Ansys Mechanical Linear and Nonlinear Dynamics

Module 09: Transient Analysis

Release 2022 R2

Please note:
- These training materials were developed and tested in Ansys Release 2022 R2. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.

Module 09 Learning Outcomes

- After completing this module, you will:
  - Be able to incorporate non-linear behavior within dynamic analysis.
  - Understand the differences between the two transient solution methods and be able to apply the one most-suited to your application.
  - Learn how to choose an appropriate solution time step to ensure accurate response calculations.
  - Be able to properly account for the initial displacement/velocity/acceleration conditions that may be present at the start of your transient simulation.
  - Understand how damping is accounted for in transient analysis.

Module 09 Topics

A. Definition and Purpose

B. Solution Methods

1) Full Method
2) Mode-Superposition Method

C. Nonlinearities

D. The Newton-Raphson Method

- Load Steps and Substeps
- Equilibrium Iterations

E. Analysis Settings—Full Method

1) Step Controls and Time Step
2) Solver Controls
3) Nonlinear Controls
4) Output Controls
5) Damping Controls

F. Initial Conditions

G. Loads and Supports

H. Analysis Settings—MSUP Method

1) Step Controls
2) Options
3) Damping Controls

I. Pre-Stressed MSUP Method

A. Definition and Purpose

- Transient dynamic analysis is a technique used to determine the dynamic response of a structure under the action of any general time-dependent loads.
  - Also known as time-history analysis or transient structural analysis.
  - Can include inertia and/or damping effects.
  - Can include any type of nonlinear effects, such as contact, plasticity, large deflection, hyperelasticity, etc.
  - Use it to determine time varying displacements, stresses, strains and forces.
- Typically, more involved than a static analysis
  - Generally, requires more computer resources and more "engineering" time.

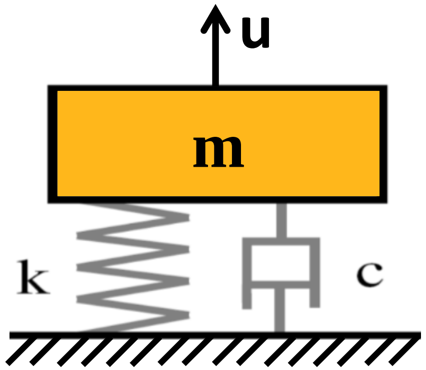
… Definition and Purpose

- Hints for preliminary work to understand the physics of the problem:
  - <u>Simplify:</u> Analyze a simpler model first to provide good insight into the behavior at minimal cost.
    - Consider using beams, shells, lumped masses.
  - <u>Nonlinearities:</u> If nonlinearities exist, first assess their affect using a static analysis.
    - It may be possible to eliminate nonlinearities in the dynamic analysis.
  - <u>Modal Analysis:</u> Assess dynamic behavior using modal analysis to determine the natural frequencies and mode shapes.
    - Natural frequencies are also useful for calculating the correct integration time step.

… Definition and Purpose

- The nonlinear governing equation for the Transient Dynamic Analysis is:

$$\overbrace{[M]\{\ddot{u}\}}^{F_{\text{inertia}}} + \overbrace{[C]\{\dot{u}\}}^{F_{\text{damping}}} + \overbrace{[K(u)]\{u\}}^{F_{\text{stiffness}}} = \overbrace{\{F(t)\}}^{F_{\text{applied}}}$$



[M]: is structural mass matrix

[C]: is structural damping matrix

[K]: is structural stiffness matrix

{F}: is the load vector

$\{\ddot{u}\}$: is nodal acceleration vector

$\{\dot{u}\}$: is nodal velocity vector

{u}: is nodal displacement vector

(t): is time

… Definition and Purpose

$$\overbrace{[M]\{\ddot{u}\}}^{F_{\text{inertia}}}+\overbrace{[C]\{\dot{u}\}}^{F_{\text{damping}}}+\overbrace{[K(u)]\{u\}}^{F_{\text{stiffness}}}=\overbrace{\{F(t)\}}^{F_{\text{applied}}}$$

- At any given time, t these equations may be thought of as a set of "static" equilibrium equations that take into account:
  - inertia forces $[M]\{\ddot{u}\}$ and
  - damping forces $[C]\{\dot{u}\}$.
- To solve these equations, Ansys uses:
  - the Newmark time integration method or
  - an improved method called HHT
- Integration time step: the time increment between successive time points.

$$\Delta t = t_n - t_{n-1}$$

… Definition and Purpose

- The Newmark method uses finite difference expansions on the time interval Δt, with a primary aim of computing displacements $\{u_{n+1}\}$:

$$\{\dot{u}_{n+1}\} = \{\dot{u}_n\} + [(1-\delta)\{\ddot{u}_n\} + \delta\{\ddot{u}_{n+1}\}]\Delta t$$

$$\{u_{n+1}\} = \{u_n\} + \{\dot{u}_n\}\Delta t + [(0.5-\alpha)\{\ddot{u}_n\} + \alpha\{\ddot{u}_{n+1}\}]\Delta t^2$$

- $\{u_{n+1}\}$ can then be obtained such that,

$$(a_0[M] + a_1[C] + [K])\,\underline{\{u_{n+1}\}} = \{F^a_{n+1}\} + [M](a_0\{u_n\} + a_2\{\dot{u}_n\} + a_3\{\ddot{u}_n\})$$
$$+[C](a_1\{u_n\} + a_4\{\dot{u}_n\} + a_5\{\ddot{u}_n\})$$

where integration constants a_0 through a_5 are functions of γ and Δt.

- γ represents numerical damping and can be directly input in the analysis settings
- γ is a function of two Newmark parameters, α and δ

$$\delta = \frac{1}{2} + \gamma$$
$$\alpha = \frac{1}{4}(1+\gamma)^2$$
$$\gamma \geq 0$$

… Definition and Purpose

- The HHT Method attempts to overcome some shortcomings of the Newmark Method:
  - In low-frequency modes, the Newmark method fails to retain accuracy as $\delta > ½$
  - The lack of numerical damping can produce unacceptable levels of numerical noise at higher frequencies

- The HHT Method damps out spurious high frequency noise while maintaining accuracy and solves for $\{u_{n+1}\}$ with:

$$(a_0[M]+a_1[C]+(1-\alpha_f)[K])\{u_{n+1}\}=(1-\alpha_f)\{F^a_{n+1}\}+\alpha_f\{F^a_n\}-\alpha_f[K]\{u_n\}+$$
$$[M](a_0\{u_n\}+a_2\{\dot{u}_n\}+a_3\{\ddot{u}_n\})+[C](a_1\{u_n\}+a_4\{\dot{u}_n\}+a_5\{\ddot{u}_n\})$$

where once again integration constants a_0 through a_5 are functions of γ, Δt, α, and δ.

$$\alpha = \frac{1}{4}(1+\gamma)^2 \qquad \delta = \frac{1}{2}+\gamma$$

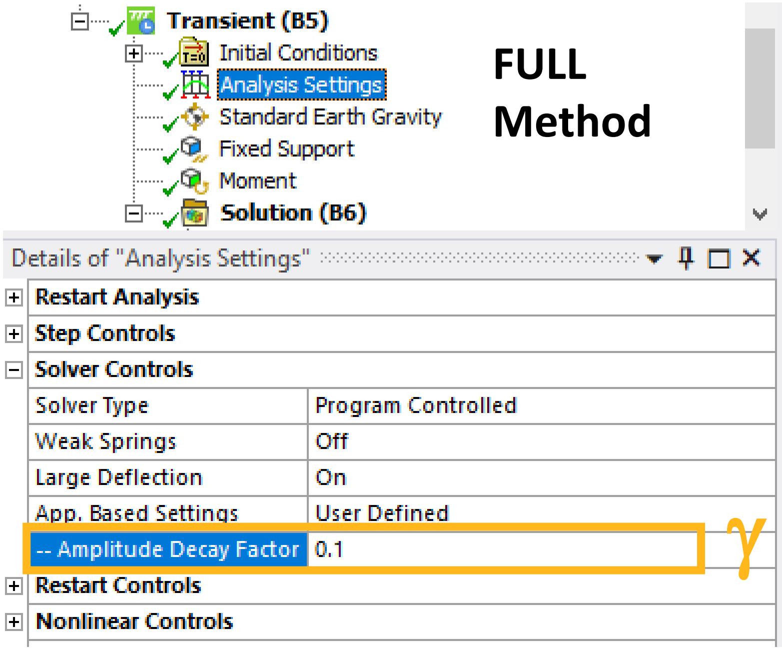
$$\alpha_f = \gamma \geq 0 \qquad \alpha_m = 0$$

… Definition and Purpose

- In Ansys Mechanical, the HHT parameters are calculated using:

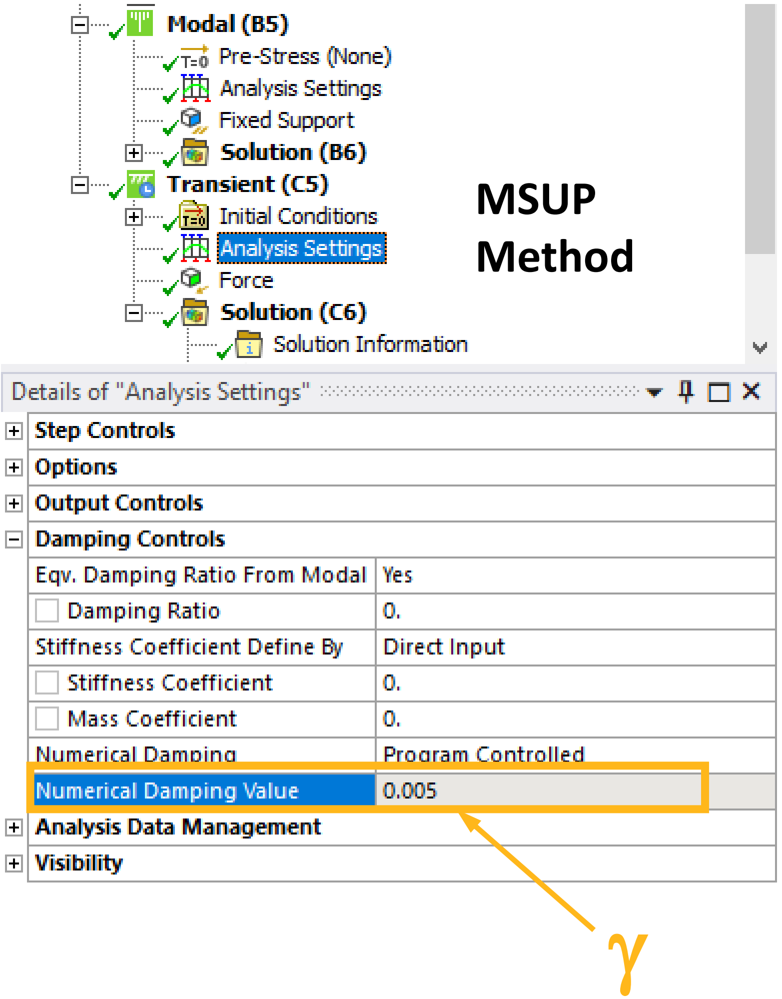
$$\alpha = \frac{1}{4}(1+\gamma)^2, \delta = \frac{1}{2}+\gamma, \alpha_f = \gamma \geq 0, \alpha_m = 0$$

- γ is a numerical damping value (amplitude decay factor).



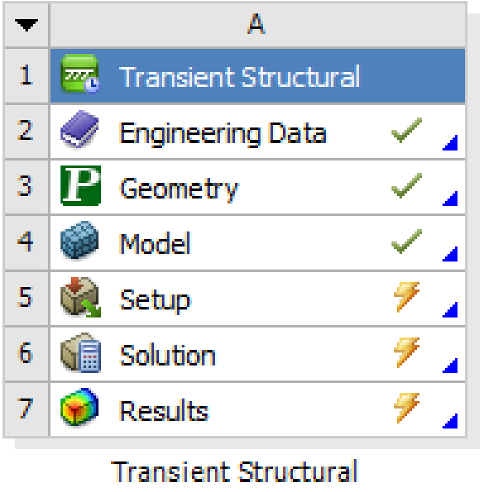

```
                    L O A D   S T E P   O P T I O N S

LOAD STEP NUMBER. . . . . . . . . . . . . . . . .     2
TIME AT END OF THE LOAD STEP. . . . . . . . . . . 0.75000E-02
AUTOMATIC TIME STEPPING . . . . . . . . . . . . .    ON
   INITIAL NUMBER OF SUBSTEPS . . . . . . . . . .    40
   MAXIMUM NUMBER OF SUBSTEPS . . . . . . . . . .  1000
   MINIMUM NUMBER OF SUBSTEPS . . . . . . . . . .    40
MAXIMUM NUMBER OF EQUILIBRIUM ITERATIONS. . . . .    15
STEP CHANGE BOUNDARY CONDITIONS . . . . . . . . .   YES
STRESS-STIFFENING . . . . . . . . . . . . . . . .    ON
TRANSIENT (INERTIA) EFFECTS
   STRUCTURAL DOFS. . . . . . . . . . . . . . . .    ON
TRANSIENT INTEGRATION PARAMETERS
   GAMMA. . . . . . . . . . . . . . . . . . . . . 0.10000
   ALPHA. . . . . . . . . . . . . . . . . . . . . 0.30250
   DELTA. . . . . . . . . . . . . . . . . . . . . 0.60000
   USING HHT TIME INTEGRATION
   ALPHAF . . . . . . . . . . . . . . . . . . . . 0.10000
   ALPHAM . . . . . . . . . . . . . . . . . . . .  0.0000
TERMINATE ANALYSIS IF NOT CONVERGED . . . . . . .YES (EXIT)
CONVERGENCE CONTROLS. . . . . . . . . . . . . . .USE DEFAULTS
```



B. Solution Methods

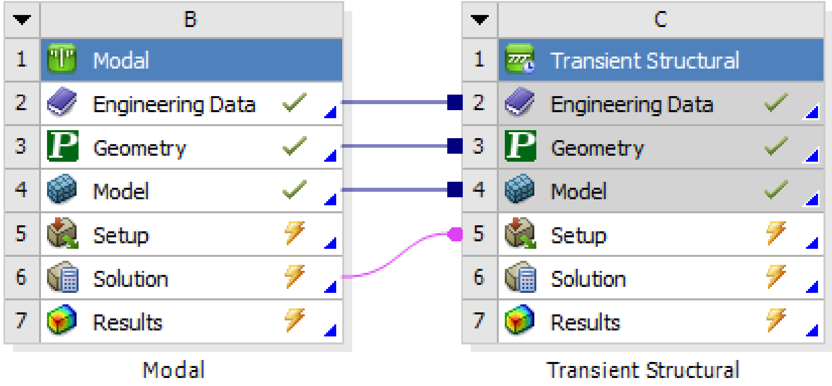
1. <u>Full Method</u>
   - Solves full equations of motion
   - Allows all types of nonlinearities
   - Accepts most load types (e.g., nodal forces, non-zero displacements, element loads, tabular boundary conditions, etc.)
   - Uses full matrices [K, M, and C]
   - Requires that mesh be fine enough to resolve the highest mode of interest

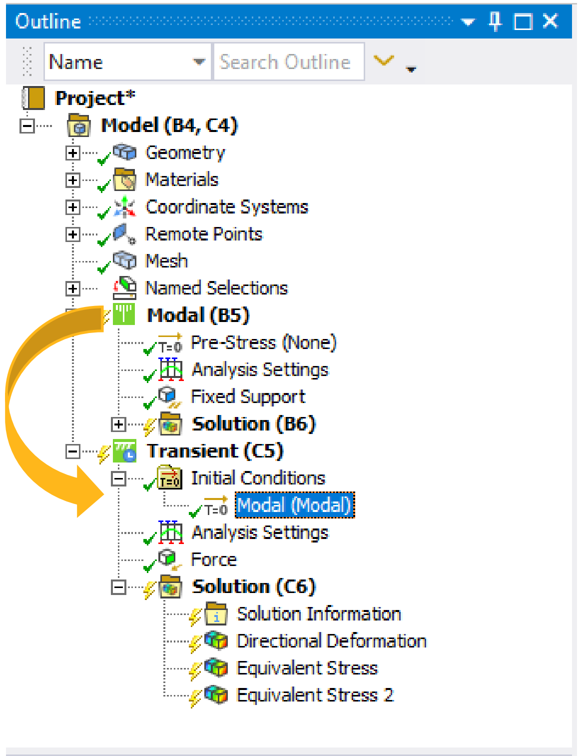


... Solution Methods

2. Mode-Superposition (MSUP) Method

- Faster and less expensive than full method
- Allows damping as a function of frequency
- Uses the natural frequencies and mode shapes from a linked modal analysis to characterize the transient dynamic response of a structure
- Scales the mode shapes obtained from a modal analysis and sums them to calculate the dynamic response
- Set up an MSUP transient analysis in the Project Schematic by linking a modal system to a transient structural system at the solution level.
- Notice, in the transient branch, that the modal analysis result becomes an initial condition.

