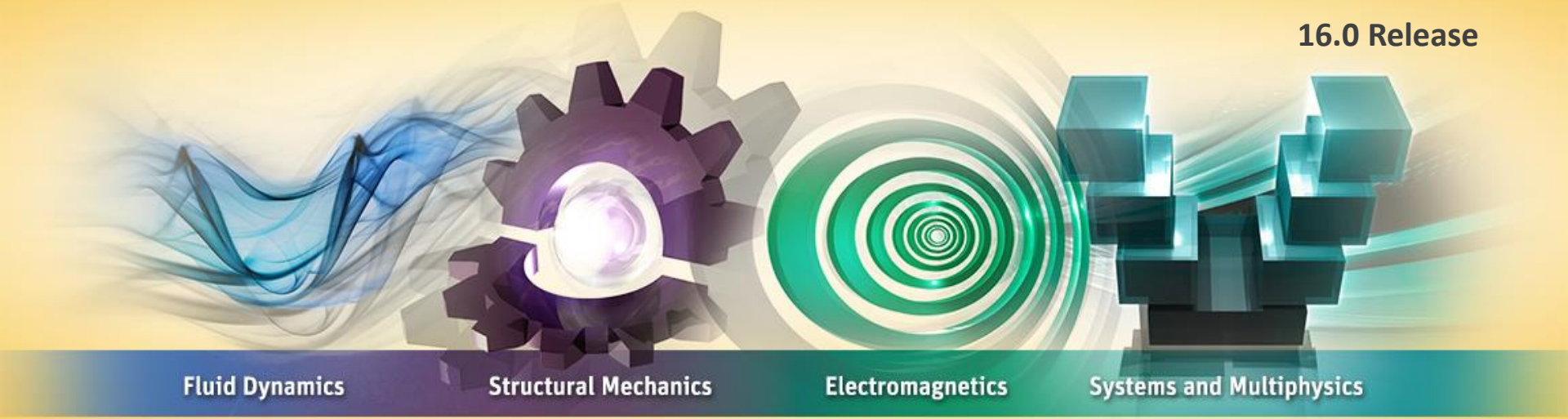
Lecture 10
Multistep Analysis

16.0 Release



Introduction to ANSYS Mechanical

Realize Your Product Promise®

Chapter Overview

In this chapter, aspects of reviewing results will be covered:

A. Multistep Analysis
B. Multistep Setup
C. Multistep Controls
D. Multistep Loads
E. Multistep Postprocessing
F. Solution Combinations
G. Workshop 10.1, Multistep Analysis

A. Multistep Analysis

Step Controls:

- Multiple steps allow a series of static analyses to be set up and solved sequentially.
- For a static analysis, the end time can be used as a counter/tracker to identify the load steps and substeps.
- Results can be viewed step by step.
- Load values for each step can be entered in the “Tabular Data” section provided.

Details of "Analysis Settings"

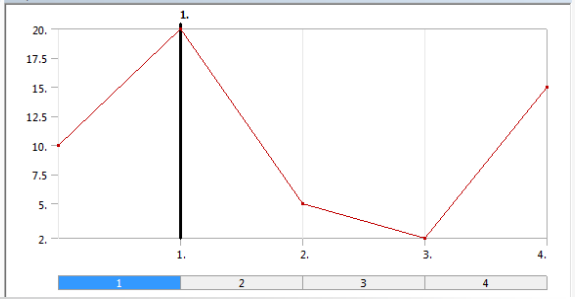
Step Controls	
Number Of Steps	10.
Current Step Number	4.
Step End Time	13. s
Auto Time Stepping	Program Controlled
Solver Controls	

The time and load value are displayed in the graphics window

Details of "Pressure"

Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
ID (Beta)	70
Type	Pressure
Define By	Normal To
Magnitude	Tabular Data
Suppressed	No
Tabular Data	
Independent Variable	Time

Graph



Tabular Data

	Steps	Time [s]	Pressure [MPa]
1	1	0.	10.
2	1	1.	20.
3	2	2.	5.
4	3	3.	2.
5	4	4.	15.
*			

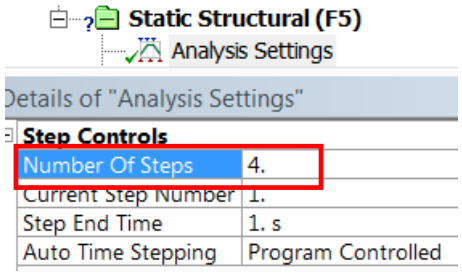
A: 3D Multibodied Bolt
Pressure
Time: 1. s
1/9/2014 3:40 PM

