEP)* – value written to NUMSTEP field
- *Reference System (REFSYS)* – Frame of Reference
  - "0..Fixed" – specifies "fixed" frame of reference and writes 'FIX' to REFSYS field
  - "1..Rotating" (Default) – specifies "rotating" frame of reference and writes 'ROT' to REFSYS field



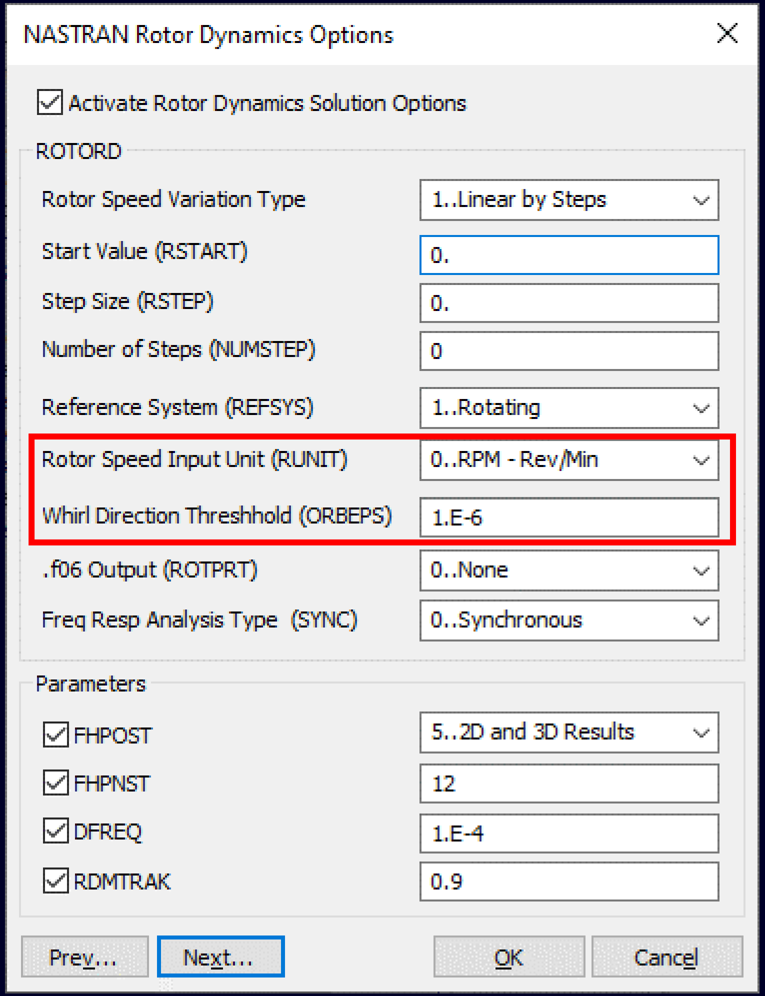
SIEMENS

Simcenter Femap 2306
Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

ROTORD section – writes first two lines of ROTORD entry

- *Rotor Speed Input Unit (RUNIT)* – selects units for rotor speed input, written to RUNIT field
  - "0..RPM – Revs/Min" (Default) – writes 'RPM' characters
  - "1..CPS – Cycles/Sec" – writes 'CPS' characters
  - "2..HZ – Hertz" – writes 'HZ' characters
  - "3..RAD – Radians/Sec" – writes 'RAD' characters

- *Whirl Direction Threshold (ORBEPS)* – threshold value for detection of whirl direction written to ORBEPS field



SIEMENS

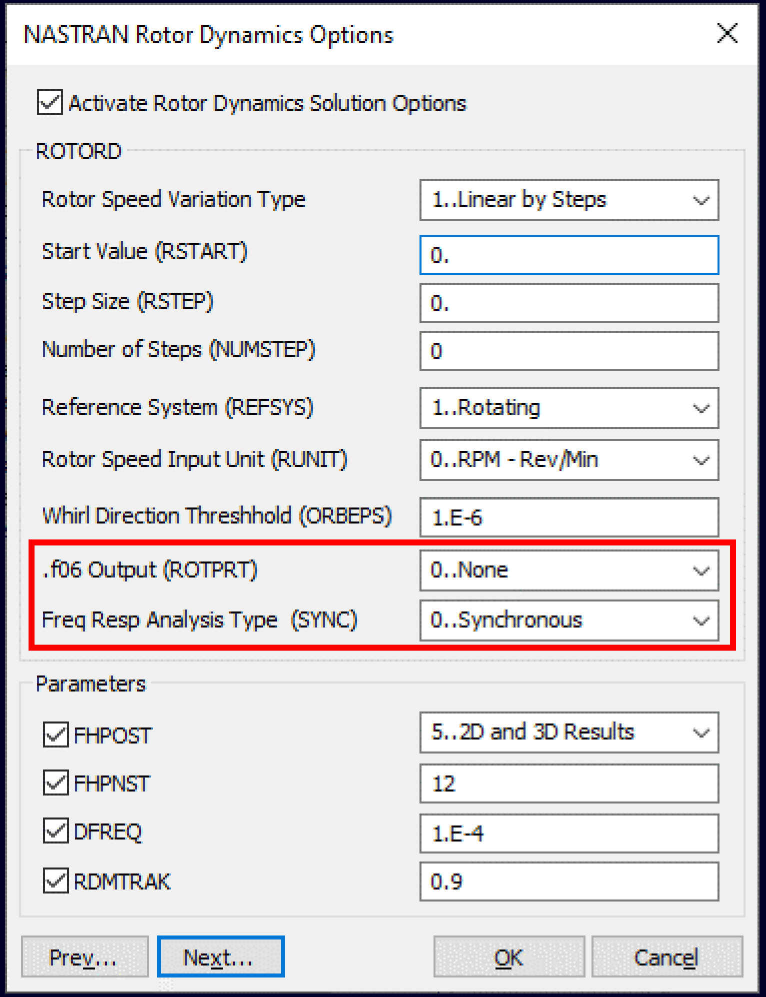
Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

ROTORD section – writes first two lines of ROTORD entry

- *.f06 Output (ROTPRT)* – controls output written to .f06 file by specifying value for ROTPRT field
  - "0..None" (Default) – writes 0, prints nothing
  - "1..Generalized Matrices" – writes 1, prints:
    - Generalized matrices
    - Final nonlinear bearing values at each frequency/time
  - "2..Eigenvalue Summary/Eigenvectors" – writes 2, prints:
    - Eigenvalue summary and eigenvectors at each RPM
    - Intermediate nonlinear bearing values at each iteration
  - "3..Both" – writes 3, prints combination of 1 and 2

- Freq Resp Analysis Type (SYNC) – option for frequency Response analysis
  - 0..Asynchronous – writes value of 0 to SYNC field
  - 1..Synchronous (Default) – writes value of 1 to SYNC field

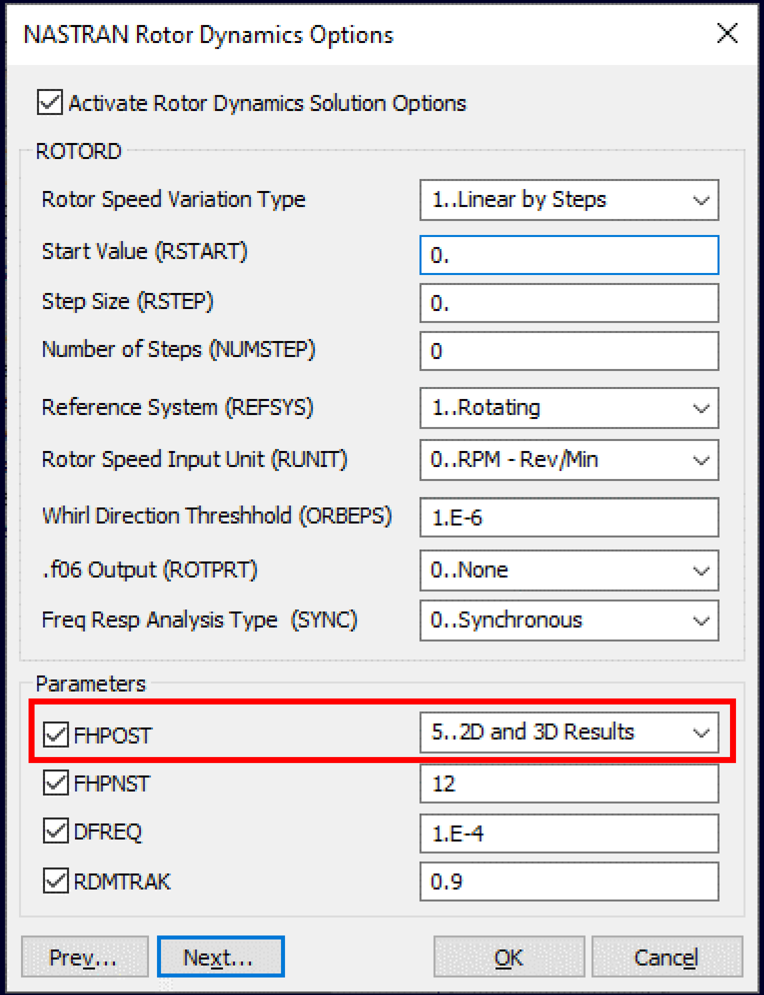


Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

Parameters section

- FHPOST – Specifies the results output option when the axisymmetric Fourier elements are requested with the FHAR Case Control Command by writing PARAM,FHPOST
  - “1..2D and 3D Results + 3D Recombination” – writes value of 1, which Requests 3D harmonic results for the axisymmetric Fourier elements at incremental angle locations around the complete 360 degree circumference by using FHPNST parameter to define the angle increments per 90 degrees. In addition, writes out all output requested by option “5..2D and 3D Results”
  - “4..2D Results” – writes value of 4, which requests 2D results on the modeled axisymmetric geometry
  - “5..2D and 3D Results” (Default) – writes value of 5, which requests 2D results on the modeled axisymmetric geometry and 3D results on the modeled solid element geometry