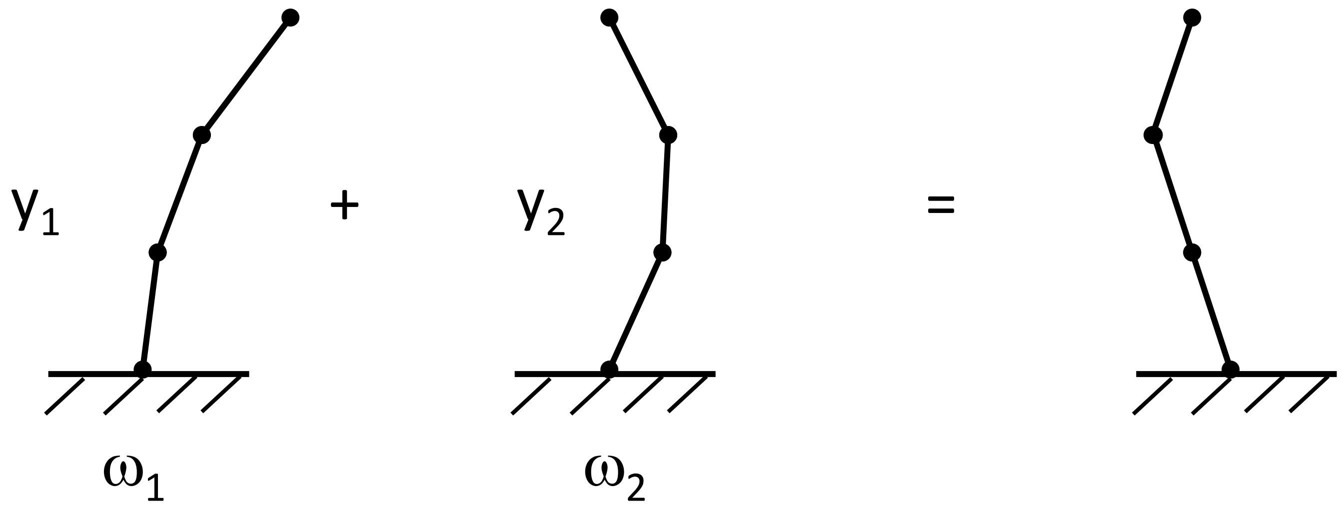




… Solution Methods

… MSUP Method

- Example:



  - Here, the sum of mode shape 1 and mode shape 2 approximates the final response. Since mode shapes are relative, the coefficients y_1 and y_2 are required.
  - Mode shapes (eigenvectors) are also known as generalized coordinates, and in this case, coefficients y_1 and y_2 are the DOFs.

… Solution Methods

… MSUP Method

- The equations of motion are *(linear only)*:

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \{F(t)\}$$

- Instead of using nodal coordinates, generalized coordinates will be used. Assume that the deformation {u} can be constructed from a linear combination of mode shapes $\varphi_{i,}$ where n is the number of modes:

$$\{u\} = \sum_{i=1}^{n} y_i\{\varphi_i\}$$

- From this, the equations of motion can be written in generalized coordinates as follows (with some additional substitutions not shown here for brevity):

$$\ddot{y}_i + 2\omega_i\xi_i\dot{y}_i + \omega_i^2 y_i = f_i$$

… Solution Methods

… MSUP Method

- From previous slide,

$$\ddot{y}_i + 2\omega_i\xi_i\dot{y}_i + \omega_i^2 y_i = f_i$$

Advantages of this approach:

- If a model consists of m DOF, instead of solving m equations, the modal equations only solve n DOF, where n represents the number of modes calculated in the modal analysis.
- Example: if 200 modes are extracted for a 1 million DOF model, instead of solving 1 million equations for the dynamic analysis, a user only solves for 200 DOF!
- Because of the reduced number of DOF, the solution is very fast.

… Solution Methods

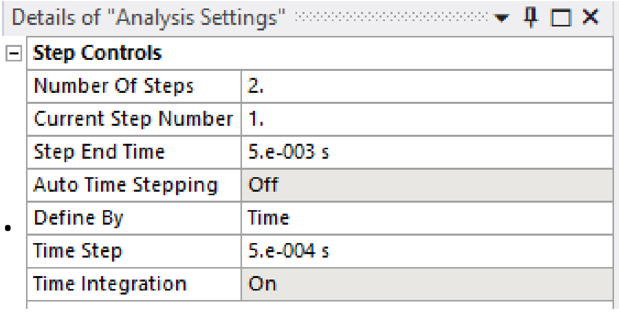
… MSUP Method

Points to remember:

- Linear combination means that only linear behavior is allowed.
- Time step is fixed (guidelines will be discussed later).
- The results are based on a truncated set of modes.
- A one-million DOF model has 1 million modes, but one typically solves for far fewer modes. In practice, only as many modes are required as are necessary to faithfully represent the dynamic response of the structure.

… Solution Methods

… MSUP Method



- Limitations of this approach:
  - Time step must remain constant (i.e., automatic time stepping is not allowed).
  - Time Integration must remain on.
  - Does not accept imposed (nonzero) displacements unless done so through base excitation (a support location applied in the modal analysis).
- All contact will behave as bonded or no separation in a modal analysis:
  - If a gap is present:
    - Nonlinear contacts will be free (no contact).
    - Bonded and no separation contact will depend on the pinball size.

Contact Type	Modal Analysis		
	Initially Touching	Inside Pinball Region	Outside Pinball Region
Bonded	Bonded	Bonded	Free
No Separation	No Separation	No Separation	Free
Rough	Bonded	Free	Free
Frictionless	No Separation	Free	Free
Frictional	Bonded	Free	Free

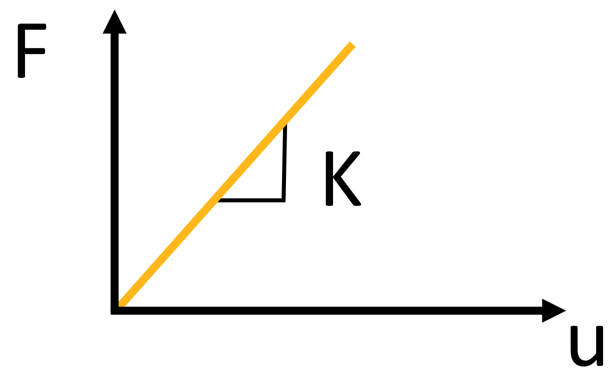
… Solution Methods (Summary)

Technique	
The Full Method	✓Relatively simple to set-up ✓Uses full matrices [K, M, and C]. ✓Allows **<u>all types of nonlinearities.</u>** ✓Calculates displacements and stresses in a single pass. ✓Accepts most load types (e.g., nodal forces, non-zero displacements, element loads, tabular boundary conditions, etc.) ✓Allows effective use of solid-model loads. x Typically, computationally more expensive than the mode-superposition method
The MSUP Method	✓Faster and less expensive than full method ✓Allows damping as a function of frequency x Time step must remain constant (i.e., automatic time stepping is not allowed). x Does not accept imposed (nonzero) displacements except as a base excitation. x Requires linear solution since it’s based upon modal analysis; no nonlinearities allowed.

C. Nonlinearities

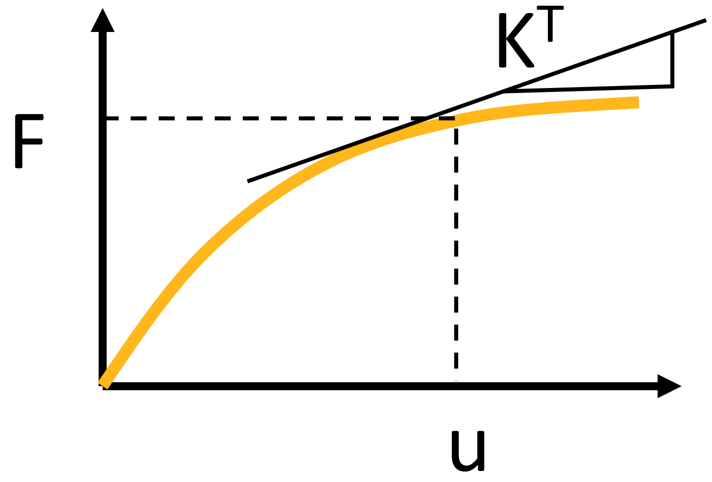
<u>Linear analysis:</u>

- Force and displacement are linearly related.
- Structural stiffness [K] is constant.
- Allows Mode-Superposition Method



<u>Nonlinear analysis:</u>

- Force and displacement are not linearly related.
- Stiffness [K^T] is not constant—it changes through the load path.
- <u>Newton-Raphson</u> method is used to solve nonlinear analysis.
- Requires Full Method

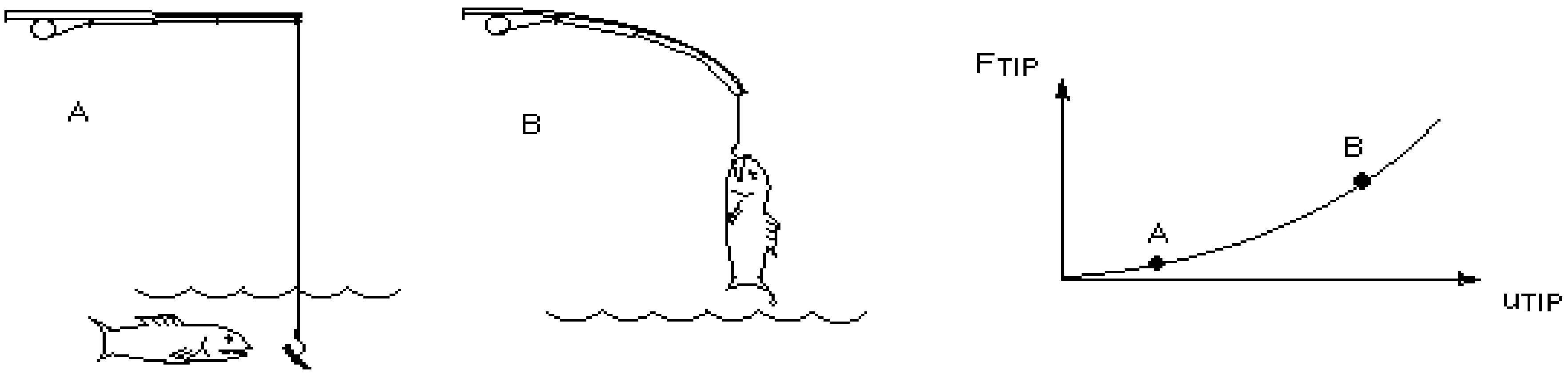


… Nonlinearities

There are three sources of structural nonlinearity:

1. <u>Geometric nonlinearities:</u>

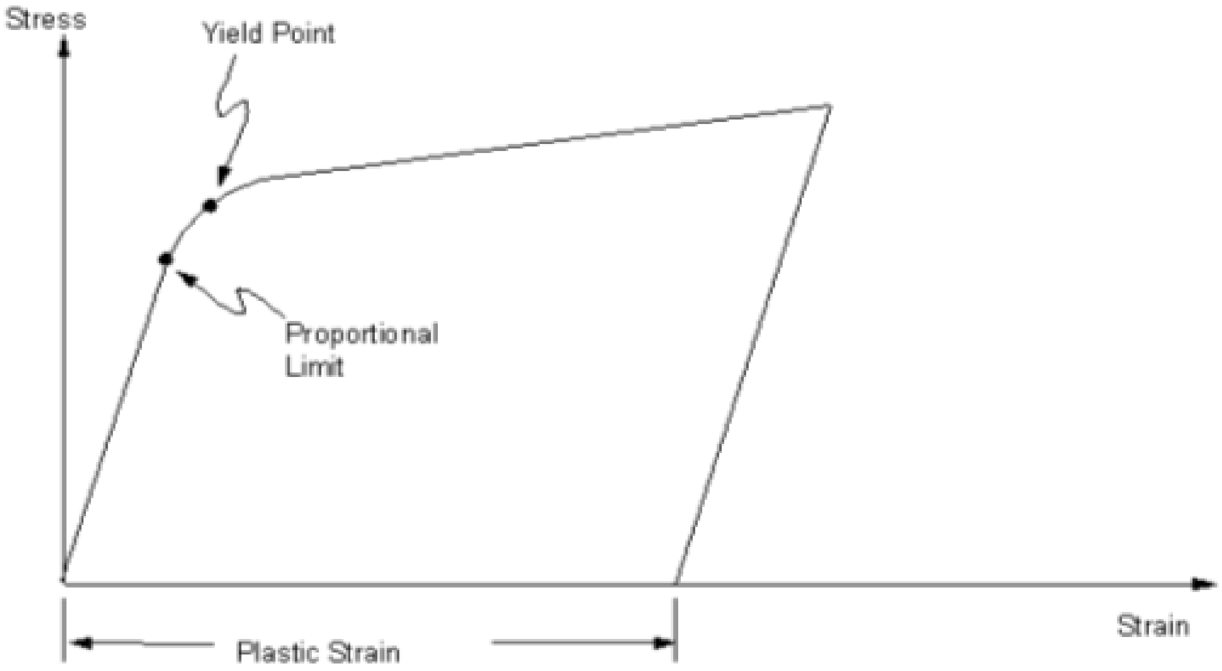
   Changing geometric configuration (large deformation, including both large strain and large rotation) causes structure to respond nonlinearly (classic fishing pole behavior).