Pressure: 20. MPa

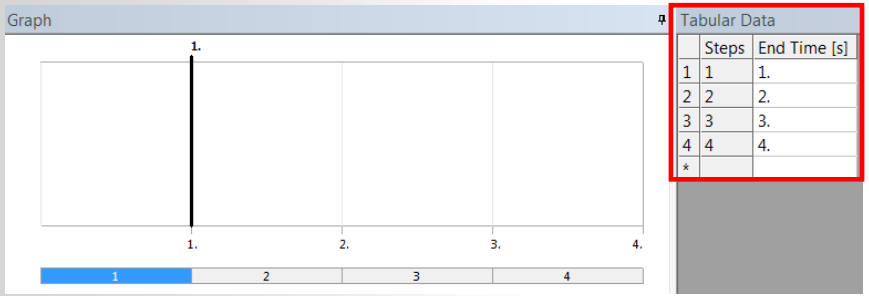
B. Multistep Setup

Multistep Setup:

- Begin by setting the desired “Number of Steps” in the “Analysis Settings” details.



- When the number of steps is defined, the table and graph areas will reflect the setting.

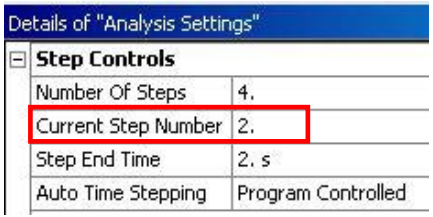


- Note the reference to “Step End Time” should be viewed as a tracking mechanism only. The term “time” here does not imply we are doing a transient analysis.

C. Multistep Controls

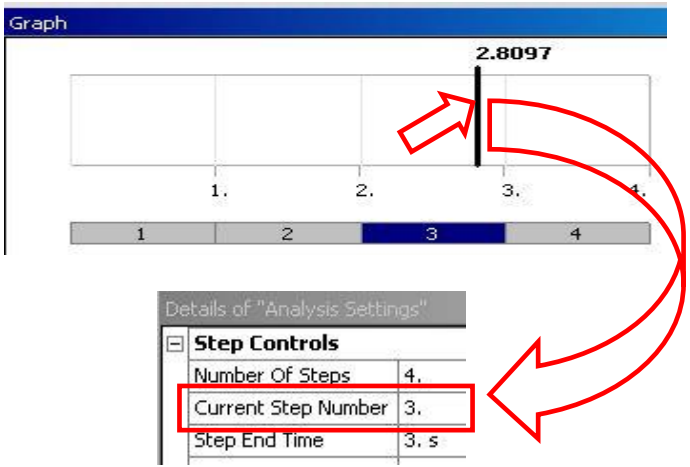
With a multistep analysis each step is controlled as an independent analysis. Loads, supports and analysis settings can be set up individually.

To ensure you are configuring the correct solution always check the "Current Step Number" field in the details.



There are several shortcuts available when selecting which load step(s) you wish to configure:

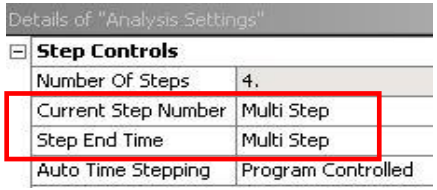
- **Click in the graph area within any time range (note, to select step 3 for example you need only click in the graph anywhere between steps 2 and 3 to activate this step).**

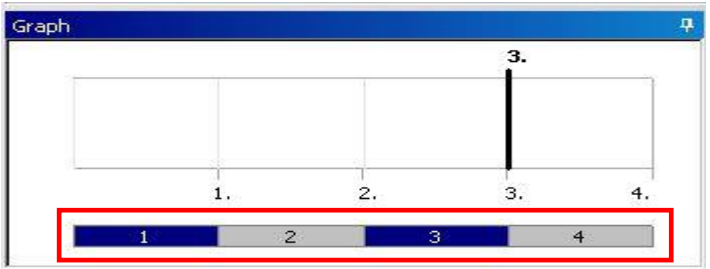


. . . Multistep Controls

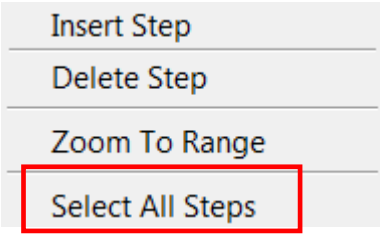
For certain settings you may wish to highlight more than one step at a time:

- **From the bar at the bottom of the graph use the shift or control keys to select multiple steps.**



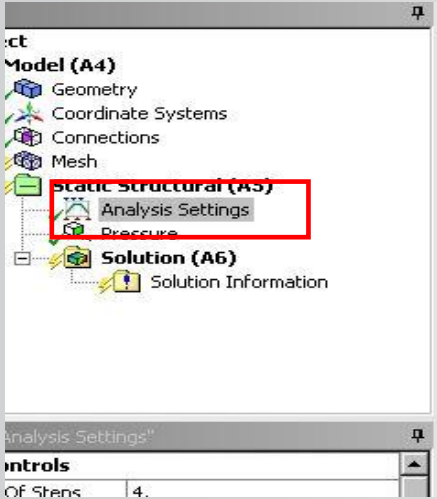


- **You can highlight all steps by right clicking in the graph and choosing “Select All Steps”.**



. . . Multistep Controls

A summary of all the different steps can be viewed by highlighting “Analysis Settings” and then selecting the “Worksheet” icon.



Analysis Settings

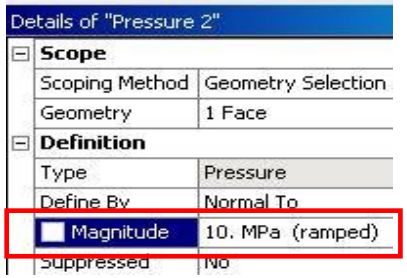
Properties	Step 1	Step 2	Step 3	Step 4
Step Controls				
Step End Time	1.	2.	3.	4.
Auto Time Stepping	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Nonlinear Controls				
Force Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Moment Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Displacement Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Rotation Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Line Search	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Stabilization	Off	Off	Off	Off
Output Controls				
Stress	Yes	Yes	Yes	Yes
Strain	Yes	Yes	Yes	Yes
Nodal Forces	Yes	Yes	Yes	Yes
Contact Miscellaneous	No	No	No	No
General Miscellaneous	Yes	Yes	Yes	Yes
Store Results At	All Time Points	All Time Points	All Time Points	All Time Points

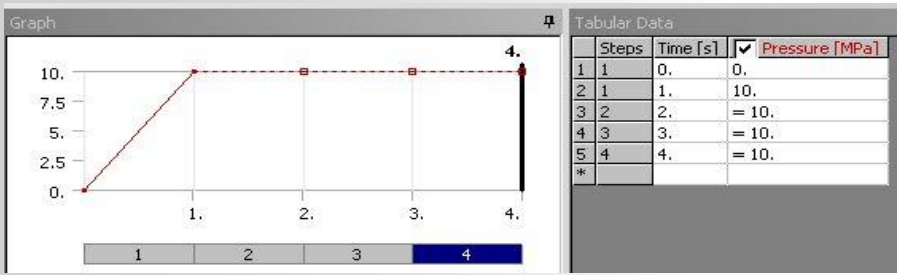
D. Multistep Loads

Since a multistep analysis is a sequence of static steps, loads can vary from one step to another.

Loads are applied the same as with a single step analysis. There are some additional considerations however:

- **For loads that will remain constant over all steps, input a magnitude just as with single step analyses.**
- **The graph and table will reflect the value.**





- **Note, since we are dealing with static analysis the fact that a load is shown as ramped is of no consequence.**

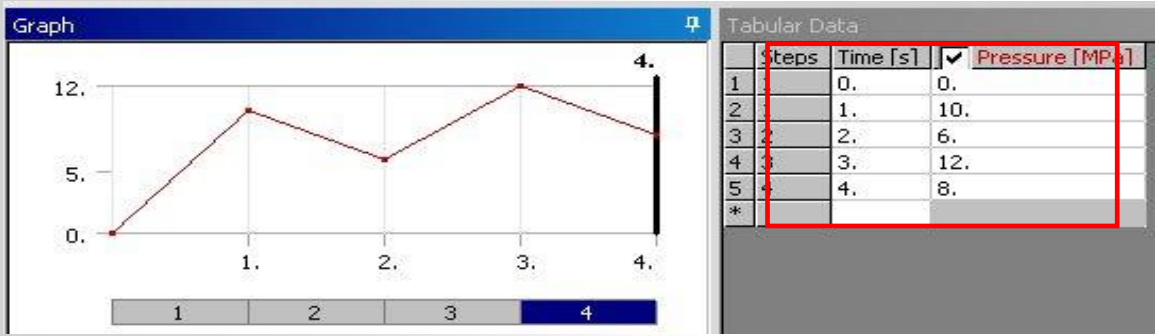
. . . Multistep Loads

To vary the load in different steps change the magnitude to "Tabular Data".