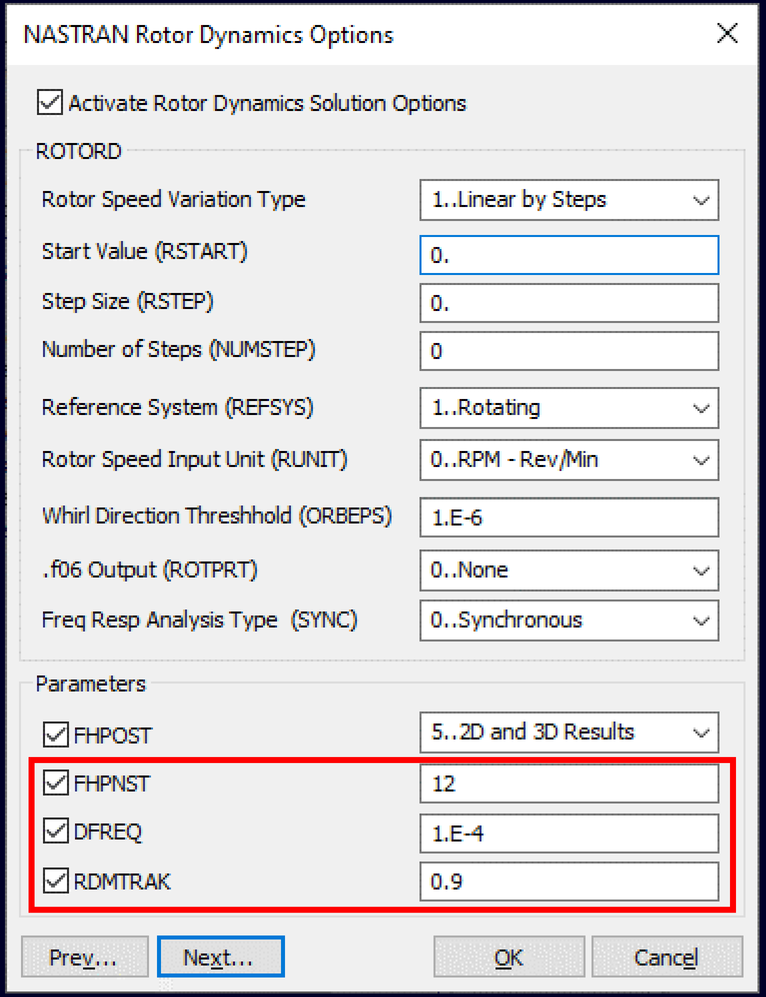
SIEMENS

Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

Parameters section

- FHPNST – Specifies the number of angle increments in a 90-degree quadrant in which the software computes the 3D harmonic results for axisymmetric Fourier elements, which allows users to view the non-axisymmetric results on a full revolute mode
  - Writes PARAM,FHPNST with entered value
  - Default value is 12, which is minimum value for FHPNST
  - Used in conjunction with “1..2D and 3D Results + 3D Recombination” option specified for FHPOST
- DFREQ – Specifies threshold for the elimination of duplicate frequencies (or rotational speeds) on all FREQi (or OMEGAi) entries, by writing PARAM,DFREQ with entered value
- RDMTRAK – Specifies threshold value for mode tracking in SOL 414,110, writes PARAM,RDMTRAK with entered value



SIEMENS

Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

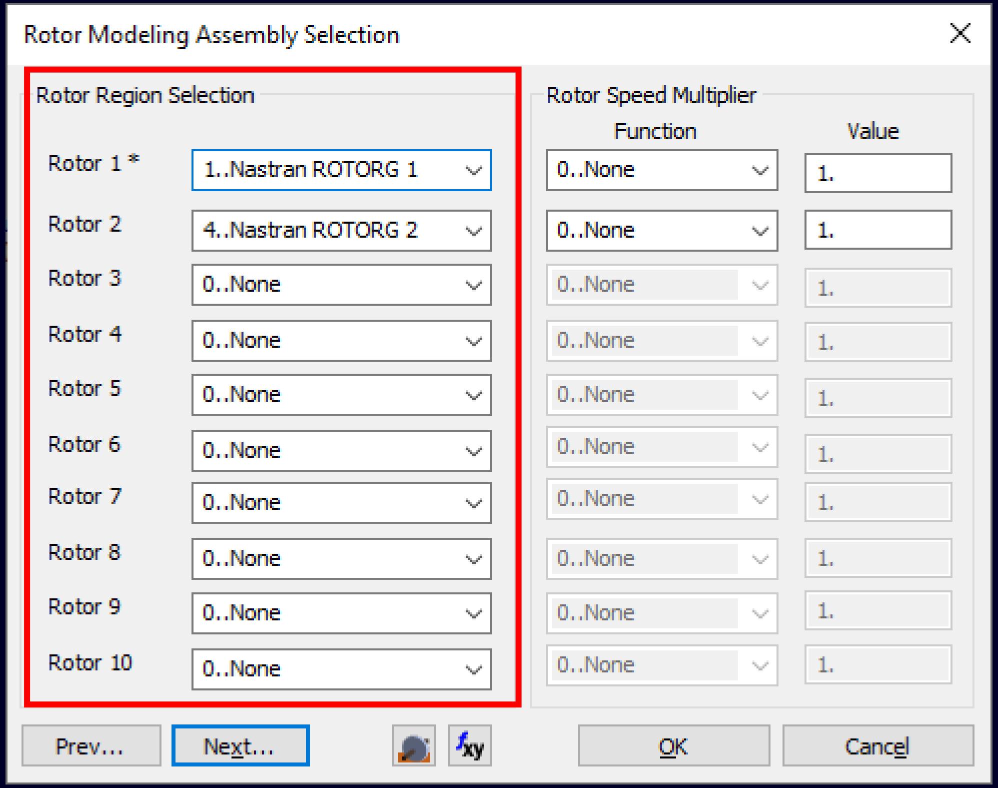
The *Rotor Modeling Assembly Selection* dialog box is used to select which Rotor Region(s) in model will be considered during the analysis, with minimum number of Rotors being 1 and maximum number being 10

Each *Rotor #*, *Function*, and *Value* combination is used to write lines 3 though 12, as needed, on the ROTORD entry

Rotor Region Selection section

- *Rotor 1* through *Rotor 10* – writes ID of selected Rotor Region to appropriate RIDi field

Note: *Rotation Axis (Z Axis)*, *Freq for Overall Damping (W3)*, *Freq for Material Damping (W4)*, and *Rotational Force Applied* on Rotor Region are written to RCORDi, W3_i, W4_i, and RFORCEi fields, as needed



SIEMENS

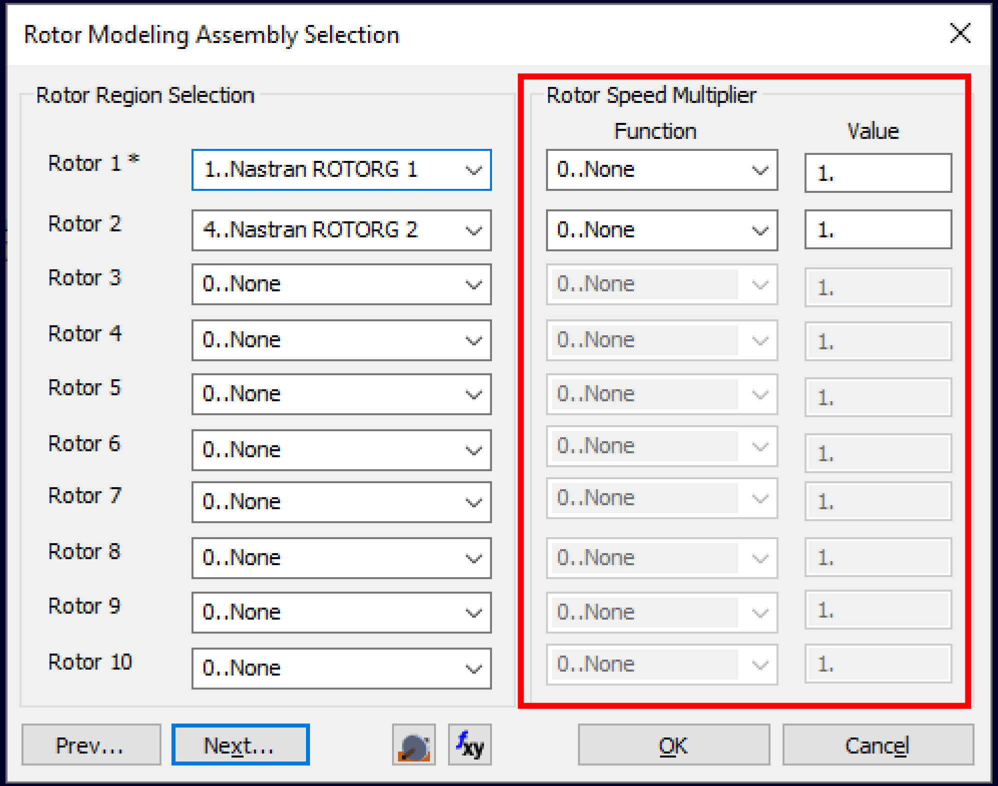
Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