… Nonlinearities

2. <u>Material nonlinearities:</u>

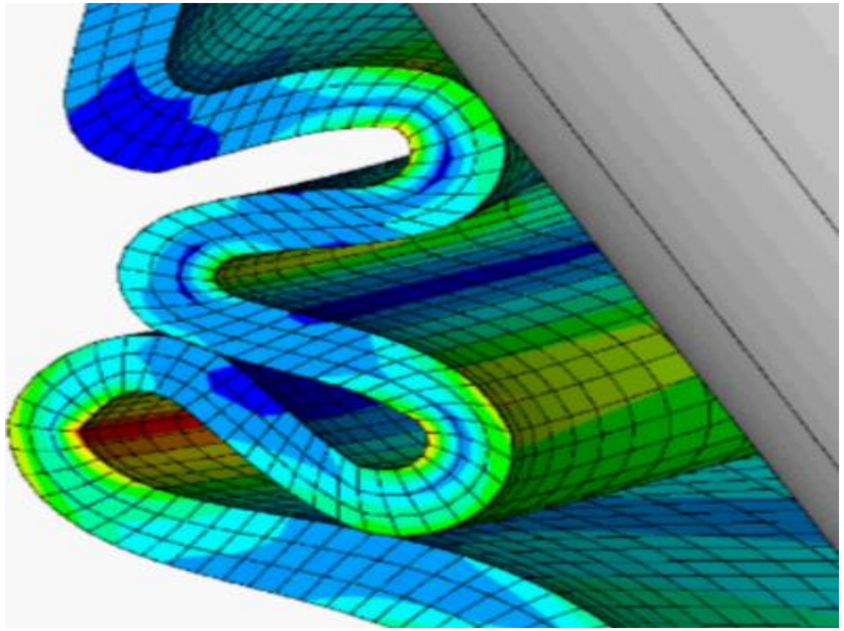
   Nonlinear stress-strain relationships (metal plasticity, creep, hyperelasticity, etc.)



… Nonlinearities

3. <u>Changing status nonlinearities:</u>

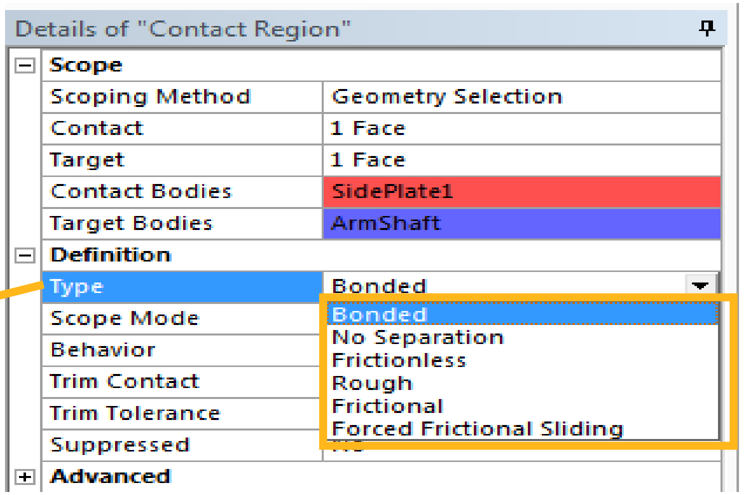
   Contact pair is either in or out of contact status, tension-only cable is either slack or taut, frictional contact, etc.



… Nonlinearities

- <u>All five contact types are allowed:</u>

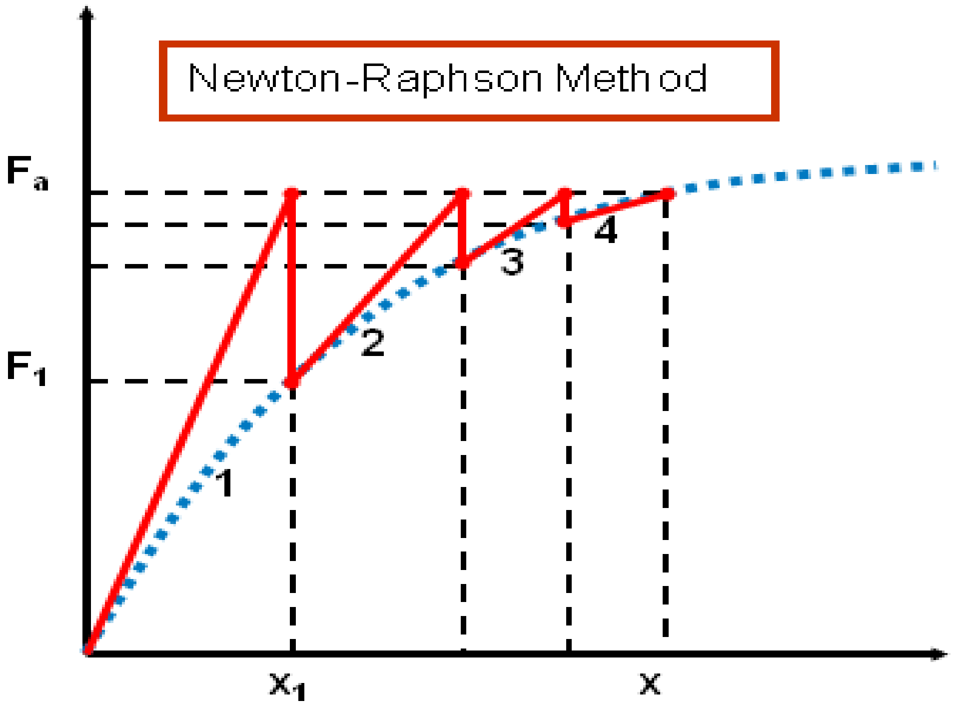
Contact Type	Iteration	Normal Behaviour (Separation)	Tangential Behaviour (Sliding)
Bonded	1	No Gaps	No Sliding
No Separation	1	No Gaps	Sliding Allowed
Rough	Multiple	Gaps Allowed	Sliding Allowed
Frictionless	Multiple	Gaps Allowed	No Sliding
Frictional	Multiple	Gaps Allowed	Sliding Allowed



- Bonded and No Separation contact are linear and require only one iteration (available for both Full and MSUP Methods).
- Frictionless, Rough and Frictional contact are nonlinear and require multiple iterations (available for Full Method only).

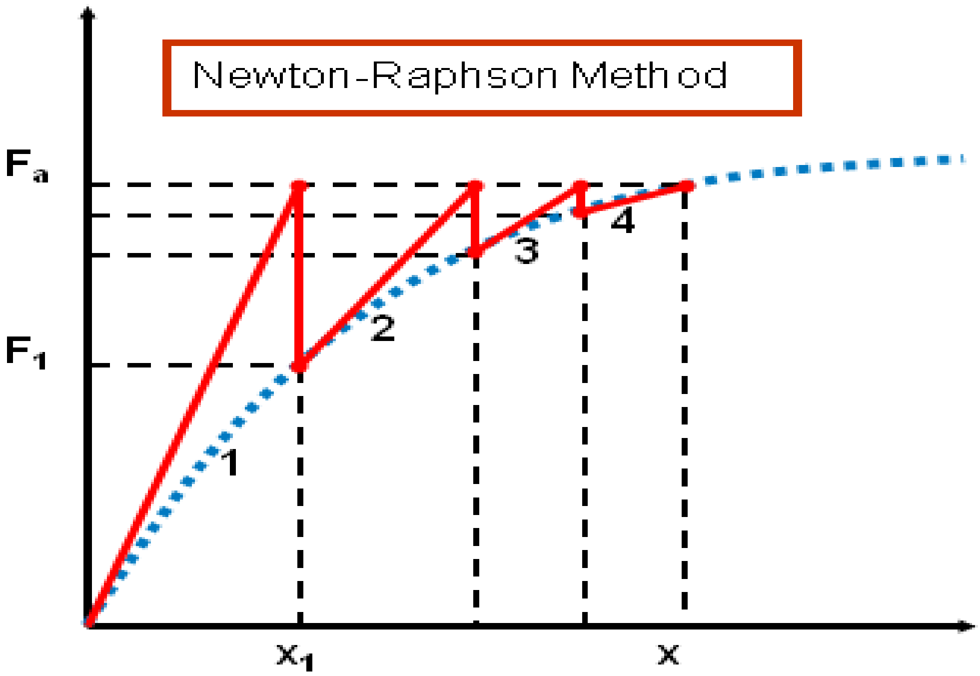
D. The Newton-Raphson Method

- In a nonlinear analysis, relationship between load and displacement cannot be determined with a single solution based on initial stiffness.
- The Newton-Raphson method uses a series of linear approximations with corrections:



… The Newton-Raphson Method

- Total external load F_a is applied in iteration 1 and displacements (x_1) are calculated.
- Using x_1, internal forces F_1 at iteration 1 are calculated.
- If $F_a \neq F_1$, the system is not in equilibrium.
- Differences between applied external and calculated internal forces (F_a - F_1) are the out-of-balance or residual forces.
- If residual forces are within an acceptable tolerance, the solution is converged.

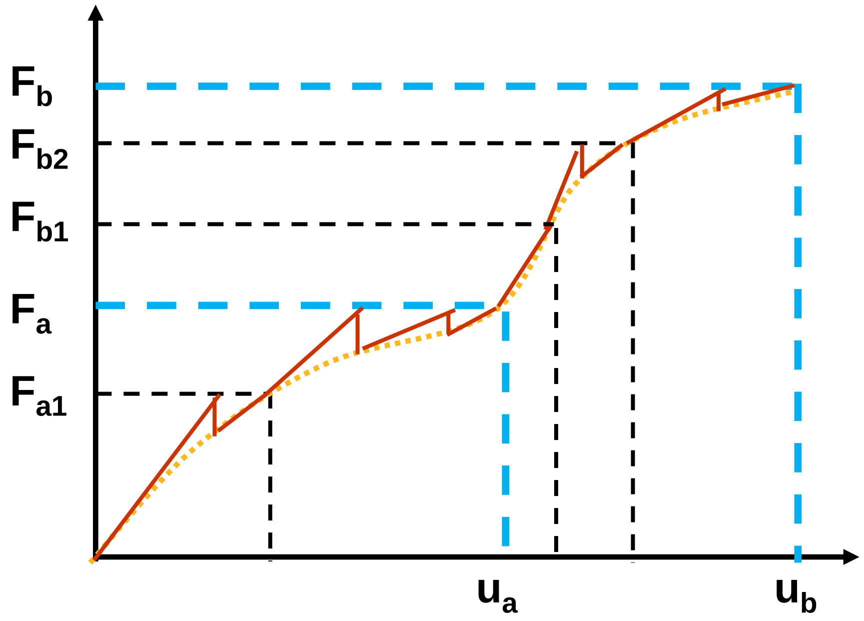


- If residual forces are outside an acceptable tolerance, the solution is not converged, so a new tangent stiffness matrix is assembled, and the process is repeated.
- In this example, the system achieves convergence after iteration 4.

… The Newton-Raphson Method

1. Load Steps and Substeps

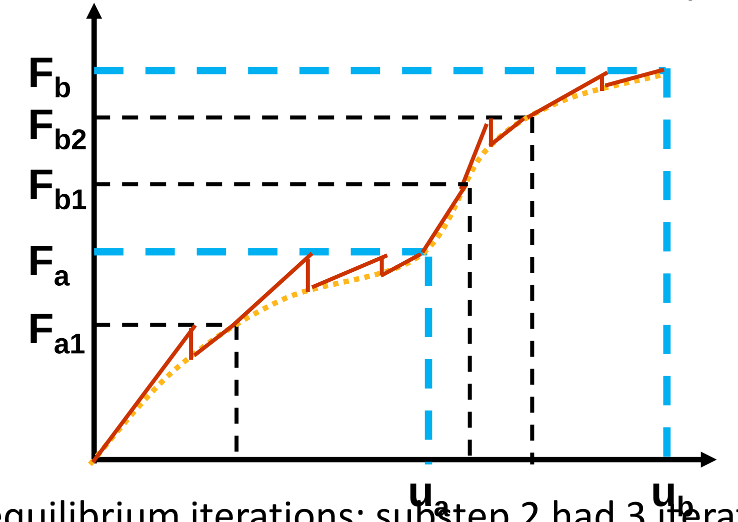
- Each solution point is defined in terms of a unique monotonically increasing time and a unique load step and substep combination.
- Load steps (known simply as Steps in Mechanical) are typically used to define changes in general loading.
  - F_a and F_b are load steps
- Substeps are typically used to increment the loading within load steps.
  - Because of the complex response, it desirable to incrementally apply the load.
  - For example, F_{a1} may be near 50% of the F_a load.
    - After F_{a1} is converged, full F_a load is applied.
    - F_a has 2 substeps while F_b has 3 substeps in this example.



… The Newton-Raphson Method

2. Equilibrium Iterations

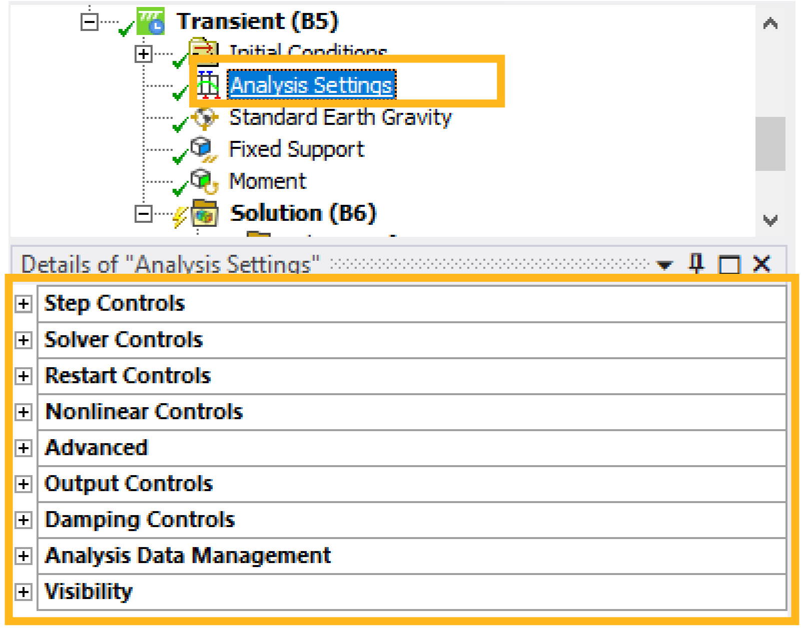
- In a nonlinear solution, equilibrium iterations are corrective solutions needed for convergence using the Newton-Raphson method.
  - Equilibrium iterations occur at the same time point (and same load step and substep).
  - In this example, the equilibrium iterations within each substep are represented as solid lines.



  - So, from above
    - Load Step F_a, substep 1 had two equilibrium iterations; substep 2 had 3 iterations.
    - Load Step F_b, substep 1 had 1 iteration, substep 2 had 2 iterations, substep 3 had 2 iterations.