Now use the table to enter the desired values for each step.

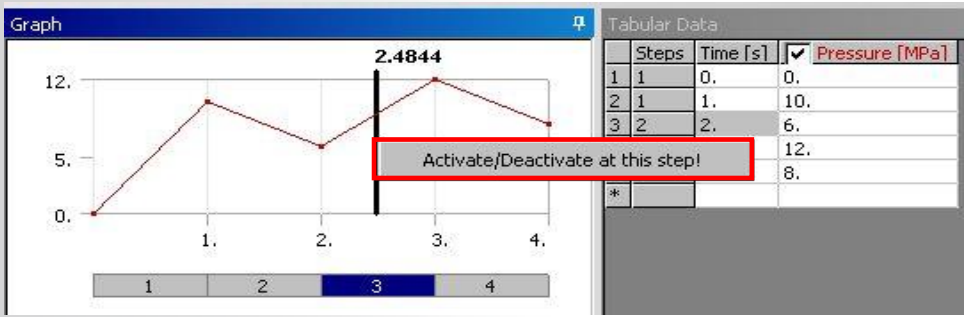
Magnitude	Tabular Data
Suppressed	No
Tabular Data	
Independent Variable	Time

Import...
Export...
Constant
Tabular
Function

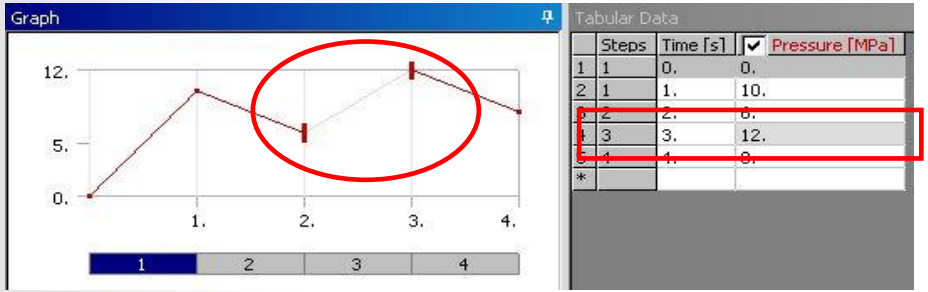


. . . Multistep Loads

For certain loads and/or supports you may wish to have the boundary condition inactive. To do this use the graph or table to choose the step(s) and right click to choose "Activate/Deactivate at this step!".



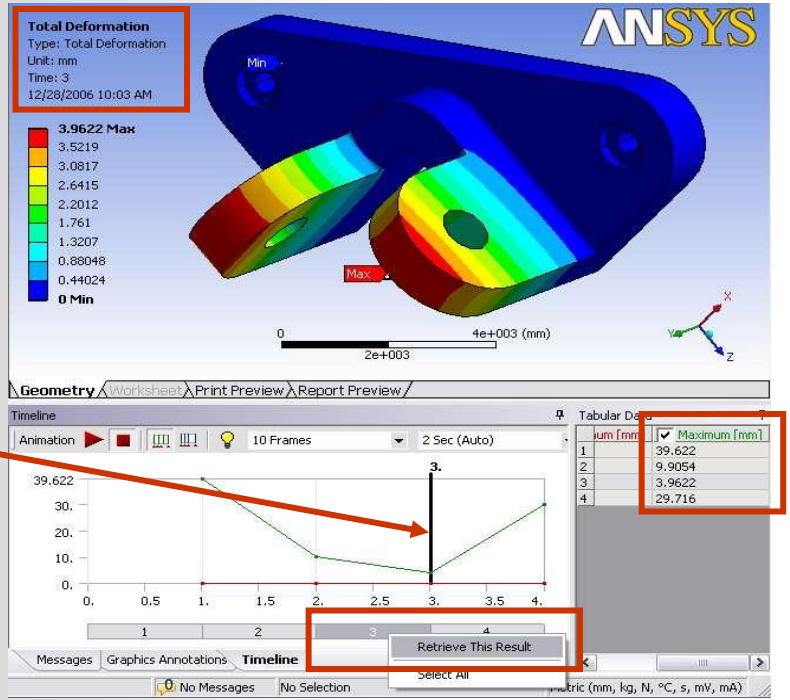
Note the graph and table will graphically indicate inactive steps.