Rotor Speed Multiplier section

- *Function* – when specified, writes ID of selected function to appropriate SPEEDi field as an integer
  - If Step Size (RSTEP) value in *NASTRAN Rotor Dynamics Options* dialog box is:
    - Non-Zero – defines values of the multiplier as a function of reference rotor speed
    - 0 – defines values of the rotor speed as a function of the sweeping parameter with sweeping parameter specified as OMEGAi for SOL 414,110, FREQi for SOL 414,111, or TSTEPi for SOL 414,129

- *Value* – if function is not specified, writes value to appropriate SPEEDi field as real number, which is a multiplier of all reference rotor speeds



SIEMENS

Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

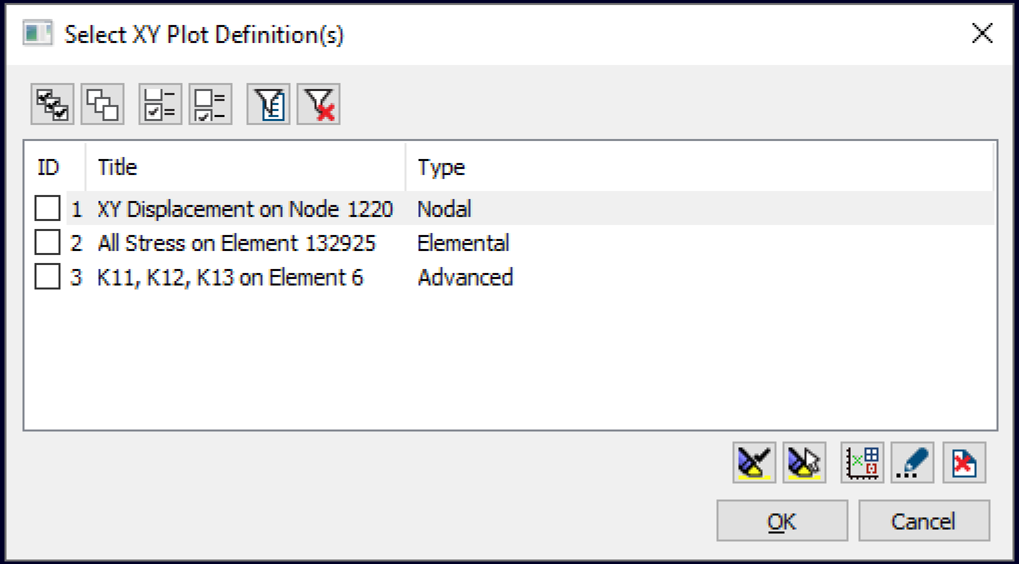
The *Select XY Plot Definitions* dialog box is used to select which XY Plot Definition entities will be written to the input file as OUTMGT entries

Available for:

- SOL 414,103 (Eigenvalue Analysis)
- SOL 414,111 (Harmonic Response)
- SOL 414,129 (Transient Response)

For convenience, this dialog box contains:

- Common icons used throughout Femap for selection and filtering of “named entities”
- Additional icons to highlight Nodes and Elements referenced by XY Plot Definition entities in the graphics window as well as create, edit, and delete XY Plot Definition entities



Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

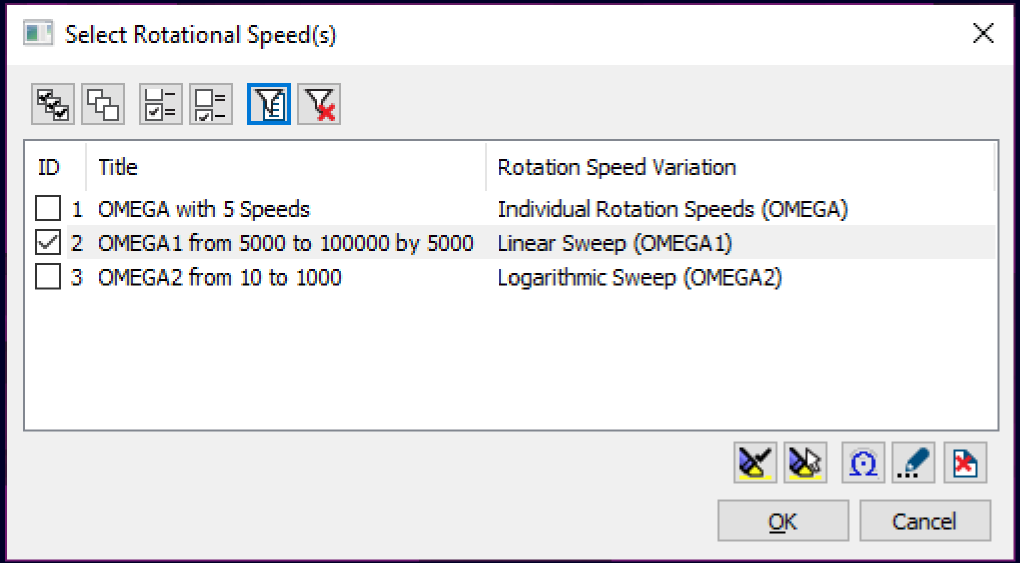
The *Select Rotational Speeds* dialog box is used to select which Rotational Speed entities will be written to the input file as OMEGA, OMEGA1, and/or OMEGA2 entries

Available for:

- SOL 414,110 (Complex Modal Analysis) once a Subcase with *Analysis Type* set to “2..Rotor Complex Modal” has been created

For convenience, this dialog box contains:

- Common icons used throughout Femap for selection and filtering of “named entities”
- Additional icons to create, edit, and delete Rotational Speed entities



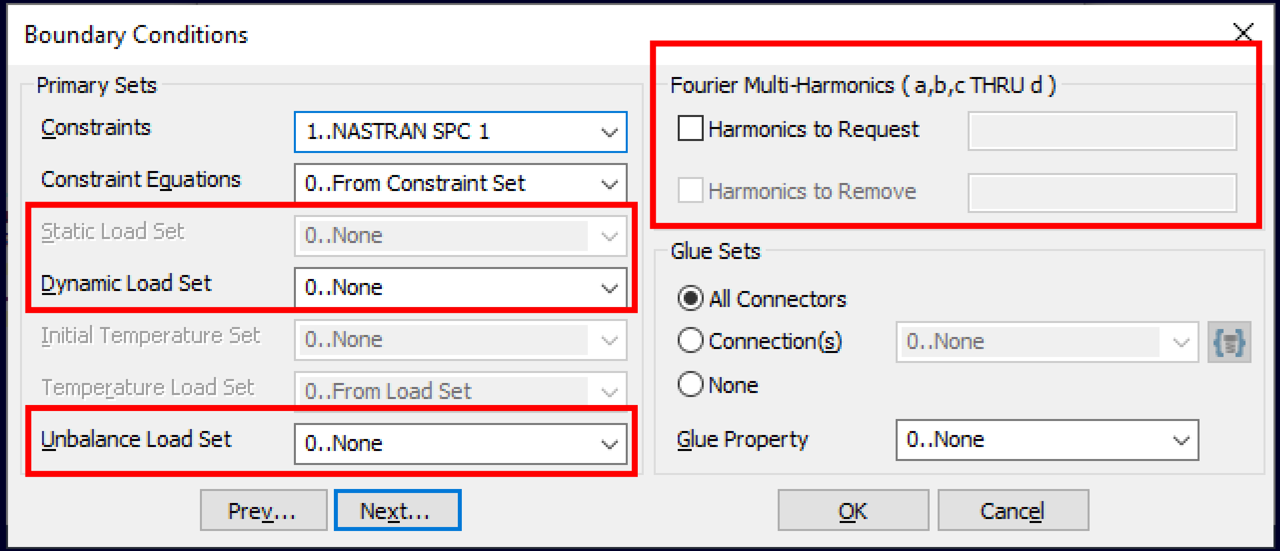
SIEMENS

Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

The *Boundary Conditions* dialog box has been updated to add Load Sets and other options specific for SOL 414 Analysis Types

- Static Load Sets – used to select a load set containing static loads
- Dynamic Load Set – used to select a load set containing dynamic excitations
- Unbalance Load Set – used to select a load set containing Unbalance Mass and/or Unbalance Moment loads
- Fourier Multi-Harmonics (a,b,c THRU d) – used to request/remove certain harmonics for an analysis (text entry)