E. Analysis Settings—Full Method

- In a full transient analysis, the control options are set under "*Analysis Settings*"
  1. Step Controls
  2. Solver Controls
  3. Restart Controls
  4. Nonlinear Controls
  5. Output Controls
  6. Damping Controls
  7. Analysis Data Management

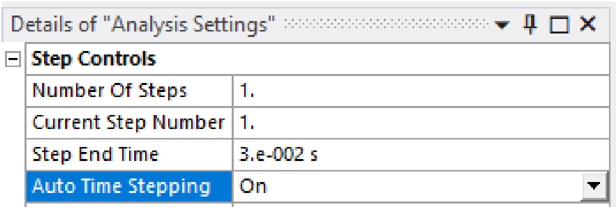


… E (1). Step Controls

- Step End Time: this is a required input, representing true chronological time consistent with the time duration of the applied loading; i.e. a 30ms half-sine shock pulse would have Step End Time = 0.030 s.
- Initial Time Step: also known as integration time step Δt, the time increment between successive time points.
  - One of the most important parameters in a transient structural analysis
  - Must be small enough to:
    - correctly describe the time-varying loads
    - capture the dynamic response
  - Running a preliminary modal analysis is suggested
  - Controls the accuracy and convergence behavior of nonlinear systems

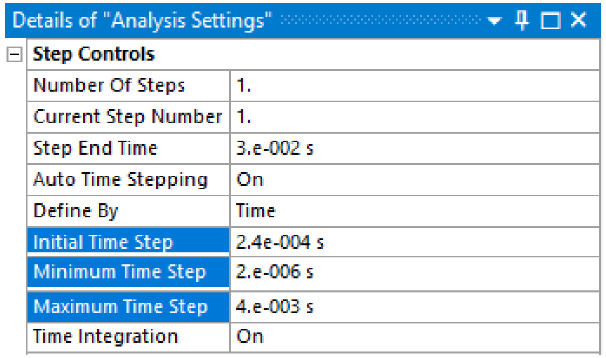
… E (1). Step Controls

- Automatic Time Stepping: adjusts the time step size (hence the load increment) throughout the solution.



  - Recommended, and on by default
  - Proper selection of the initial, minimum, and maximum time steps is important.
    - Smaller increments when convergence is difficult, larger increments when convergence is easy.
    - The maximum time step can be chosen based on accuracy concerns.
    - The minimum time step prevents Mechanical from solving indefinitely.

      (1/100 or 1/1000 of the initial time step)

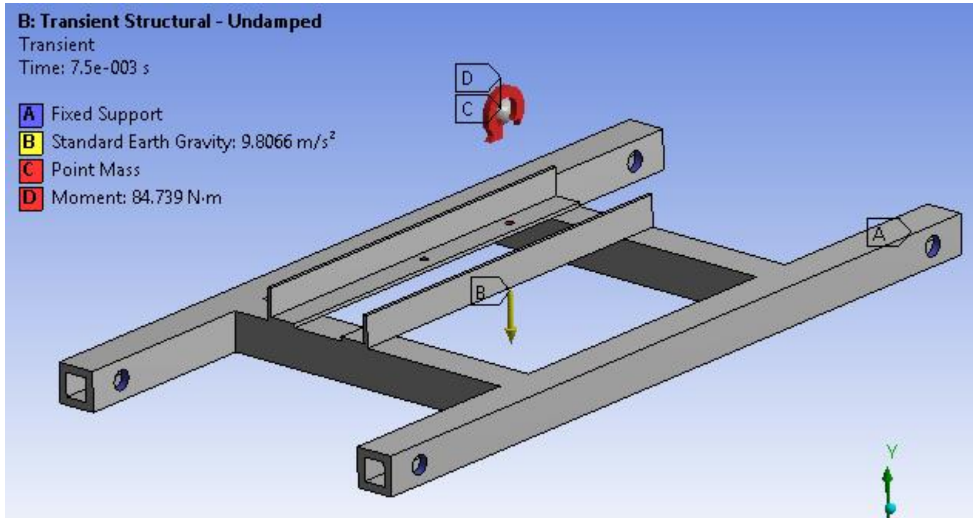


    - A general suggestion for selection of the initial time step is to use the following equation where $f_{response}$ is the frequency of the highest mode of interest (obtained from *Modal Analysis*).

$$\Delta t_{initial} = \frac{1}{20 f_{response}}$$

… E (1). Step Controls

- … E (1). Step Controls

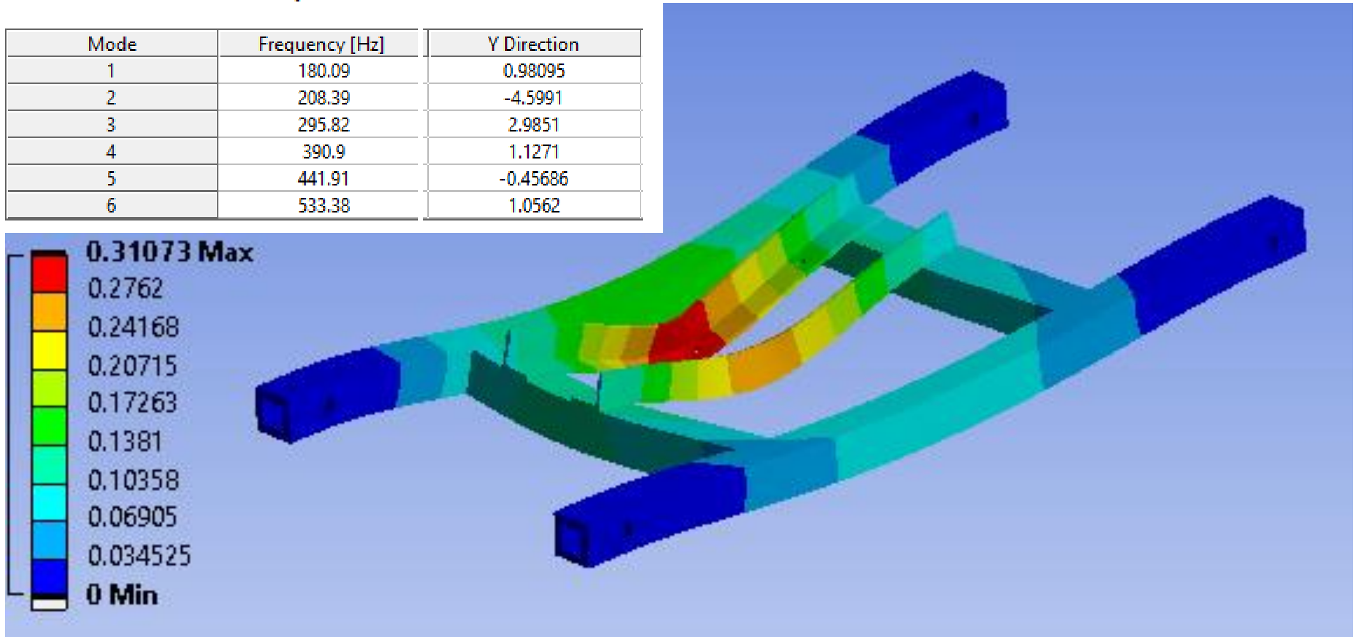


Transient Loading in Y direction

$$\Delta t_{initial} = \frac{1}{20 f_{response}} = \frac{1}{20(208.4)} = 2.4 \text{ x e-4 s}$$

Participation Factor

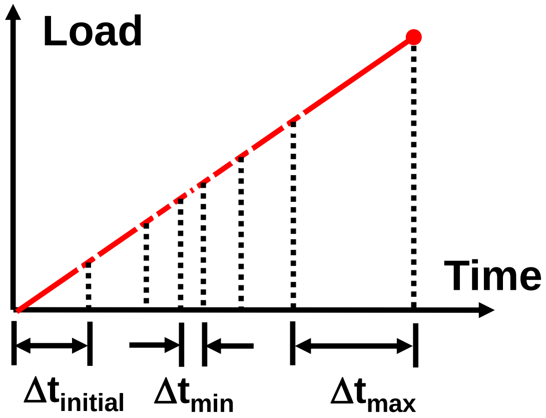
Mode	Frequency [Hz]	Y Direction
1	180.09	0.98095
2	208.39	-4.5991
3	295.82	2.9851
4	390.9	1.1271
5	441.91	-0.45686
6	533.38	1.0562



Modal Analysis shows Mode 2 (208 Hz) with largest Y Participation Factor

Details of "Analysis Settings"

Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	3.e-002 s
Auto Time Stepping	On
Define By	Time
Initial Time Step	2.4e-004 s
Minimum Time Step	2.e-006 s
Maximum Time Step	4.e-003 s
Time Integration	On



Ansys

… E (1). Step Controls

- Time Integration: indicates whether a solution step should include transient effects (e.g., structural inertia).
  - Transient effects are often turned "Off" in the 1st step to run a static equilibrium solution and, thus, to set up the Initial Conditions for a transient analysis.
    - *On:* Default for transient analyses.
    - *Off:* Do not include structural inertia or thermal capacitance in solving this step.
  - Note: with Time Integration "Off," Mechanical does not compute velocity results. Therefore, damping forces, which are derived from velocity, will equal zero.
  - Further discussion appears in Step Controls, Section F, Initial Conditions.

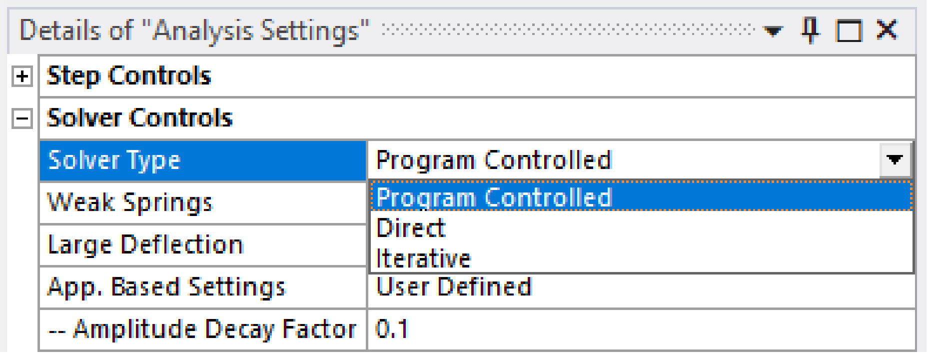
Details of "Analysis Settings"

Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	3.e-002 s
Auto Time Stepping	On
Define By	Time
Initial Time Step	2.4e-004 s
Minimum Time Step	2.e-006 s
Maximum Time Step	4.e-003 s
Time Integration	On

Ansys

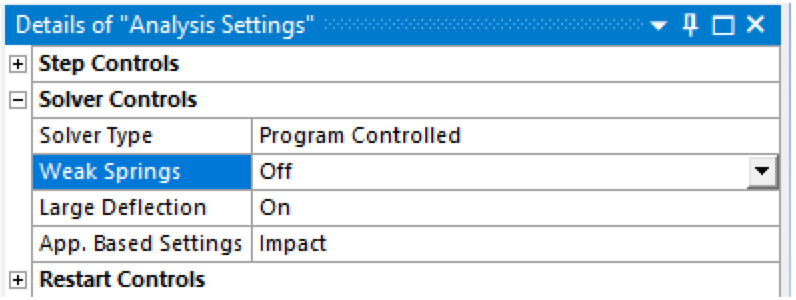
… E (2). Solver Controls

- <u>Solver Type:</u> a reference to the way Ansys builds the stiffness matrix for each Newton-Raphson equilibrium iteration:
  - Direct (Sparse):
    - more robust
    - recommended for challenging nonlinear models with non-continuum elements (shells and beams)
  - Iterative (PCG):
    - more efficient,
    - recommended for large bulk solid models dominated by linear elastic behavior.
    - The default “Program Controlled” automatically selects a solver based on the nature of the problem.

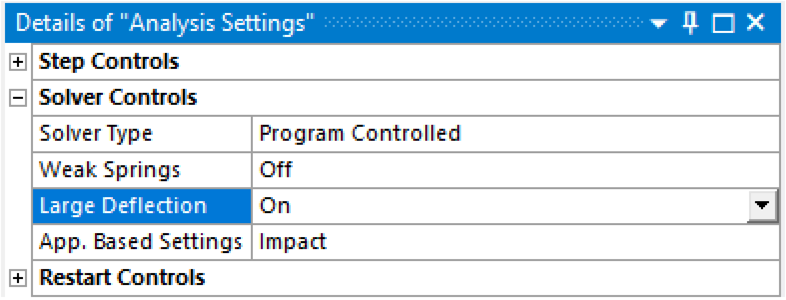


… E (2). Solver Controls

- Weak Springs: Weak Springs are typically included in static analysis to help prevent rigid body motion.
  - they are not needed for a transient analysis
  - turned off by default

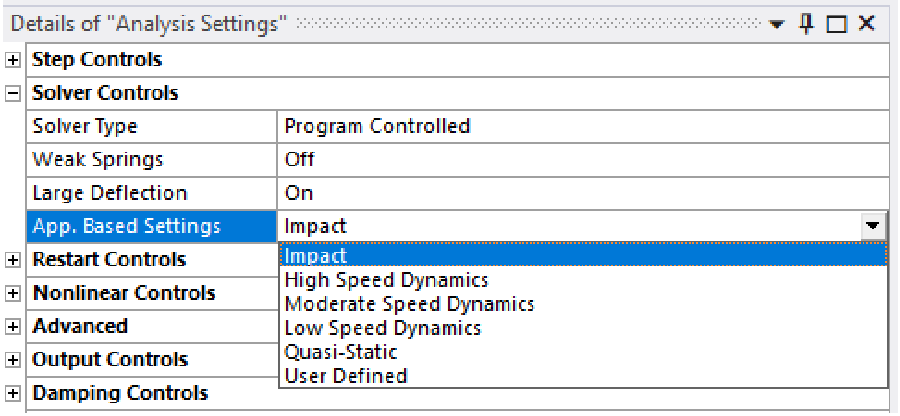


- Large Deflection: If set to “On” (default):
  - Stiffness matrix is adjusted over multiple iterations to account for changes such as:
    - large deflection
    - large rotation
    - large strain
  - Stress stiffening effect is included.
  - Spin softening effect is included.
  - The default condition requires a nonlinear solution. If you have a need to run a linear transient solution, set this to “Off.”