E. Multistep Postprocessing

Results for each individual step can be viewed after solving by selecting the desired step and RMB >“Retrieve This Result”.

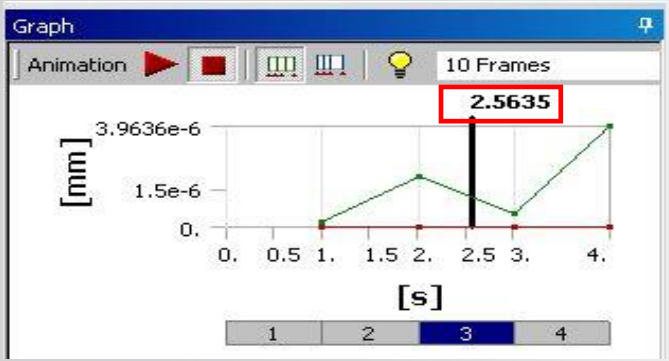
Select desired step and RMB to retrieve result

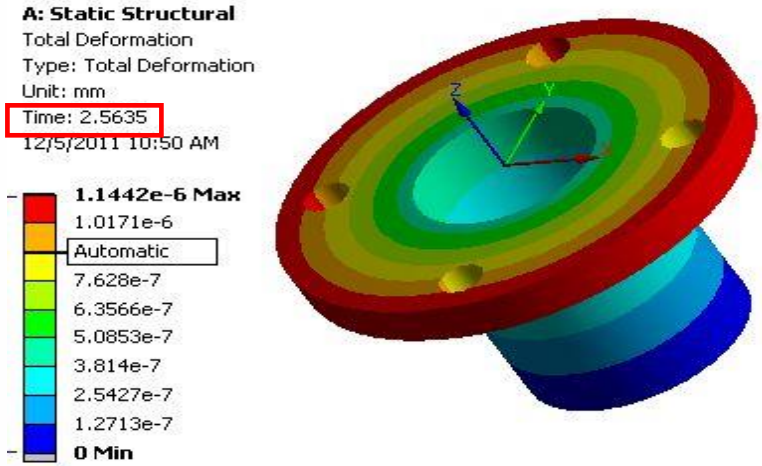


. . . Multistep Postprocessing

When post processing if you choose to retrieve results within a step, Mechanical will return an interpolated value for the result.

If, for example, you retrieve a result from time = 2.5635 (as below) you could interpret this as the point within step 3 where roughly 56% of the load has been applied.





F. Solution Combinations

Solution combinations allow you to obtain solutions that are calculated from other solutions. The advantage of this approach is the calculations are all done within the post processor so no solution time is required.

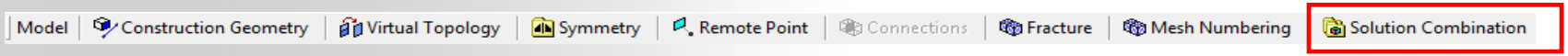
A solution combination may involve multiple environments or may be the result of multiple steps from a single environment.

Solution combinations provide a coefficient for each entry to allow combinations to be scaled.

Solution Combination = Coef 1 * Environment 1 + Coef 2 * Environment 2 + . . .

Solution combinations are valid for static and transient structural analyses.

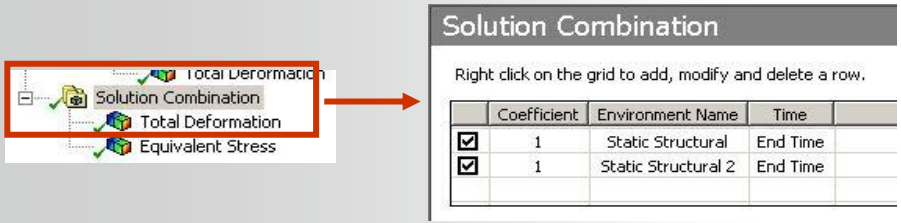
. . . Solution Combinations

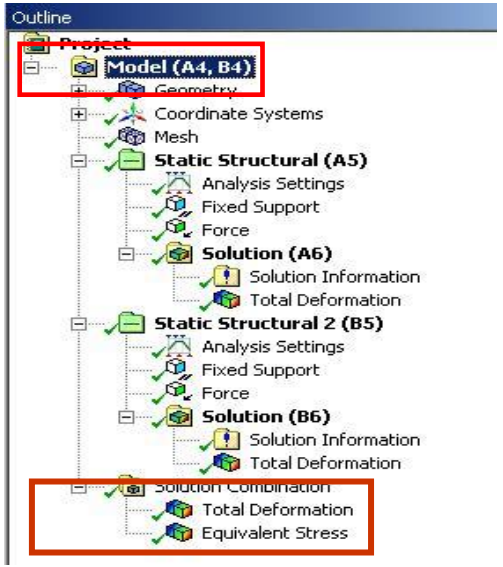


With the Model branch highlighted a “Solution Combination” can be chosen from the context menu.