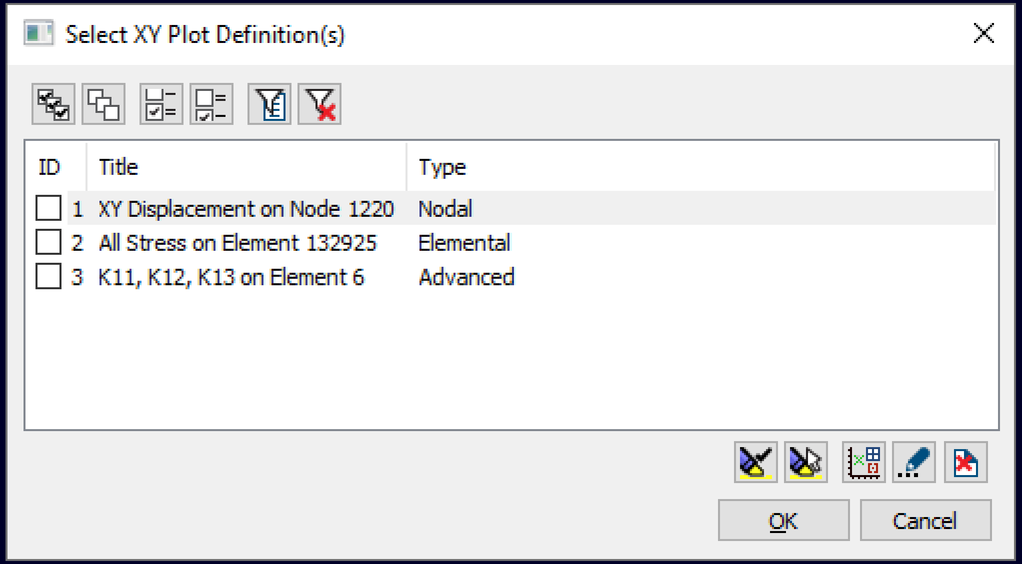
SIEMENS

Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

Available Input Summary:

- *Options* section
  - For all SOL 414 Analysis Types
    - *NASTRAN Executive and Solution Options*
    - *NASTRAN Bulk Data Options*
    - *NASTRAN Rotor Dynamics Options*
    - *Rotor Modeling Assembly Selection*
  - For all SOL 414 except SOL 414,101
    - *NASTRAN Dynamic Analysis*
  - For all SOL 414 except SOL 414,101 and SOL 414,110
    - *Select XY Plot Definition(s)*
  - For SOL 414,129 Only
    - Strategy Parameters (NLCNTLG) – Added *Axisymmetric Harmonic Coupling Options* specifically for this analysis type

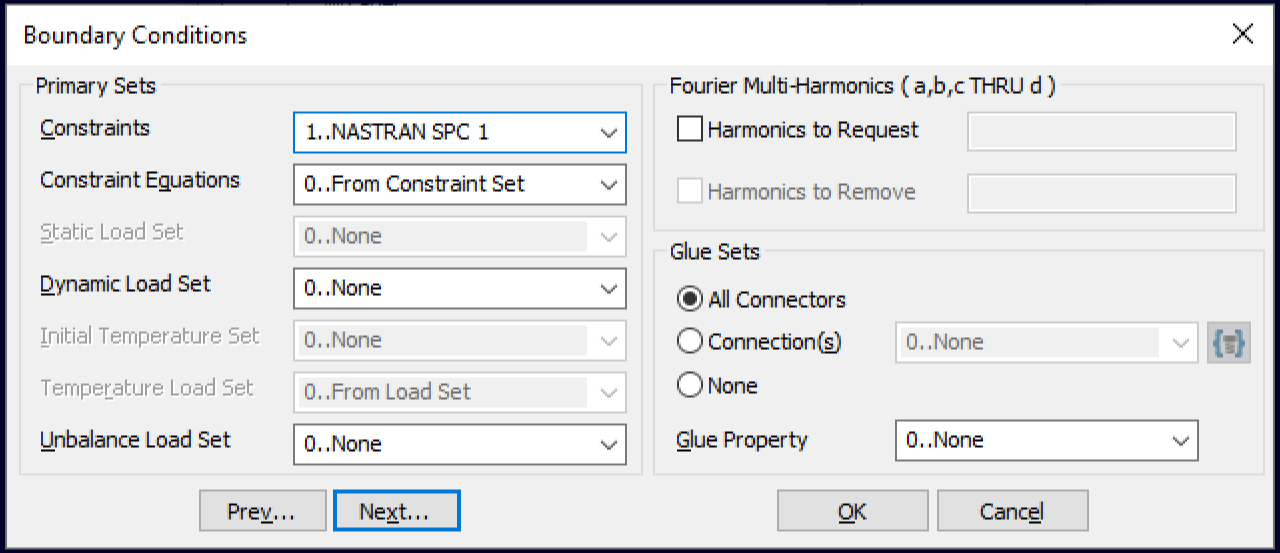


SIEMENS

Simcenter Femap 2306
Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

Available Input Summary:

- *Global Requests and Conditions* section
  - For all SOL 414 Analysis Types
    - *Boundary Conditions*
    - *NASTRAN Output Requests*
  - For SOL 414,129 Only
    - *Strategy Parameters (NLCNTL2)*



SIEMENS

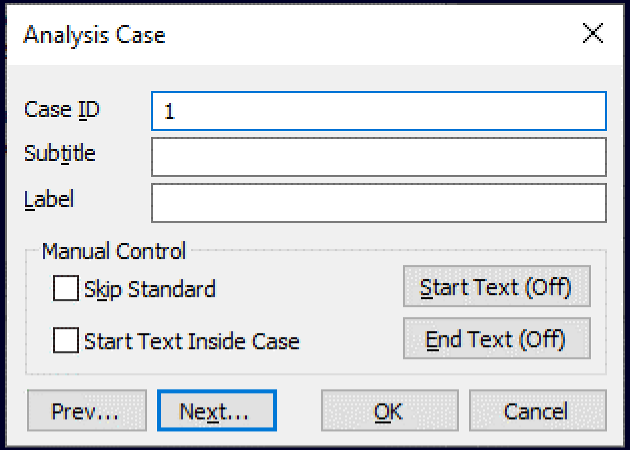
Simcenter Femap 2306
Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

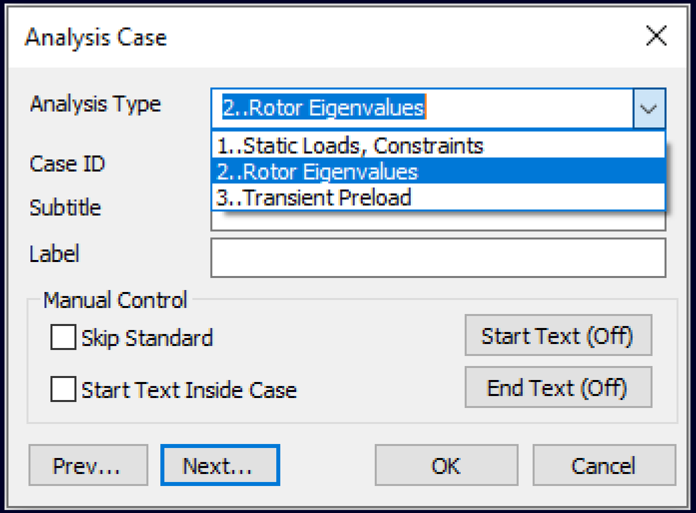
Available Input Summary (Subcase):

- For all SOL 414 Analysis Types
  - *Analysis Case* – see specific SOL 414 analysis types
  - *Boundary Conditions*
  - *NASTRAN Output Requests*

- For SOL 414,101 (Maneuvers)
  - *Analysis Case* – No Analysis Types – No additional dialog boxes

- For SOL 414,103 (Eigenvalue Analysis)
  - *Analysis Case* – 3 Analysis Types:
  - "1..Static Loads, Constraints"
  - "2..Rotor Eigenvalues" – *NASTRAN Modal Analysis*
  - "3..Transient Preload" – *Strategy Parameters (NLCNTL2) and Subcase # Time Step*





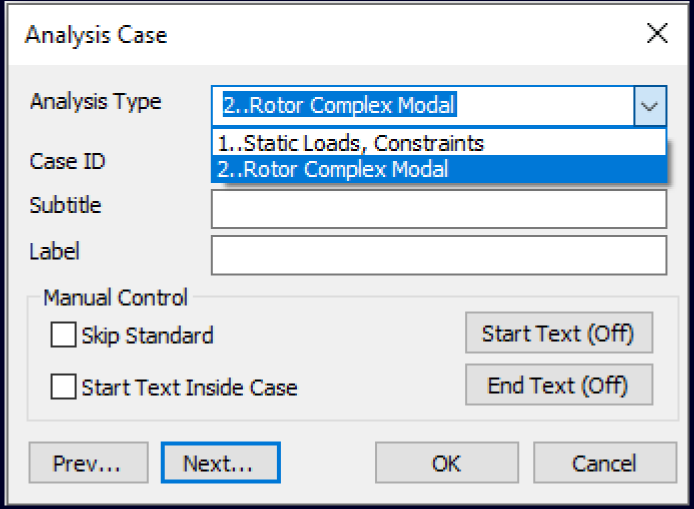
SIEMENS

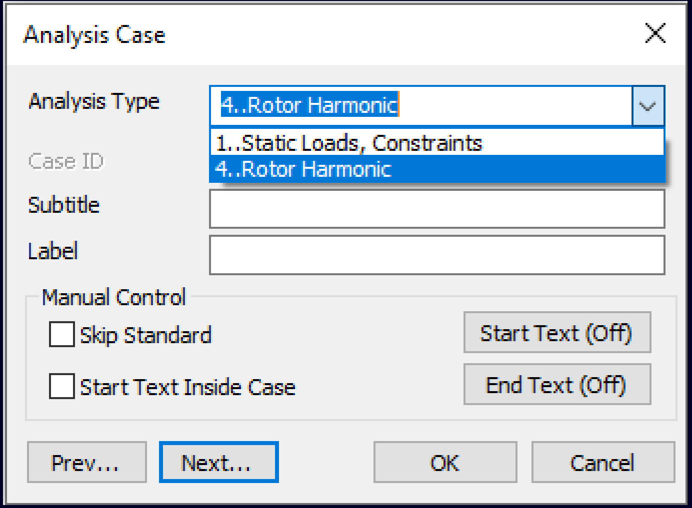
Simcenter Femap 2306

Analysis and Solver Support – Simcenter Nastran – Rotor Dynamics

Available Input Summary (Subcase):