… E (2). Solver Controls

- Application Based Settings: Exposes the amplitude decay factor, γ, for HHT time integration as discussed in section A.
  - Impact: impact with no numerical dissipation, γ=0
  - High Speed: high speed with small dissipation, γ=.005
  - Moderate Speed: moderate speed with γ=.1
  - Low Speed: high numerical dissipation, γ=.414
  - Quasi-Static: high numerical dissipation useful for static analyses that fail to converge (buckling and instability problems)
  - User Defined: as discussed in section A.
  - For more information, see 5.6. Transient Dynamic Analysis Options



… E (3). Nonlinear Controls

- <u>Newton-Raphson Option:</u>
  - Controls when and how the stiffness matrix is updated during a nonlinear solution
  - Generally speaking, the Program Controlled default will allow Mechanical to choose the most robust form depending on physics of the problem.
    - <u>Full option</u>: stiffness matrix is updated at every iteration
    - <u>Modified option</u> allows less frequent update of stiffness, such as only during first or second iteration of each substep
    - Unsymmetric option: useful for problems containing frictional contact where $\mu > 0.2$

Details of "Analysis Settings"

+ **Step Controls**	
+ **Solver Controls**	
+ **Restart Controls**	
− **Nonlinear Controls**	
Newton-Raphson Option	Program Controlled Program Controlled Full Modified Unsymmetric
Force Convergence	
Moment Convergence	
Displacement Convergence	
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Program Controlled
--Method	Energy
--Energy Dissipation Ratio	1.e-004
--Activation For First Substep	No
--Stabilization Force Limit	0.2

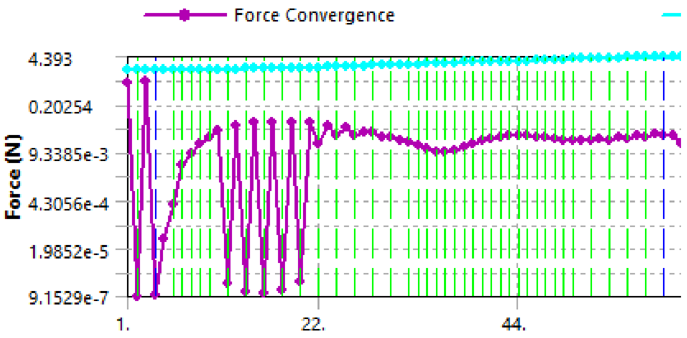
… E (3). Nonlinear Controls

- <u>Convergence Criteria:</u>
  - Tolerances on Convergence are calculated automatically.
  - Tolerances are used during the Newton-Raphson process to dictate when a model is Converged or “balanced”.
    - The default convergence criterion works very well for most engineering applications.
    - For special situations, users can override these defaults to *tighten* or *loosen* the convergence tolerance.
    - A tighter tolerance gives better accuracy but can make convergence more challenging.
    - Convergence monitored in Solution Information
    - Moment balance occurs when ROT DOF are present

Details of "Analysis Settings"

Step Controls	
Solver Controls	
Restart Controls	
Nonlinear Controls	
Newton-Raphson Option	Program Controlled
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Program Controlled
--Method	Energy
--Energy Dissipation Ratio	1.e-004
--Activation For First Substep	No
--Stabilization Force Limit	0.2

```
  DISP CONVERGENCE VALUE    = 0.8694E-05  CRITERION= 0.8152E-08
EQUIL ITER   1 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.1630E-06
  FORCE CONVERGENCE VALUE   = 0.1025E-08  CRITERION= 0.5926E-03 <<< CONVERGED
  DISP CONVERGENCE VALUE    = 0.4730E-13  CRITERION= 0.8152E-08 <<< CONVERGED
EQUIL ITER   2 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC =  0.1979E-14
>>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION   2
```




```
 FORCE CONVERGENCE VALUE   = 0.2080E-07  CRITERION= 0.3412E-02
 MOMENT CONVERGENCE VALUE  = 0.1484E-09  CRITERION= 0.3975E-01
EQUIL ITER   1 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.3449E-09
 FORCE CONVERGENCE VALUE   = 0.1294E-07  CRITERION= 0.3481E-02 <<< CONVERGED
 MOMENT CONVERGENCE VALUE  = 0.8429E-10  CRITERION= 0.4056E-01 <<< CONVERGED
```


Ansys

… E (3). Nonlinear Controls

- <u>Line Search:</u>
  - can be useful for enhancing convergence
  - When active, line search multiplies the displacement increment by a program-calculated scale factor between 0 and 1 when a stiffening response is detected.
  - You might consider setting Line Search on in the following cases:
    - If the structure is force-loaded
    - If the structure is a "flimsy" structure which exhibits increasing stiffness (such as a fishing pole)
    - If the convergence pattern is oscillatory

Details of "Analysis Settings"

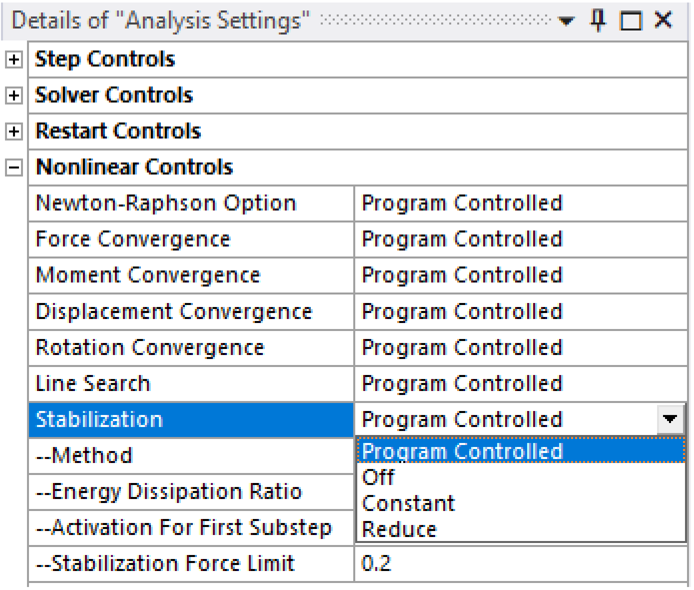
+ Step Controls	
+ Solver Controls	
+ Restart Controls	
− Nonlinear Controls	
Newton-Raphson Option	Program Controlled
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Program Controlled
--Method	Energy
--Energy Dissipation Ratio	1.e-004
--Activation For First Substep	No
--Stabilization Force Limit	0.2

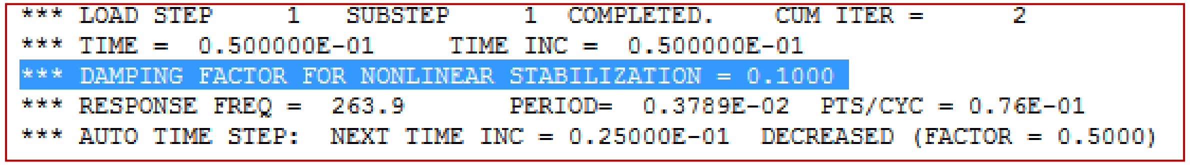
```
EQUIL ITER   1 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.3747E-01
 LINE SEARCH PARAMETER =  0.9955      SCALED MAX DOF INC = -0.3730E-01
 FORCE CONVERGENCE VALUE  =  1.155      CRITERION= 0.3481E-02
 MOMENT CONVERGENCE VALUE = 0.1830E-02  CRITERION= 0.4036E-01 <<< CONVERGED
EQUIL ITER   2 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.1694E-03
 LINE SEARCH PARAMETER =   1.000      SCALED MAX DOF INC = -0.1694E-03
 FORCE CONVERGENCE VALUE  = 0.3133E-03  CRITERION= 0.3551E-02 <<< CONVERGED
 MOMENT CONVERGENCE VALUE = 0.1070E-04  CRITERION= 0.4137E-01 <<< CONVERGED
>>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION   2
```


Ansys

… E (3). Nonlinear Controls

- <u>Nonlinear Stabilization:</u>
  - Can help achieve convergence
  - Adds artificial dampers to all of the DOFs in the system
  - Keys for controlling nonlinear stabilization:
    - Off - Deactivate stabilization (Default)
    - Constant - The energy dissipation ratio or damping factor remains constant during the load step.
    - Reduce - The energy dissipation ratio or damping factor is reduced linearly to zero at the end of the load step from the specified or calculated value.





… E (4). Output Controls

- Transient solutions typically generate much larger results files due to the smaller time steps used.
  - Default output controls include only stress, strain, and contact data output.
  - Nodal Forces: Turn “On” to report forces on unconstrained nodes. (Otherwise, nodal forces are available only at support locations)
  - Contact Miscellaneous: Turn “On” if contact forces are required. (Available through a Force Reaction Probe with “Location Method” set to “Contact Region”)
  - General Miscellaneous: Turn “On” if User-Defined Results are of interest.
  - Store Results At: One may choose to limit the frequency at which results are written, thus saving disk space. Options are:
    - Last Time Point → not recommended; all time history is lost
    - Equally Spaced Points → n number of points throughout history
    - Specified Recurrence Rate → Every nth time step

Output Controls	
Stress	Yes
Surface Stress	No
Back Stress	No
Strain	Yes
Contact Data	Yes
Nonlinear Data	No
Nodal Forces	No
Volume and Energy	No
Euler Angles	No
General Miscellaneous	No
Contact Miscellaneous	No
Store Results At	All Time Points
Result File Compression	Program Controlled
Damping Controls	

… E (5). Damping Controls

- The complete damping matrix [C] for nonlinear transient analysis is given by:

$$[C]=\alpha[M]+\beta[K]+\sum_{i=1}^{N_{ma}}\alpha_i^m[M_i]+\sum_{i=1}^{N_{ma}^{MD}}\sum_{k=1}^{N_{sa}}\alpha_p[M_k]_i$$

$$+\sum_{j=1}^{N_{mb}}\beta_j^m[K_j]+\sum_{j=1}^{N_{mb}^{MD}}\sum_{n=1}^{N_{sb}}\beta_q[K_n]_j+\sum_{k=1}^{N_e}[C_k]+\sum_{l=1}^{N_g}[G_l]+\frac{g}{2\pi\Omega}[K]+\sum_{j=1}^{N_m}\frac{m_j}{2\pi\overline{\overline{\Omega}}}[K_j]$$

- α : Global Mass-Matrix Multiplier (alpha damping, ALPHAD)
- β : Global k-Matrix Multiplier (beta damping, BETAD)
- α_i^m: Mass matrix multiplier for material *i* (alpha damping, MP,ALPD)
- β_j^m: Stiffness matrix multiplier for material *j* (beta damping, MP,BETD)
- C_k: Element damping (via the various Connection elements, COMBIN14, MPC184, etc.)
- g : constant structural damping coefficient (DMPSTR)
- m_j: constant structural damping coefficient for material *j* (MP,DMPS)

Other terms not highlighted include mass and stiffness damping based upon elements with mass and stiffness proportional damping and defined sections (not native to Mechanical) and gyroscopic damping (Rotor dynamics analysis)

… E (5). Damping Controls

- Full Transient Analysis accepts Rayleigh and Element damping.

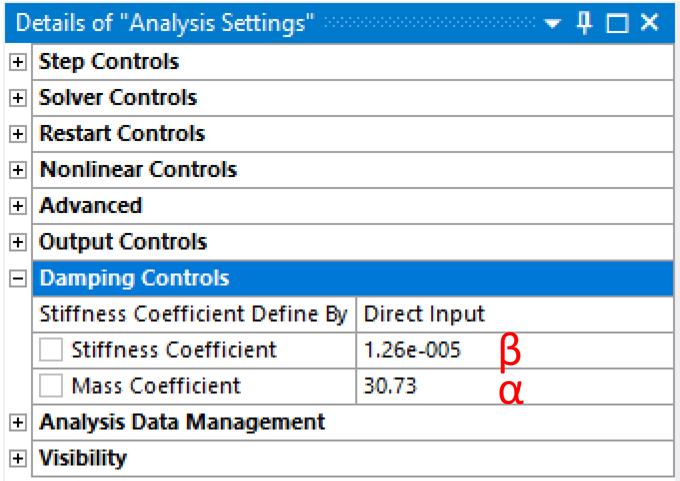
1. Rayleigh Damping (Defined Globally):
   - Alpha damping and Beta damping are defined by Rayleigh damping constants α and β. The damping matrix [C] is calculated by using these constants to multiply the mass matrix [M] and stiffness matrix [K]:

$$[C] = \alpha[M] + \beta[K]$$

Equivalent damping

$$\xi = \frac{\alpha}{2\omega} + \frac{\beta\omega}{2}$$

   - Values of α and β are not usually known directly but are calculated from damping ratio ξ as discussed in Module 02.



… E (5). Damping Controls

- Full Transient Analysis accepts Rayleigh and Element damping.

2. Rayleigh Damping (Defined per material):
   - Mass-Matrix Damping Multiplier, K-Matrix Damping Multiplier

$$[C] = \sum_{i=1}^{N_{ma}} \alpha_i^m [M_i] + \sum_{j=1}^{N_{mb}} (\beta_j^m)[K_j]$$

Equivalent damping

$$\xi_i = \frac{\alpha}{2\omega_i} + \frac{\beta\omega_i}{2}$$

Properties of Outline Row 3: Structural Steel

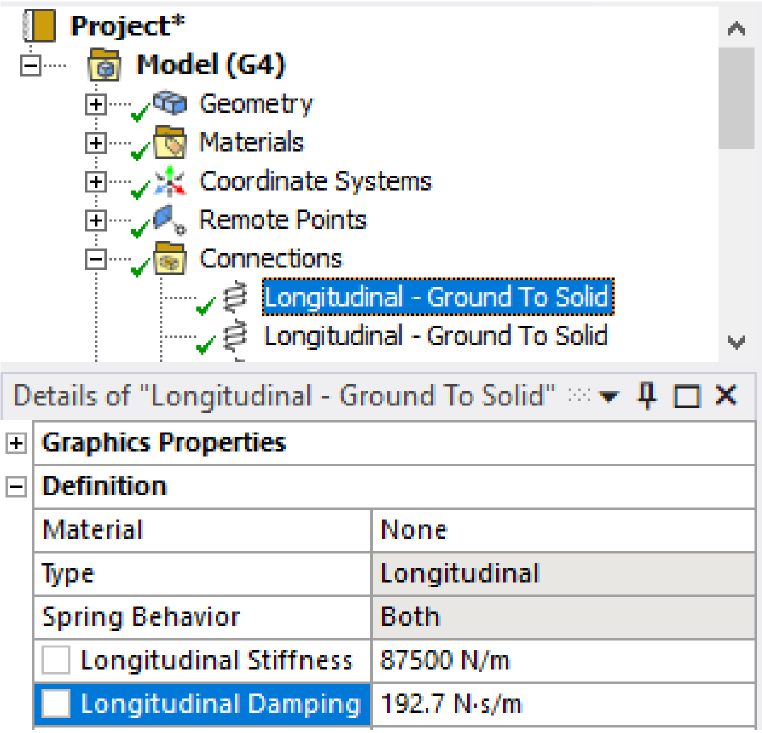
	A	B
1	Property	Value
2	Material Field Variables	Table
3	Density	7850
4	Isotropic Secant Coefficient of Thermal Expansion	
6	Material Dependent Damping	
7	Damping Ratio	0.02
8	Constant Structural Damping Coefficient	= 0.04
9	Damping Factor (α)	
10	Mass-Matrix Damping Multiplier	30.73 α^m
11	Damping Factor (β)	
12	k-Matrix Damping Multiplier	1.26E-05 β^m

… E (5). Damping Controls

- Full Transient Analysis accepts Rayleigh and Element damping.

3. Element Damping:
   - Element damping involves element types having viscous damping characteristics; Body-Body and/or Body-Ground Spring Connections, Joints, Bearings and Bushings are some examples.

$$\sum_{k=1}^{N_e}[C_k]$$

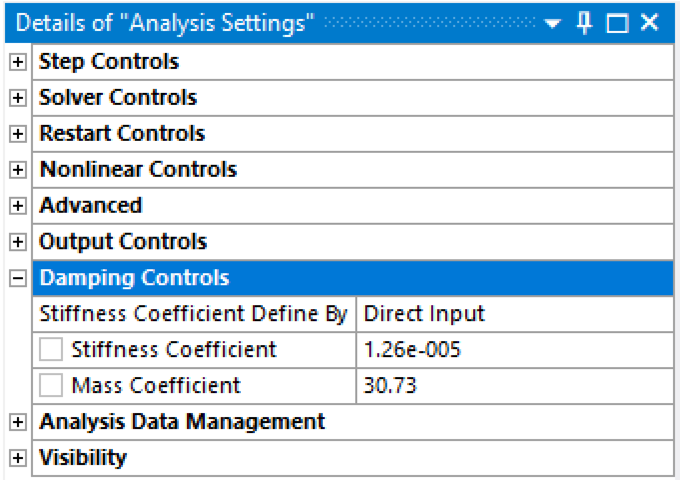


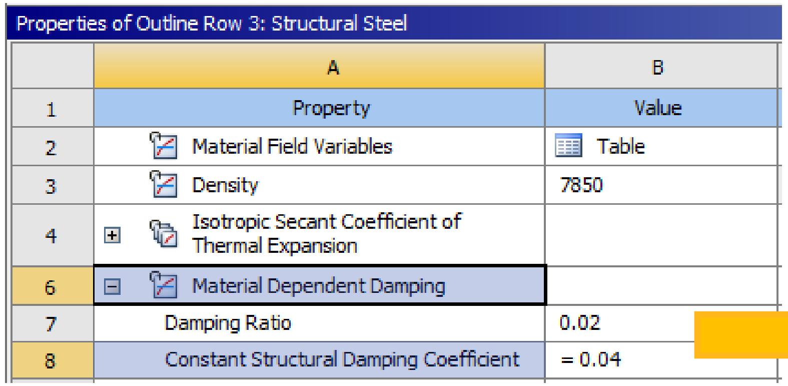
… E (5). Damping Controls

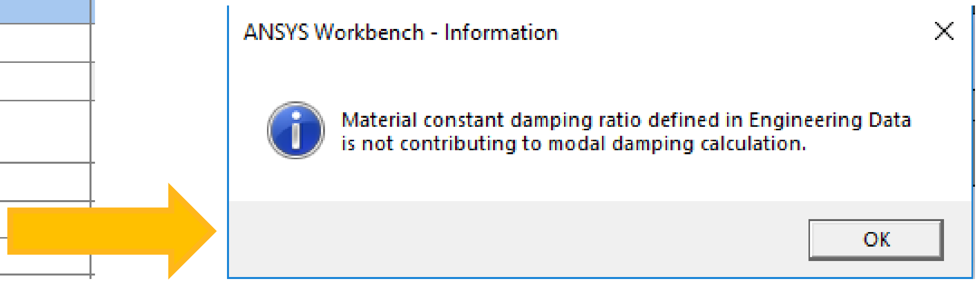
- Full Transient Analysis also supports hysteretic damping in the form of the Constant Structural Damping Coefficient, in both the global form (g) and per material (m_j).

$$[c] = \frac{g}{2\pi\Omega}[K] + \sum_{j=1}^{N_m} \frac{m_j}{2\pi\Omega}[K_j]$$

- Per Module 02, however, the option to define either term does not appear natively:
  - in Mechanical, Constant Structural Damping Coefficient is not shown within Damping Controls.
  - In Engineering Data, the option appears within the Material Dependent Damping property, however a solution produces the following warning:

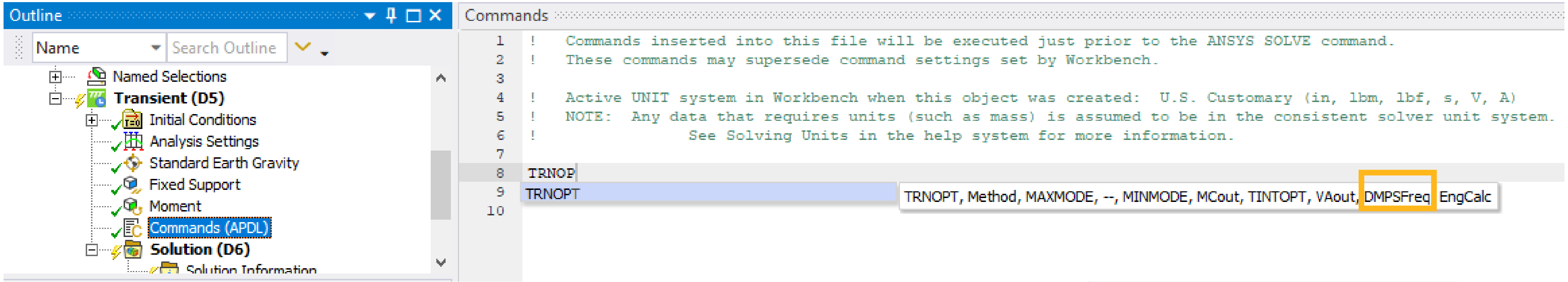


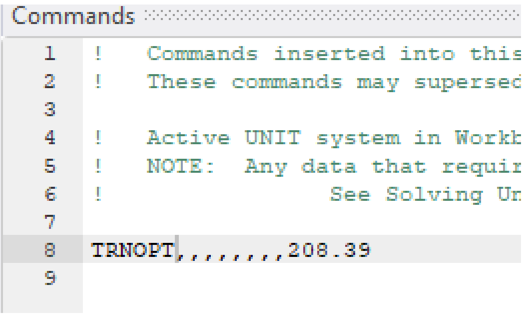




… E (5). Damping Controls

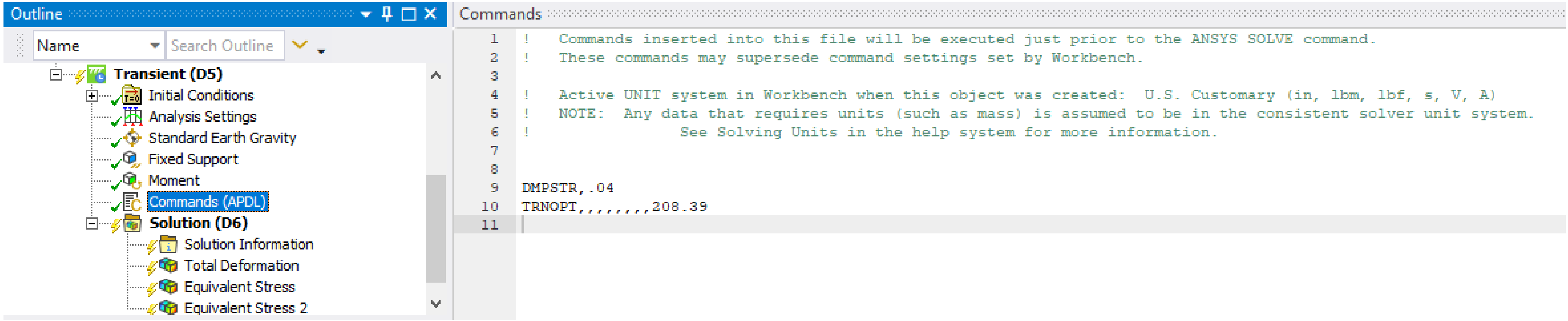
- To define Constant Structural Damping Coefficient (m_j) on a material basis:
  - Define it first within Engineering Data, per previous slide,
  - Use a Command Object in Mechanical along with the TRNOPT command
    - DMPSFreq represents the frequency used for calculation of equivalent viscous damping.





… E (5). Damping Controls

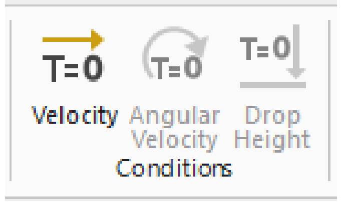
- To define Constant Structural Damping Coefficient (g) globally:
  - Use a Command Object in Mechanical along with the DMPSTR and TRNOPT commands



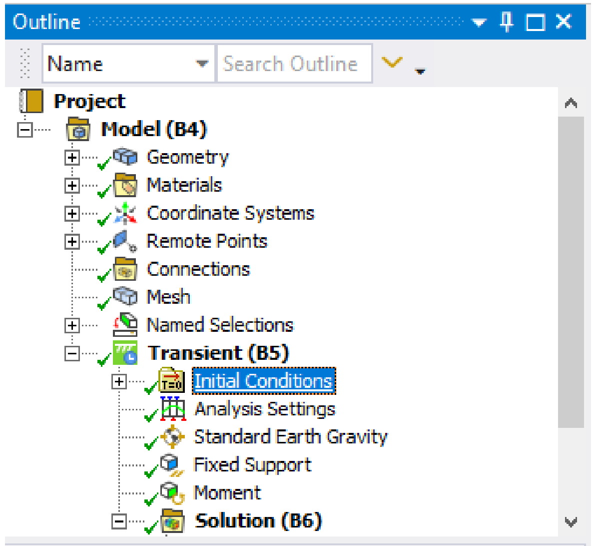
  - Note: Defining Constant Structural Damping coefficients either globally or per material will have an effect very similar to that of stiffness (beta) damping; because it is based upon a given frequency, it will result in less damping at lower frequencies and more damping at higher frequencies.

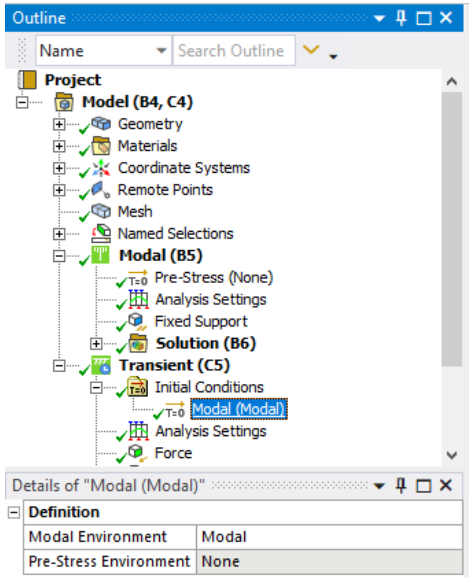
F. Initial Conditions

- Initial conditions are the conditions at Time = 0
  - The default initial condition is that the structure is *"at rest"*, that is, both initial displacement and initial velocity are zero.
  - For transient drop test analysis, initial velocity can be scoped to one or more parts of the structure.



  - The remaining parts of the structure which are not part of the scoping will retain the *"at rest"* initial condition.
  - For MSUP transient analyses, initial conditions are imported from the Modal analysis, and no additional conditions can be added.





… Initial Conditions

- Some initial conditions require definition over two load steps:

<u>1. Initial Displacement = 0, Initial Velocity ≠ 0</u>

First step displacement = 0.005 mm

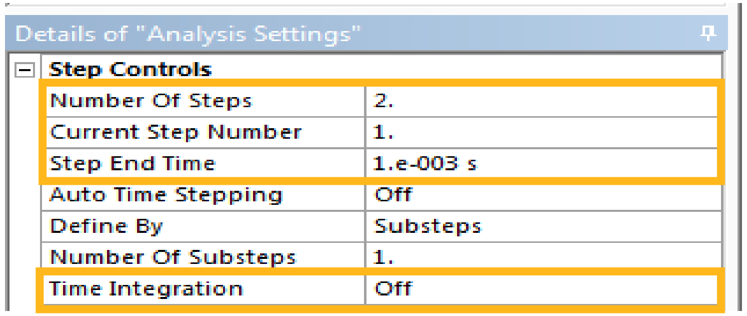
First step end time = 0.001 sec.

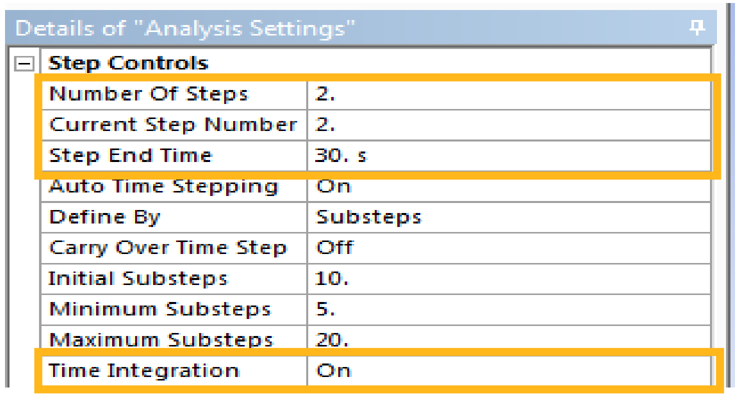
Use an arbitrary small (near 0) displacement along with arbitrary end time to achieve desired velocity.

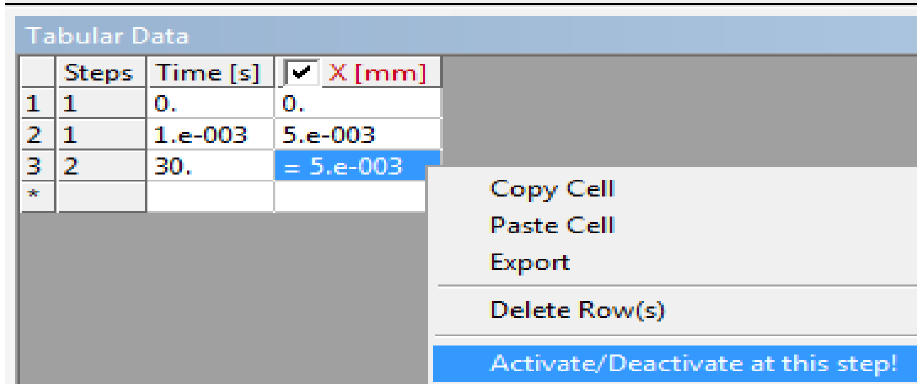
Initial velocity of (0.005/0.001) = 5 mm/sec.

– Deactivate the specified displacement load in the second step so that the part is free to move with the specified initial velocity.

Make sure that time integration effects are turned <u>off</u> for the first step and <u>on</u> for the second step.







Ansys

… Initial Conditions

2. Initial Displacement ≠ 0, Initial Velocity ≠ 0

Initial velocity of 0.5 mm/sec.

Initial (actual) displacement of 0.1 mm

The first step end time = (0.1/0.5) = 0.2 sec.

Details of "Analysis Settings"

Step Controls	
Number Of Steps	2.
Current Step Number	1.
Step End Time	0.2 s
Auto Time Stepping	Off
Define By	Substeps
Number Of Substeps	1.
Time Integration	Off

Tabular Data

	Steps	Time [s]	X [mm]
1	1	0.	0.
2	1	0.2	0.1
3	2	5.	= 0.1

3. Initial Displacement ≠ 0, Initial Velocity = 0

Initial displacement of 0.1 mm

The first step end time = 0.001 sec.