- For SOL 414,110 (Complex Modal Analysis)
  - *Analysis Case* – 2 Analysis Types
    - “1..Static Loads, Constraints”
    - “2..Rotor Complex Modal” – *NASTRAN Modal Analysis* and *Select Rotational Speed(s)*
- For SOL 414,111 (Harmonic Response)
  - *Analysis Case* – 2 Analysis Types:
    - “1..Static Loads, Constraints”
    - “4..Rotor Harmonic” – *NASTRAN Dynamic Analysis*, *NASTRAN Modal Analysis*, and *Strategy Parameters (NLCNTL2)*
- For SOL 414,129 (Transient Response)
  - *Analysis Case* – No Analysis Types – *Strategy Parameters (NLCNTL2)* and *Subcase # Time Steps*





Simcenter Femap 2306

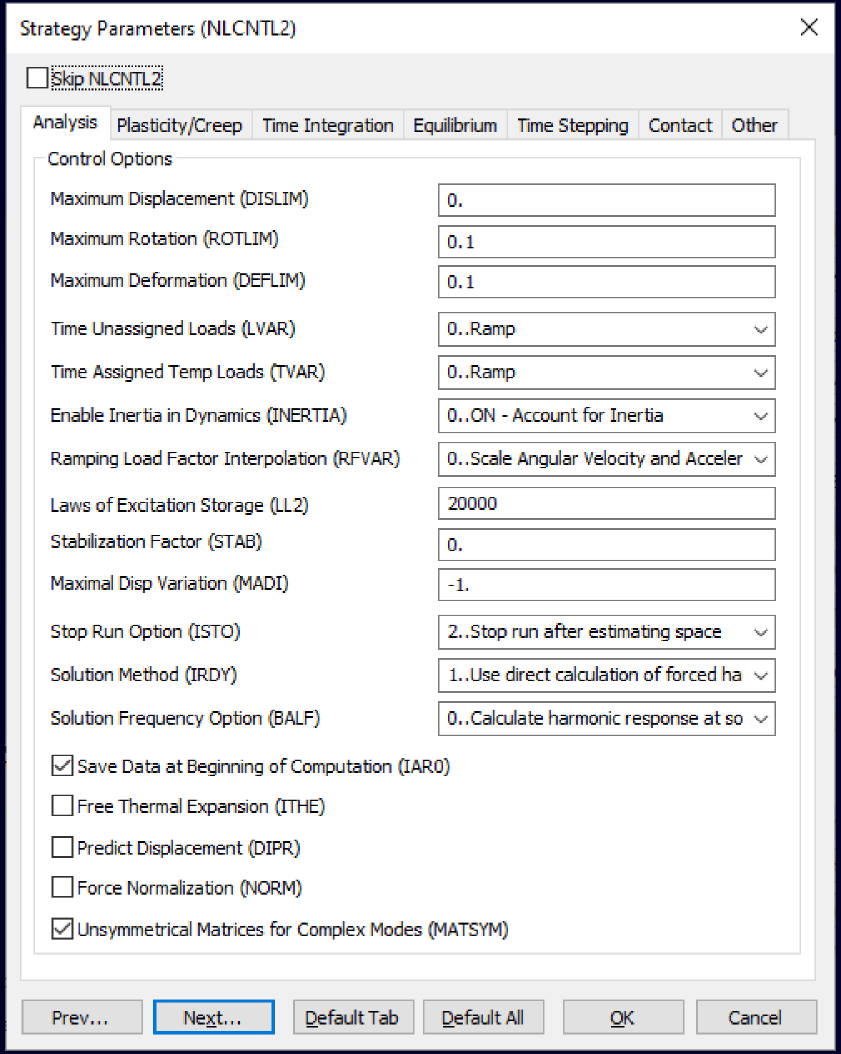
Analysis and Solver Support – Simcenter Nastran – SOL 402

Updated name of “Control Options” in Analysis Set Manager tree structure to “Strategy Parameters”

Changed name of dialog box to “Strategy Parameters (NLCNTL2)” and enhanced it to feature tabs (formally):

- *Analysis* (Analysis Control – 2 places)
- *Plasticity/Creep* (Plasticity and Creep Control)
- *Time Integration* (Time Integration)
- *Equilibrium* (Equilibrium Iteration and Convergence)
- *Time Stepping* (Time Stepping and Viscous Material Options)
- *Contact* (Contact)
- *Other* (Internal Restart and Diagnostic)

Added *Default Tab* and *Default All* buttons



SIEMENS

Simcenter Femap 2306
Analysis and Solver Support – Simcenter Nastran – SOL 402

Additions and Updates for Multi-Step Kinematics (SOL 402) in Analysis Set Manager

- *Strategy Parameters (NLCNTL2) – Analysis* tab
  - Added *Store Run Option (ISTO)* drop-down
  - Added *Solution Method (IRDY)* drop-down
  - Added *Solution Frequency Option (BALF)* drop-down
- *Strategy Parameters (NLCNTL2) – Equilibrium* tab
  - Added *Relaxed Force Tol (PRDR)* value
  - Added *Relaxed Energy Tol (PRDE)* value
- *Strategy Parameters (NLCNTL2) – Other* tab
  - Added *Storage Cycle for Element Stress (IA4)* value
  - Added *Complex Eigenvalue Frequency Step (IA12)* value

- *Strategy Parameter (NLCNTLG)*
  - Added *Check Initial Stiffness (KINI)* option – can be used in SOL 414 as well

SIEMENS

Updated name of “Control Options” in Analysis Set Manager tree structure to “Strategy Parameters”

Changed name of dialog box to “Strategy Parameters (NLCNTL2)” and enhanced it to feature tabs (formally):

- *Analysis* (Analysis Control – 2 places)
- *Plasticity/Creep* (Plasticity and Creep Control)
- *Time Integration* (Time Integration)
- *Equilibrium* (Equilibrium Iteration and Convergence)
- *Time Stepping* (Time Stepping and Viscous Material Options)
- *Contact* (Contact)
- *Other* (Internal Restart and Diagnostic)

Added *Default Tab* and *Default All* buttons

Simcenter Femap 2306

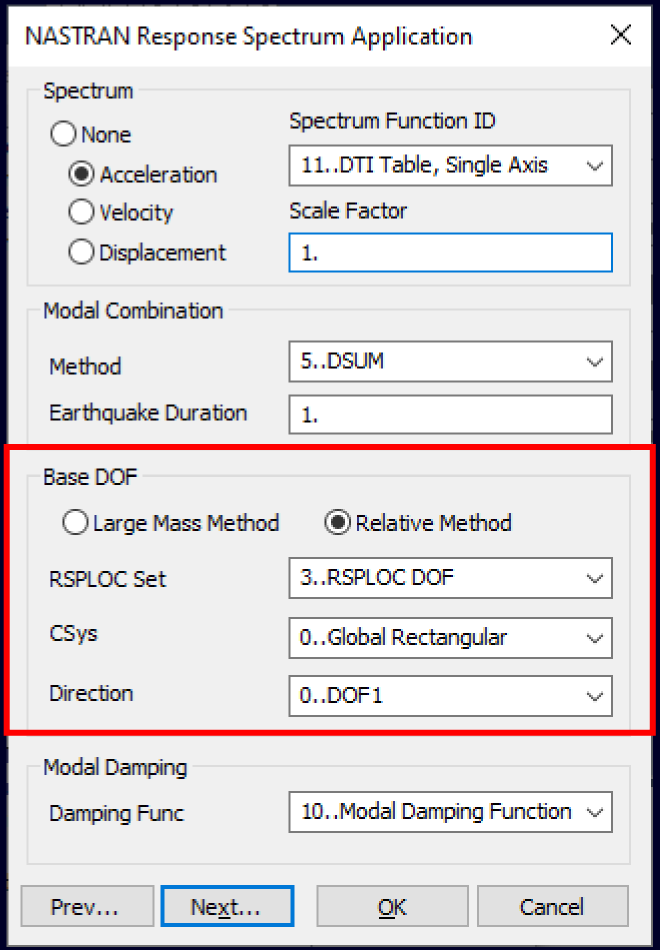
Analysis and Solver Support – Simcenter Nastran – Response Spectrum Application

Added support for Simcenter Nastran's ability to use the *Relative Method* instead of the *Large Mass Method* for *Base DOF*

Relative Method – writes RSPLOC entry

- *RSPLOC Set* – Constraint set containing nodes where spectrum is to be applied by writing nodes constrained in any DOF to the GIDi fields
- *CSys* – writes ID of coordinate system to CID field
- *Direction* – integer corresponding to selected option written to DIR field

When used, ID of RSPLOC is then written to a RSPLOIDi field on the RSAPPLY entry (support added in Femap 2301)



SIEMENS

Simcenter Femap 2306
Analysis and Solver Support – Nastran and ABAQUS

Simcenter Nastran and MSC Nastran
- Added support to write/read Femap Layup Titles to/from comments in input files
- Added support to write 0.0 for stiffness values to Spring/Damper properties (“K” values on PBUSH) as this is viable for these solvers