Note the step application of the displacement and a minimum of at least 2 substeps

Details of "Analysis Settings"

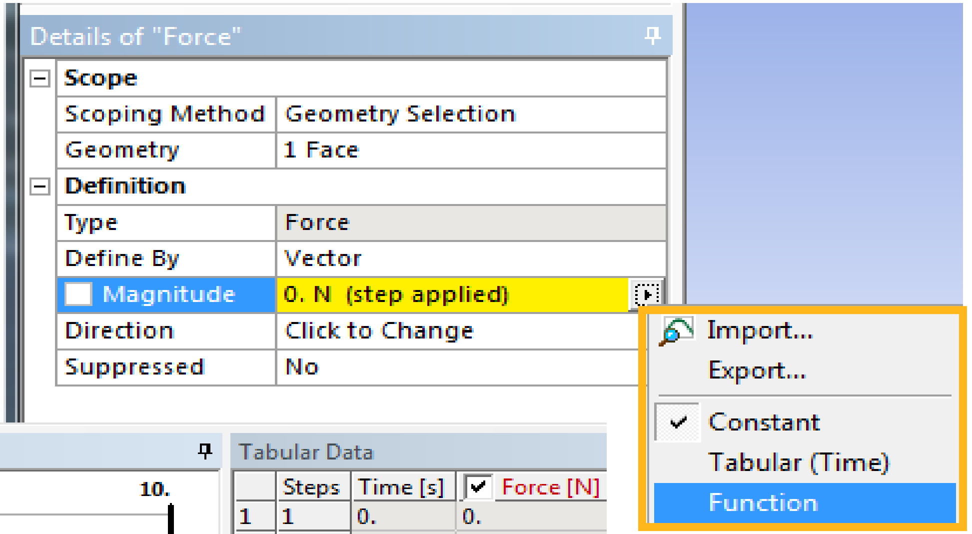
Restart Analysis	
Step Controls	
Number Of Steps	2.
Current Step Number	1.
Step End Time	1.e-003 s
Auto Time Stepping	Off
Define By	Substeps
Number Of Substeps	3.
Time Integration	Off

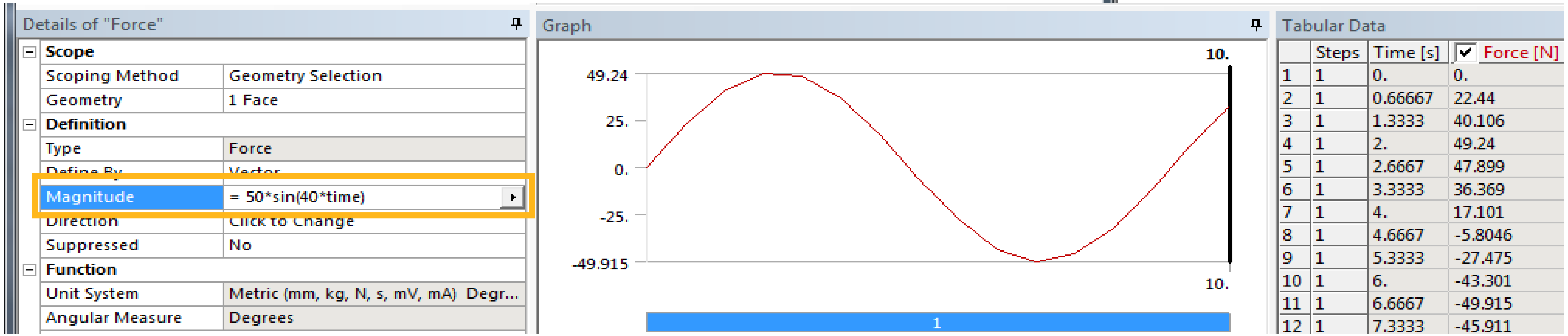
Tabular Data

	Steps	Time [s]	X [mm]
1	1	0.	0.1
2	1	1.e-003	0.1
3	2	5.	= 0.1

G. Loads and Supports

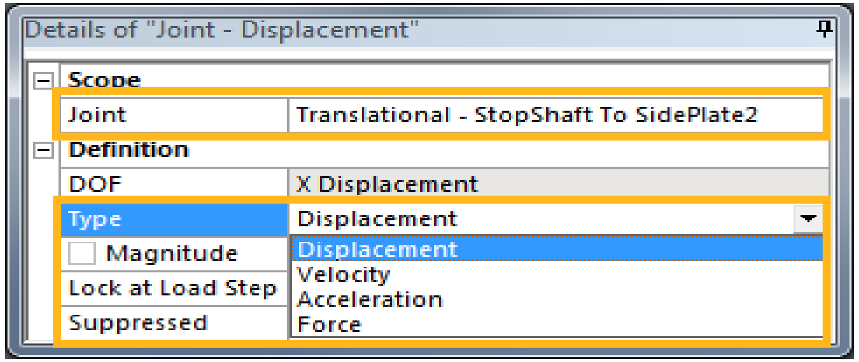
- For the Full Method, all inertial and structural loads and all structural supports are allowed. Some limitations exist for the MSUP Method (more on this shortly).
- Joint Loads may be used to kinematically drive joints.
- Magnitude can be:
  - Constant,
  - Tabular (time-varying)
  - Function

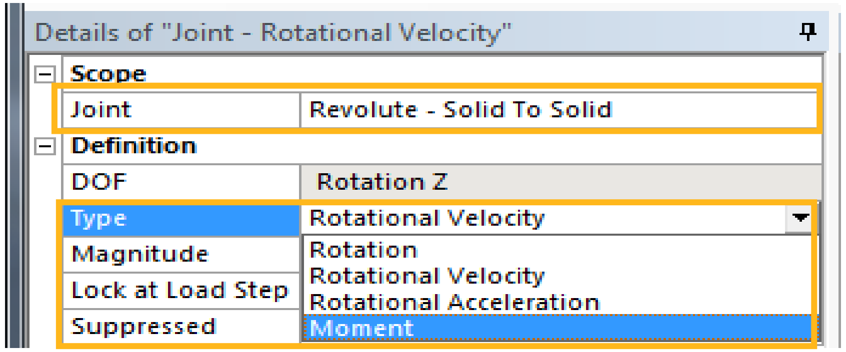




… Loads and Supports

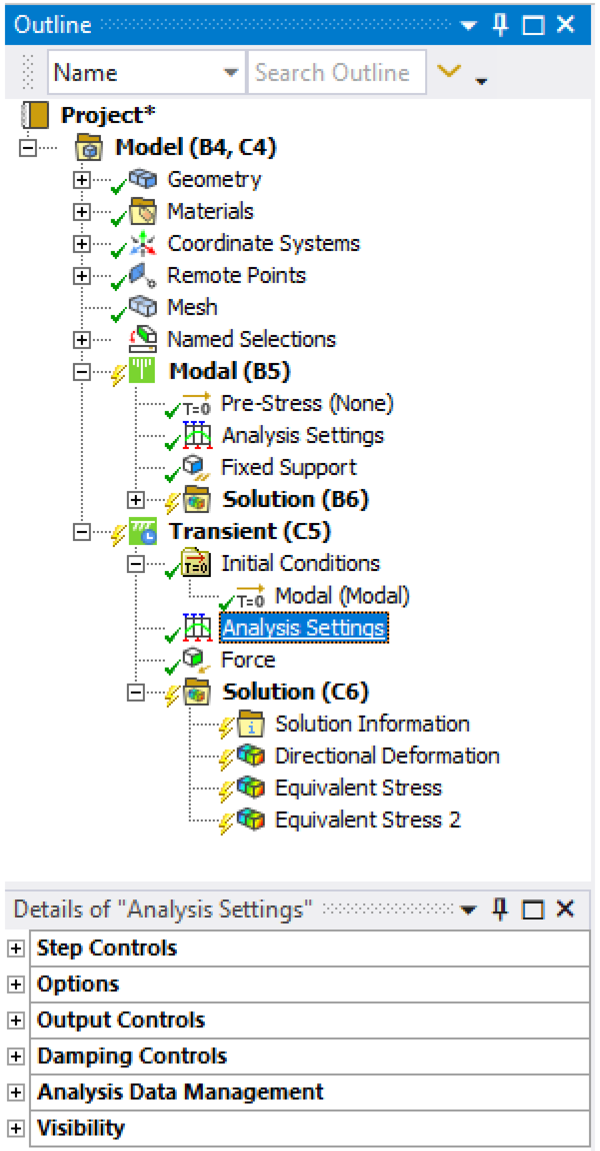
- Joints define the allowed motion (kinematic constraint) on surface(s).
- Various types of joints can be defined for flexible or rigid bodies
- Absolute DOF are specified.
- A Joint Load object is used to apply a kinematic driving condition to a single DOF on a Joint object.
- For translation DOF
  - displacement, velocity, acceleration, or force is applied.
- For rotation DOF
  - rotation, angular velocity, angular acceleration, or moment is applied.





H. Analysis Settings – MSUP Method

- Under "*Analysis Settings*" in MSUP transient dynamics, there are several control options that need to be considered:
  1. Step Controls
  2. Options
  3. Damping Controls



Ansys

… H(1). Step Controls

- Time step must remain constant (i.e., automatic time stepping is not allowed).
  - Auto time stepping is always turned off
- Substeps or time step value is defined and is applicable to ALL the load steps
- The time integration is always turned on

Details of "Analysis Settings"

Step Controls	
Number Of Steps	2.
Current Step Number	2.
Step End Time	0.2 s
Auto Time Stepping	Off
Define By	Time
Time Step	5.e-004 s
Time Integration	On

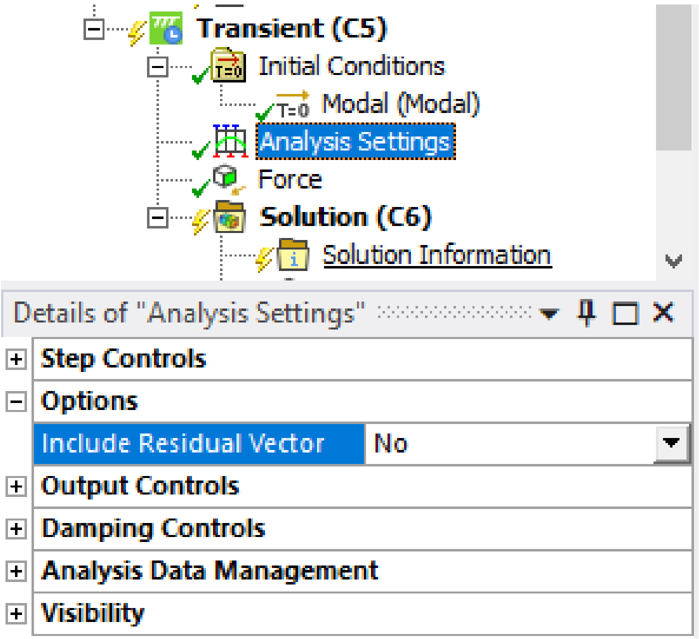
Worksheet

Analysis Settings

Properties	Step 1	Step 2
Step Controls		
Step End Time	5.e-003	0.2
Auto Time Stepping	Off	Off
Define By	Time	Time
Time Step	5.e-004	5.e-004
Time Integration	On	On
Output Controls		
Stress	Yes	Yes
Surface Stress	No	No
Back Stress	No	No
Strain	Yes	Yes
Contact Data	Yes	Yes
Nodal Forces	No	No
General Miscellaneous	No	No
Store Results At	All Time Points	All Time Points

… H(2). Options

- Include Residual Vector :
  - Allows inclusion of residual vectors for MSUP Transient Structural analyses.
  - In MSUP analysis, the dynamic response will be approximate when the applied loading excites the higher frequency modes of a structure.
  - The residual vector method employs additional modal transformation vectors in addition to the eigenvectors in the modal transformation .
  - This feature accounts for high-frequency dynamic responses with fewer eigenmodes.
  - The default setting is “No”



… H(3). Damping Controls

- The damping matrix [C] in MSUP Transient is not calculated explicitly, but instead damping is defined directly in terms of a damping ratio ξ^d for mode i :

$$\xi_i^d = \xi + \xi_i^m + \frac{\alpha}{2\omega_i} + \frac{\beta\omega_i}{2}$$

ξ : constant modal damping ratio (DMPRAT)

ξ_i^m: modal damping ratio for mode shape i (MP,DMPR during modal analysis)

α : Global Mass matrix multiplier (alpha damping, ALPHAD)

β : Global k-Matrix Multiplier (beta damping, BETAD)

… H(3). Damping Controls

- Values for ξ, α, and β can be entered on a Global basis via the Damping Controls section of Analysis Settings (below left):
- The value for ξ^m is entered on a material basis within Engineering Data as part of the undamped Modal Analysis (below right):
  - Set the “Eqv. Damping Ratio from Modal” = Yes in order to see the effects of damping in the Transient

Details of "Analysis Settings"

Step Controls		
Options		
Output Controls		
Damping Controls		
Eqv. Damping Ratio From Modal	No	
Damping Ratio	2.e-002	ξ
Stiffness Coefficient Define By	Direct Input	
Stiffness Coefficient	1.26e-005	β
Mass Coefficient	30.73	α
Numerical Damping	Program Controlled	
Numerical Damping Value	0.005	
Analysis Data Management		
Visibility		

Properties of Outline Row 3: Structural Steel

	A	B	
1	Property	Value	
2	Material Field Variables	Table	
3	Density	7850	
4	Isotropic Secant Coefficient of Thermal Expansion		
6	Material Dependent Damping		
7	Damping Ratio	0.01	ξ^m
8	Constant Structural Damping Coefficient	= 0.02	

Details of "Analysis Settings"

Step Controls	
Options	
Output Controls	
Damping Controls	
Eqv. Damping Ratio From Modal	Yes
Damping Ratio	0.
Stiffness Coefficient Define By	Direct Input
Stiffness Coefficient	0.
Mass Coefficient	0.
Numerical Damping	Program Controlled
Numerical Damping Value	0.005

… H(3). Damping Controls

- From Module 02, MSUP Transient also supports a special case of Element Damping provided that the upstream Modal analysis is conducted using the Reduced Damped solver.
  - In this case, the full damping matrix must be retained in the MSUP Transient analysis.

$$[C_m]=[\Phi^T][C][\Phi]+\frac{g}{2\pi\Omega}[\Phi^T][K][\Phi]+\sum_{j=1}^{N_m}\frac{m_j}{2\pi\Omega}[\Phi^T][K_j][\Phi]+[\Xi]$$

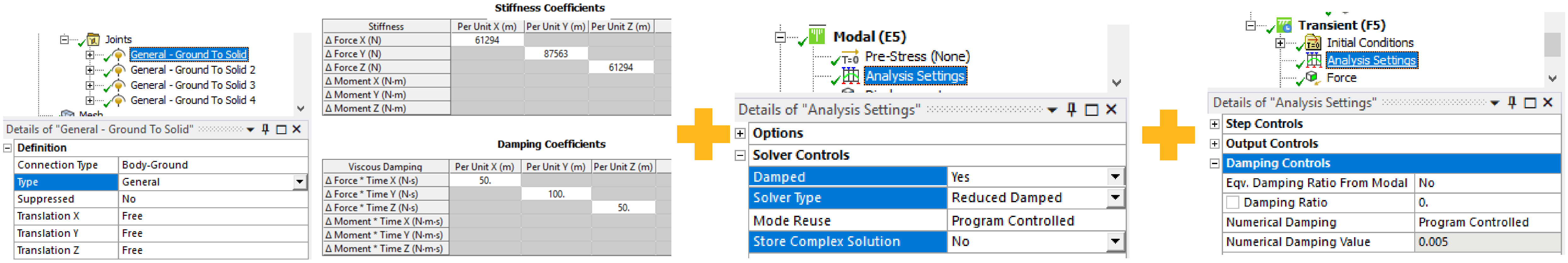
- $[C_m]$ is the damping matrix in the modal basis
- $[C]$ is identical to that used within the Full Transient method (slide 41) and which contains the term

$$\sum_{k=1}^{N_e}[C_k]$$

- $[C_k]$: Element damping (via the various Connection elements, COMBIN14, MPC184, etc.)

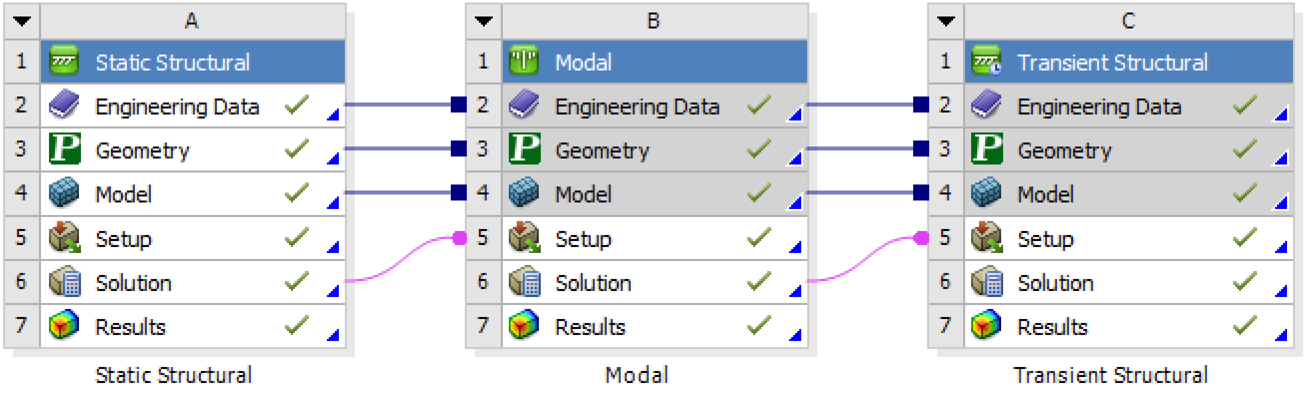
… H(3). Damping Controls

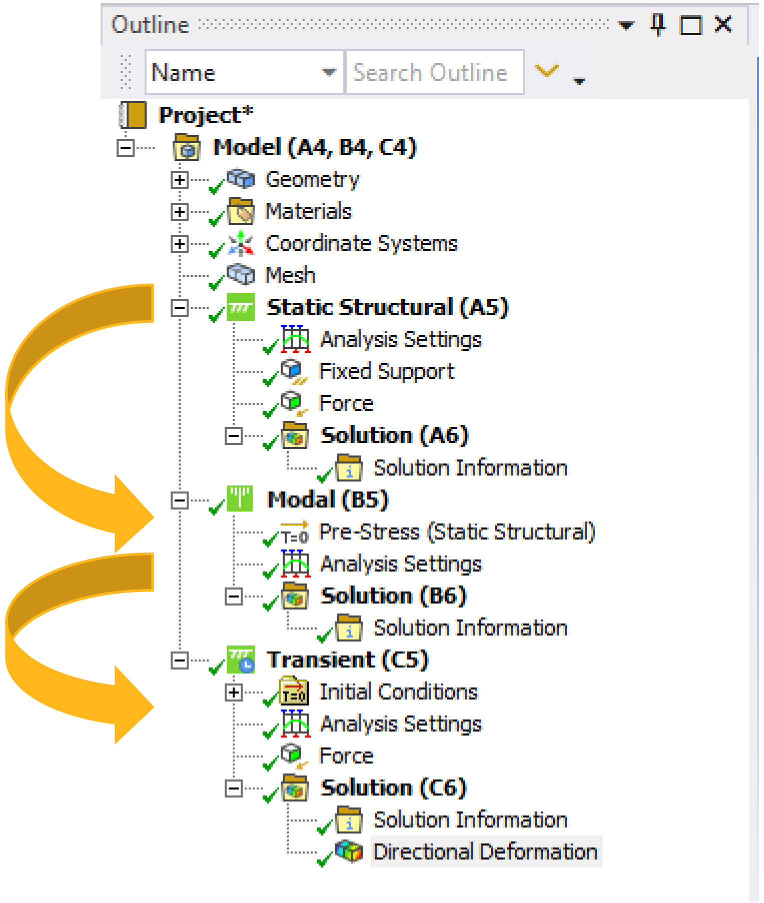
- Below are the requirements to include Element Damping in an MSUP Transient analysis:
  - Modal analysis with Connections that include damping (Body/Ground Springs, General Joints, etc.)
  - Reduced Damped solver in Modal
    - Store Complex Solution = No
  - Damping need not be defined in the MSUP transient, although it may be if desired.
    - “Eqv. Damping From Modal” is not needed in this scenario.



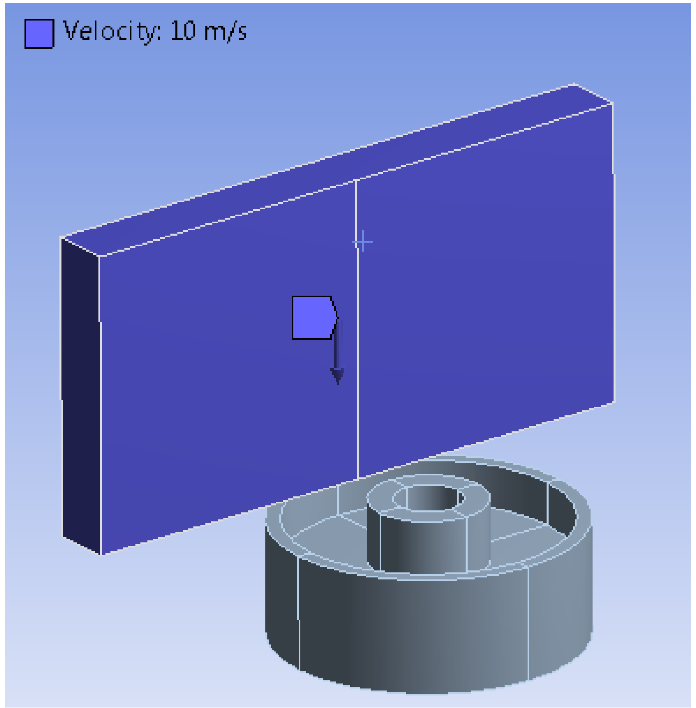
I. Pre-Stressed Mode-Superposition Method

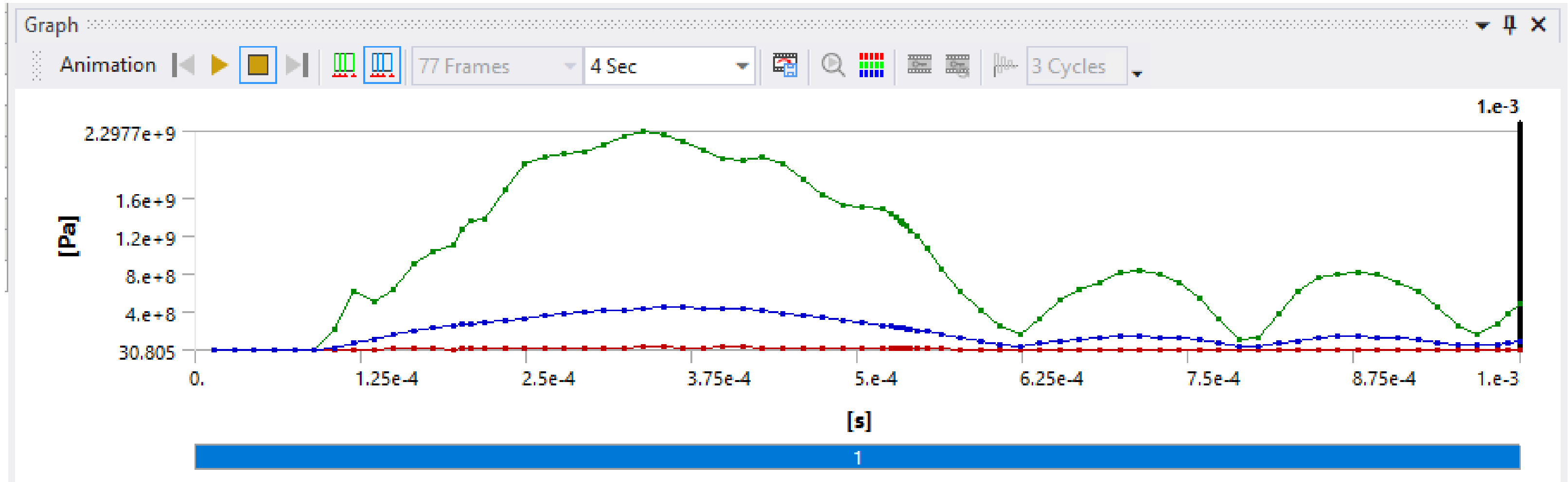
- You can perform a Mode Superposition Transient Structural analysis that is linked to a pre-stressed Modal analysis:



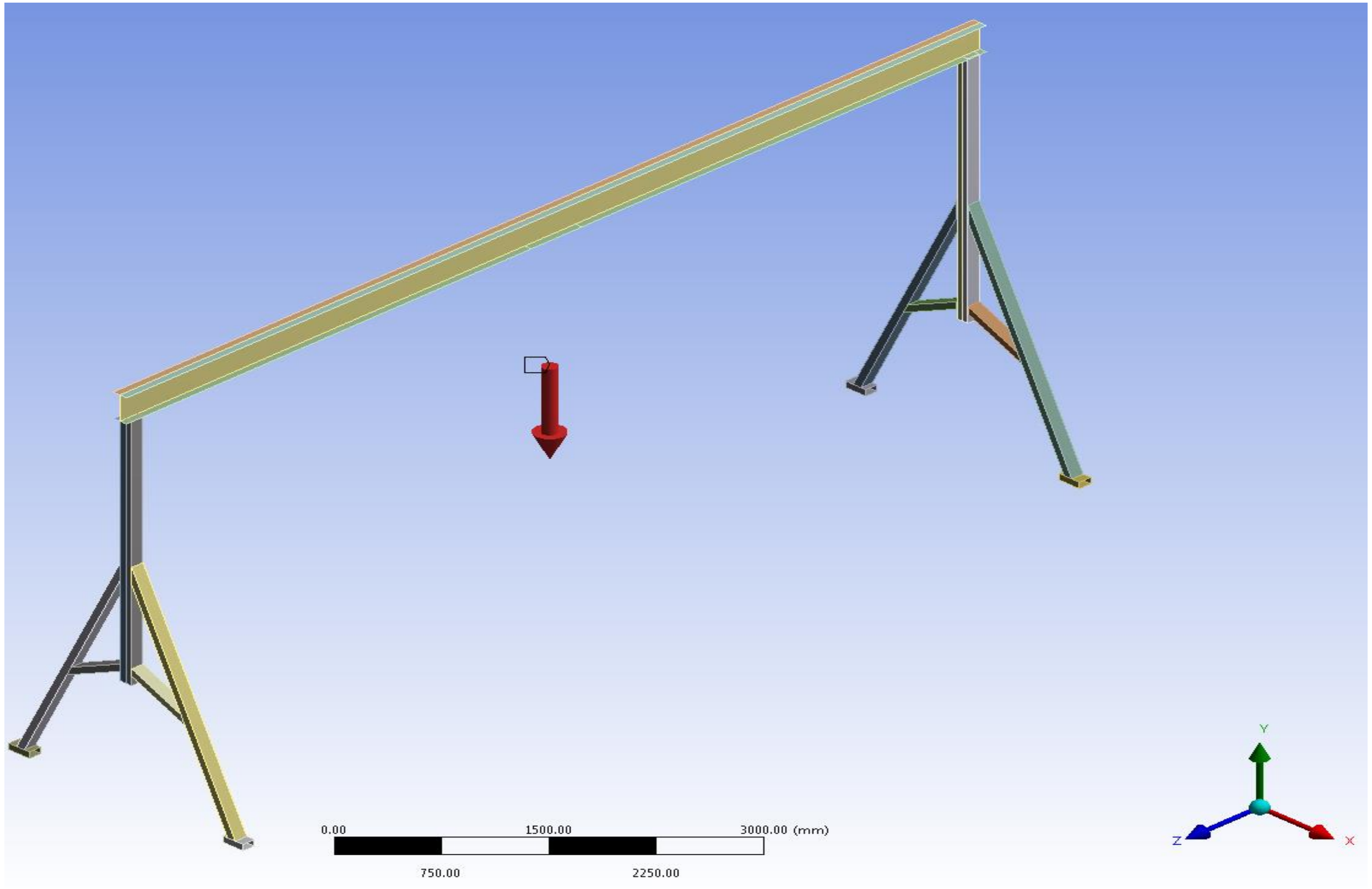


Workshop 09.1: Caster Wheel

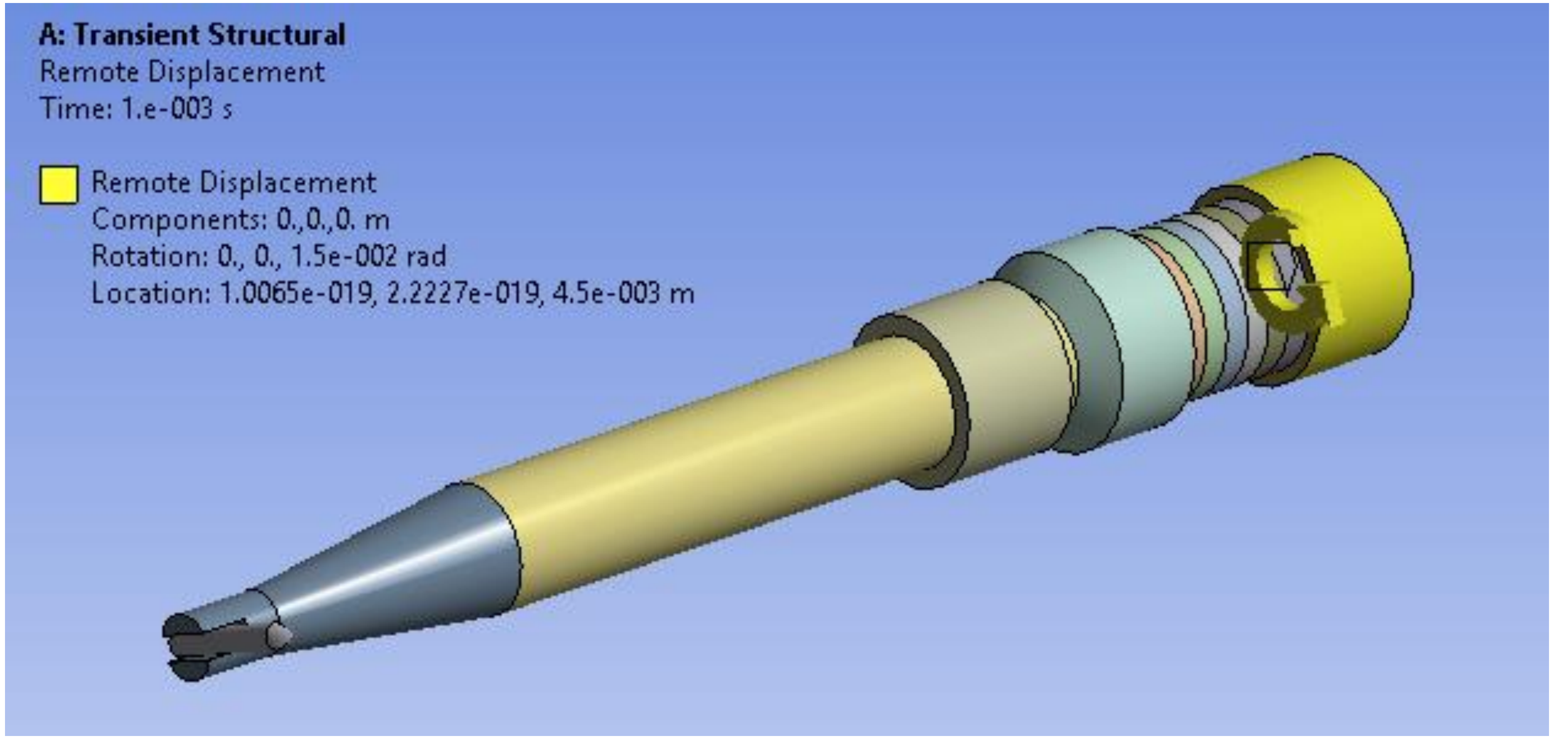




Workshop 09.2: Gantry Crane



Workshop 09.3: Wire Bonder



Ansys

End of presentation