ABAQUS
- Improvements to exporting load and constraint definitions
  - Consistently create *NSET and *ELSET from load definitions and constraint definitions
  - Create accurate *NSET and *ELSET for load definitions and constraint definitions applied on different geometric entities such as points, curves, and/or surfaces
  - Appropriately export load Definitions on elements, curves, and points are as *CLOAD and *DLOAD
  - *NSET and *ELSET are referenced in *BOUNDARY, *CLOAD, and *DLOAD to reduce input file size

- Improvements to exporting and importing *TIE keyword

Simcenter Femap 2306

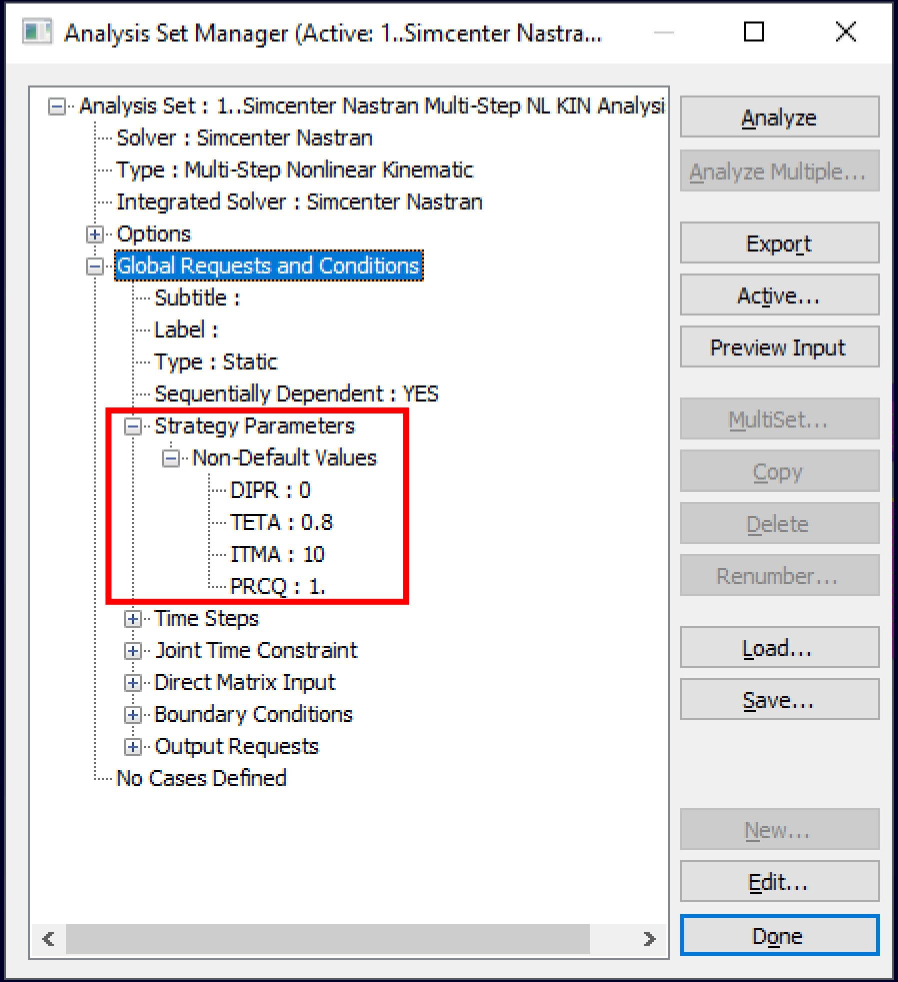
Analysis and Solver Support – Analysis Set Manager

Analysis Set Manager

Added *Non-Default Values* branch under *Strategy Parameters* which only displays non-default values specified in the *Strategy Parameters (NLCNTL2)* dialog box for Analysis Types:

- “28..Multi-Step Nonlinear Kinematic” (SOL 402)
- “38..Rotor Dynamics Transient Response” (SOL 414,129)

Note: Similar functionality is planned for other Analysis Types in future releases, beginning with “27..Multi-Step Structural” (SOL 401)



What’s new

Simcenter Femap 2306

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

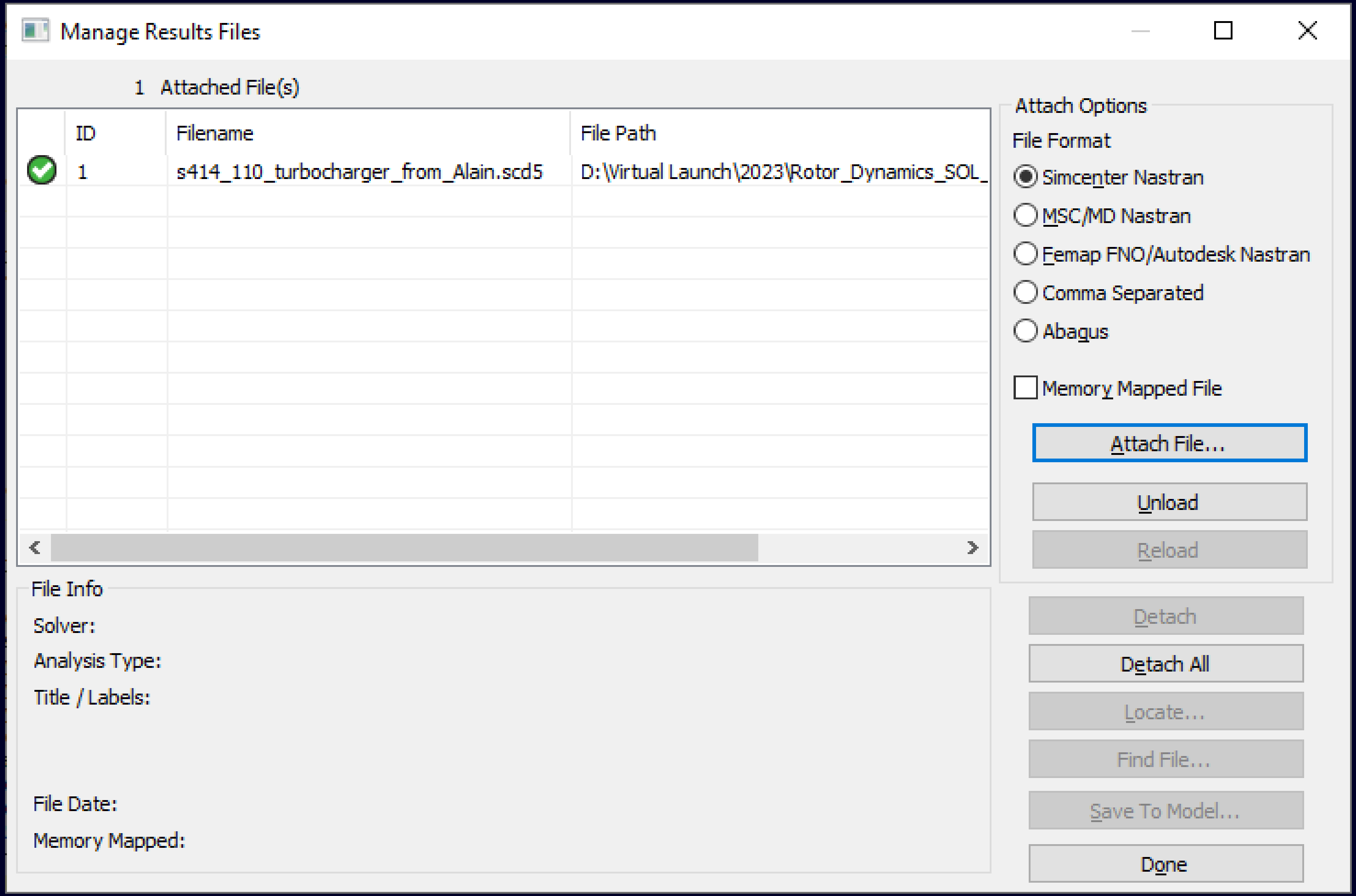
Miscellaneous and API

SIEMENS

Simcenter Femap 2306
Postprocessing – Support for results from Simcenter Nastran SOL 414 analysis types

Overview of working with results generated by Simcenter Nastran SOL 414 analysis

- VERY IMPORTANT: SOL414 does NOT generate an OP2 results file, thus support was added for attaching to the Simcenter Data File (SCD5) generated by SOL 414
  - SCD5 is based on the HDF5 file format
  - SCD5 is the only output file which can currently be used for post-processing SOL414 results in Femap



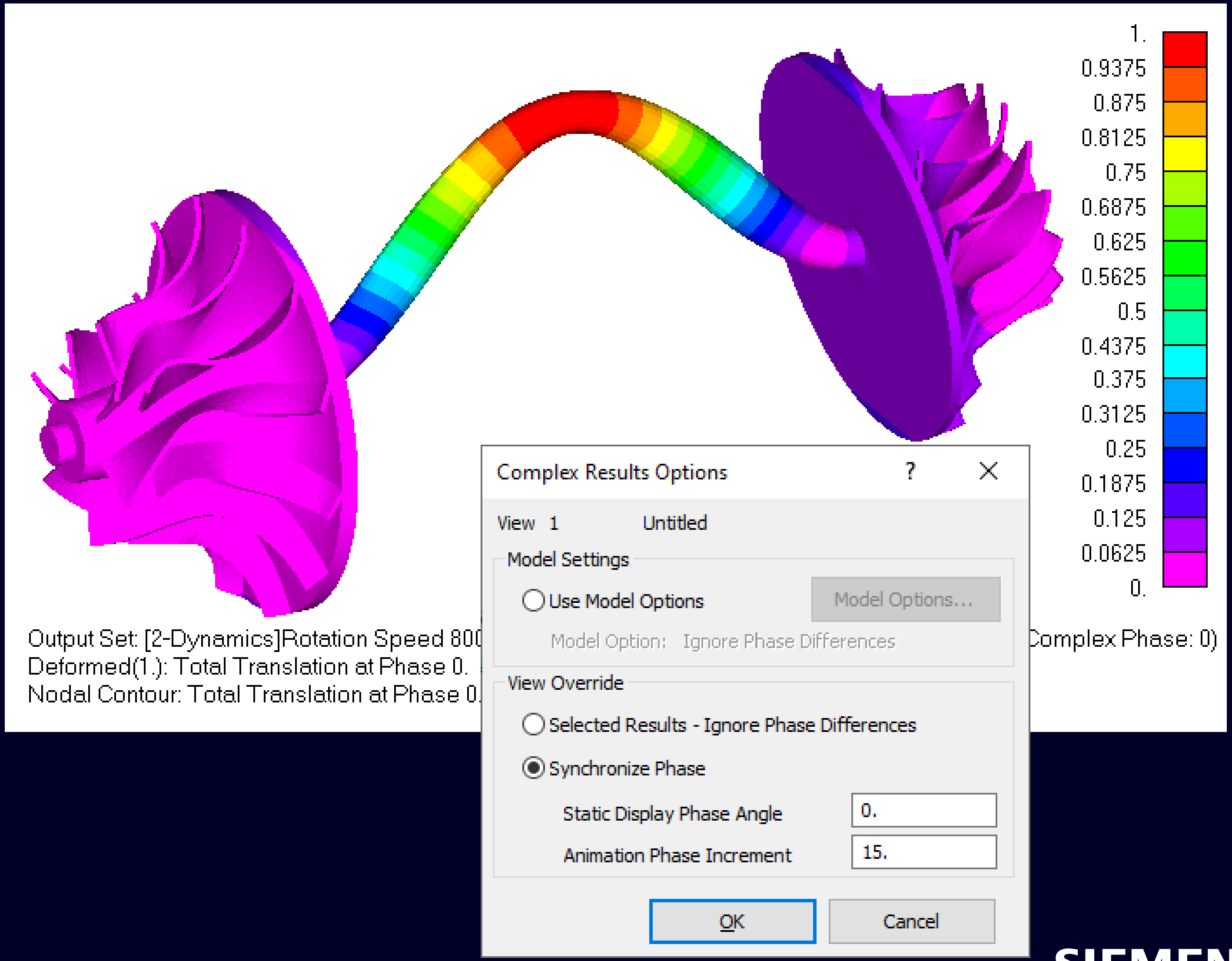
SIEMENS

Simcenter Femap 2306

Postprocessing – Support for results from Simcenter Nastran SOL 414 analysis types

The SCD5 results file contains two types of results which are stored in Femap:

- Results for nodes/elements which are stored within Output Sets as Output Vectors which can be plotted in the graphics window
  - Typically, the results from SOL 414 are displayed a contoured plot, either deformed or animating, based on options specified in Complex Results dialog box



SIEMENS

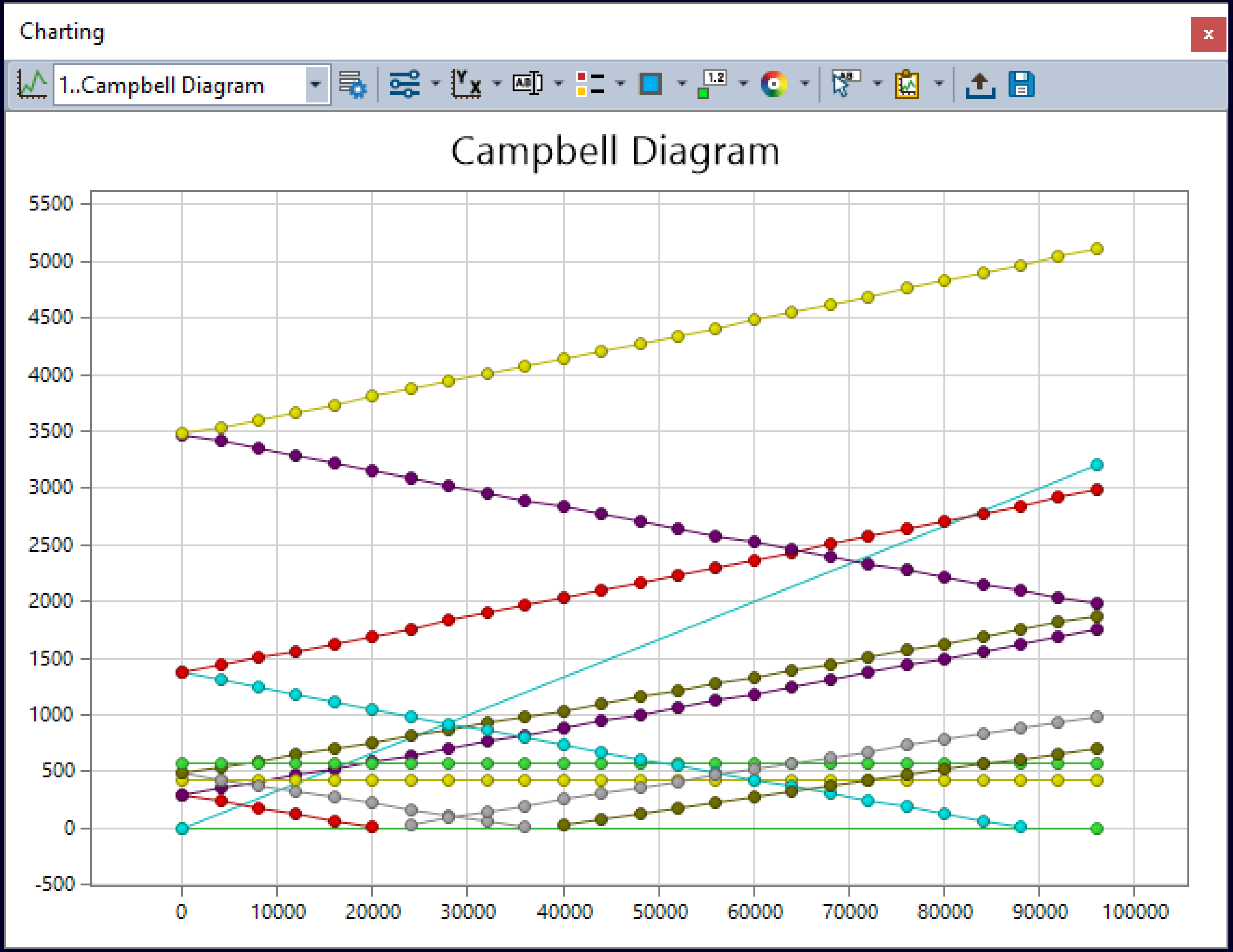
Simcenter Femap 2306

Postprocessing – Support for results from Simcenter Nastran SOL 414 analysis types

The SCD5 results file contains two types of results which are stored in Femap:

- Results in XY format which are stored as Femap functions which can be viewed in the Charting pane
  - Typically, specific functions are selected to generate a "Campbell Diagram", which can be used to determine critical speeds, among other things.

- Lines with 2 points originating at 0,0 are 0P (Green) and 2P (Cyan)
- All other lines are "EigenFrequency vs Rotation Speed" for various modes



SIEMENS

What's new

Simcenter Femap 2306

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

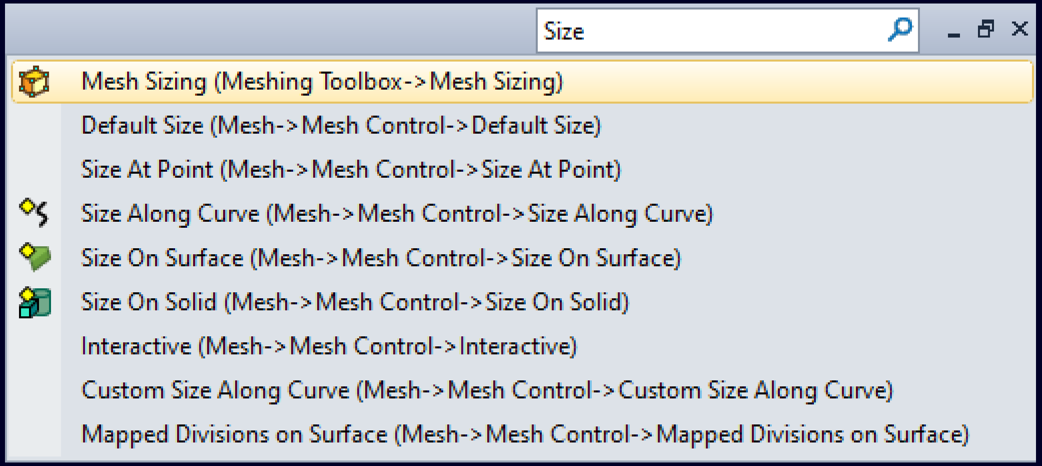
Simcenter Femap 2306

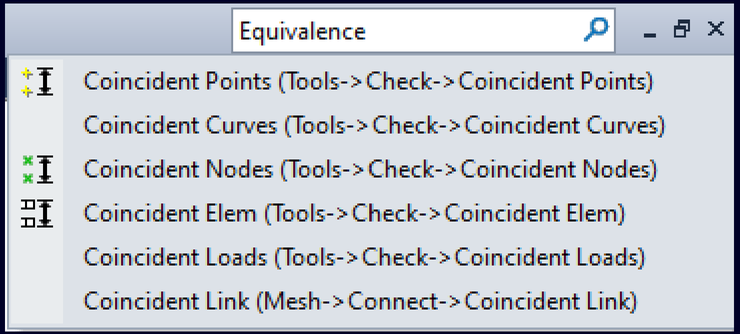
Miscellaneous – Command Finder Enhancements

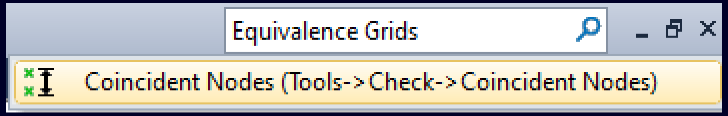
Command Finder Enhancements

- Search now includes commands found in Dockable Panes
  - If selected command is located within a Pane, the Pane becomes visible
  - If selected command is located with a particular “Tool” within a “Toolbox”, that tool will be opened and “pulsate” a few times to draw user’s attention

- Search now includes “keywords” which are known to be the nomenclature found in other CAE applications
  - For instance, “Equivalence” and “Grids”







SIEMENS

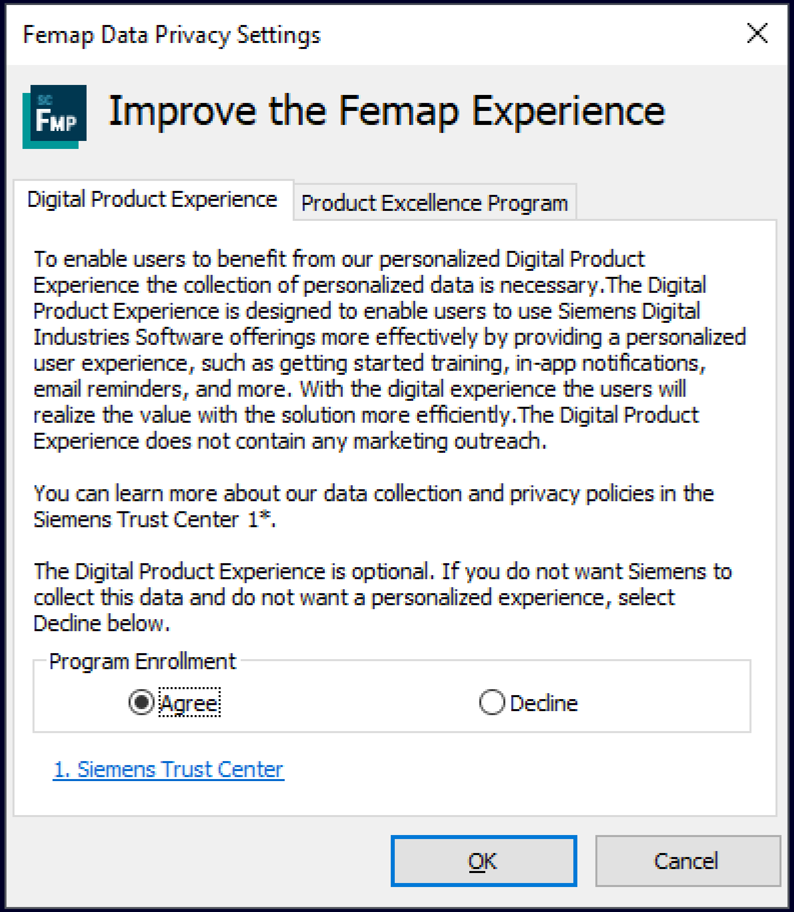
Simcenter Femap 2306
Miscellaneous – Digital Product Experience & Product Excellence Program

Help Improve the Femap Experience by choosing Program Enrollment options in Help -> Data Privacy

Enhancements to comply with new data privacy laws

Femap collects information about how our customers use the product to help improve product features and user experience

- No proprietary or personally identifiable information is collected
- Participation is completely optional
- Does not affect product performance



SIEMENS

Simcenter Femap 2306
API – New functionality

Added GeometryInterface Object to provide access to all Geometry translators using a single Object

Rotor Dynamics

- Added Objects for new Rotor Dynamics entity types (XY Plot Definition, Bearing Speed Definition, and Rotational Speed Definition)

- Updated Element and Property Objects to support Bearing, Gear, and Fourier Element/Property types

- Updated Analysis Manager and Analysis Case Objects for new options, dialog boxes, etc

- Updated LoadMesh Object for unbalance loads

Simcenter Femap 2306
API – New functionality

Added feFileMessageSelectAll, feFileMessageClear, and feFileMessageSave2 to perform actions in the Messages dockable pane

Added feSurfaceRuled2 to access all available options of the Geometry -> Surface -> Ruled / Between Curves command

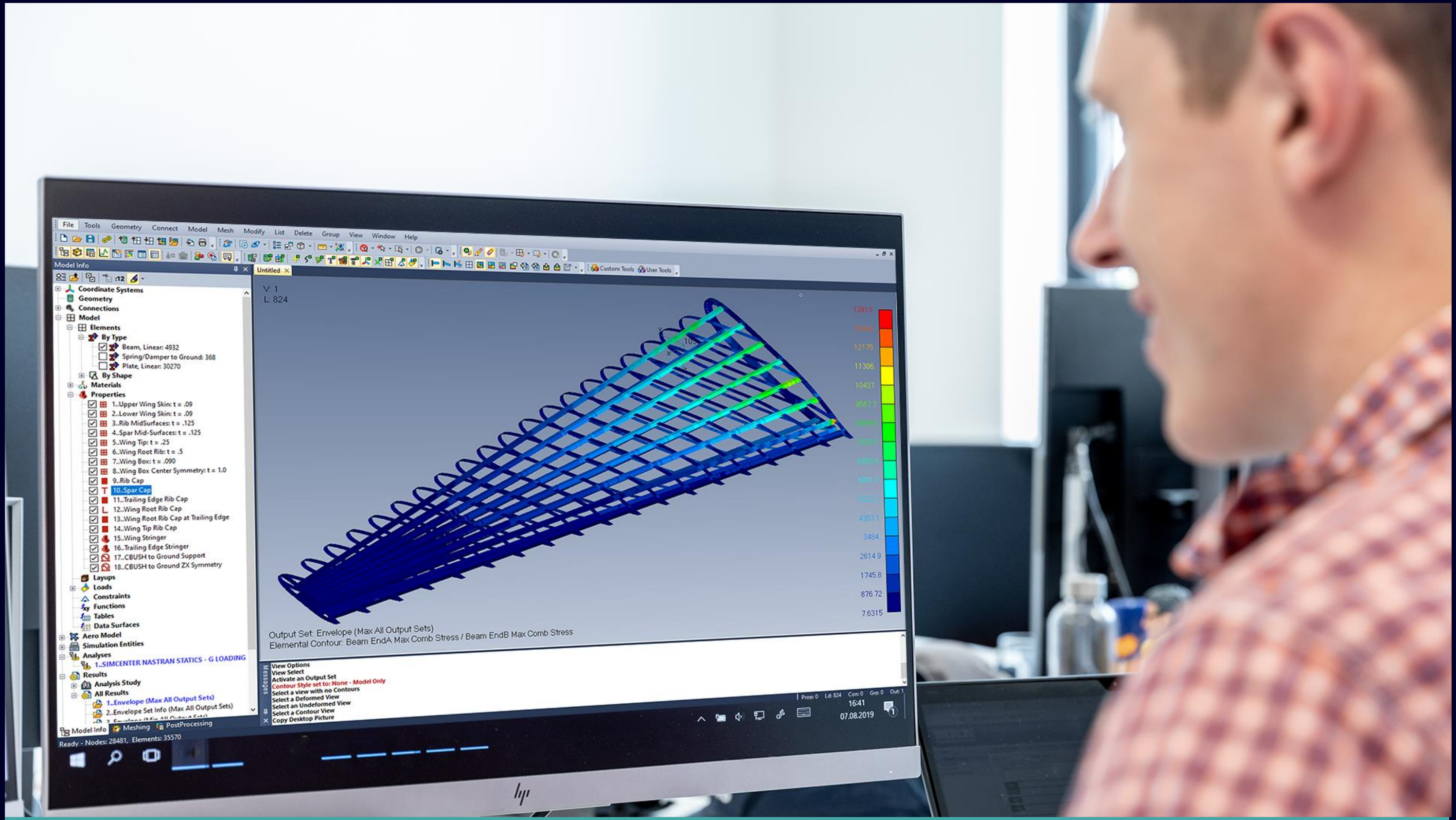
Added feMeshSizeSurface2 to access all available options of the Mesh -> Mesh Control -> Size On Surface command

Added Pref_ViewLogoTransparent to set the “Transparent Logo” option on the “Views” tab of File -> Preferences

SIEMENS

Try Simcenter Femap for free

30-day free trial



www.femap.de

SIEMENS

Join the Simcenter Femap Community

Explore – Browse our different **blogs** dedicated to each solution, read our articles and hear about the latest news.

Share – Ask questions in our **forum**, get answers, connect with other users, and benefit from their experience.

Learn – Find the information you need in the **knowledge base** and improve your skills.

siemens.com/plm/community/simcenter

SIEMENS