A new branch is inserted where combined results can be requested and retrieved.

With the Solution Combination branch highlighted, the worksheet view allows multiple environments to be combined.

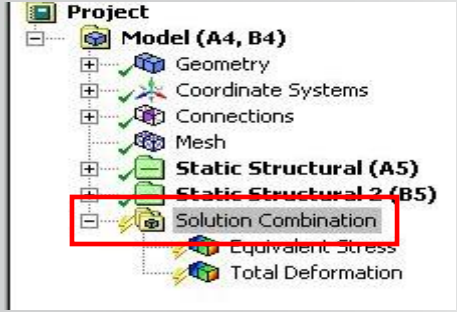


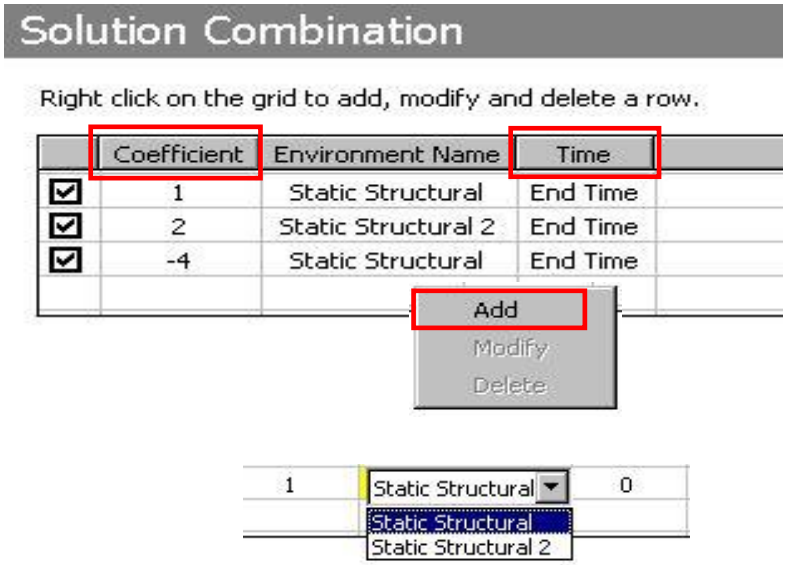


. . . Solution Combinations

Solution combinations are set up using the Solution Combination Worksheet (highlight the Solution Combination branch):

- **RMB in the worksheet to add or delete rows.**
- **Select the environment from a drop down list.**

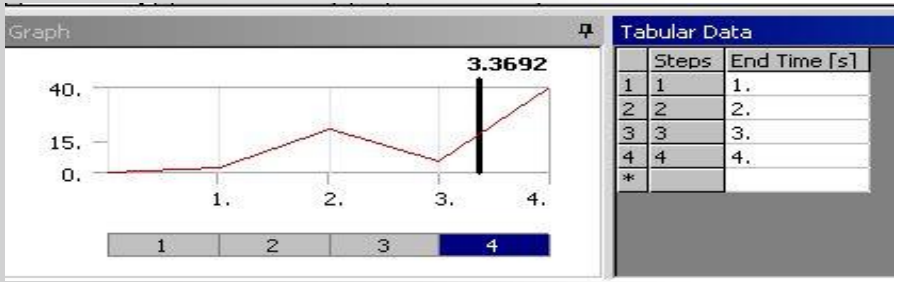


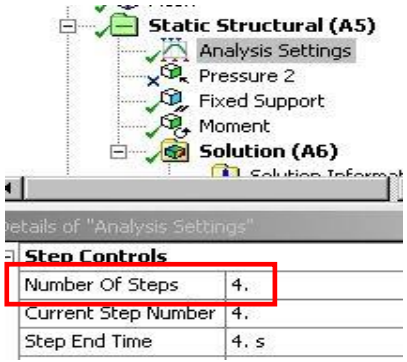


Note both a "Coefficient" (scale factor) and a time point (multi-step or transient analyses), can be specified in the worksheet.

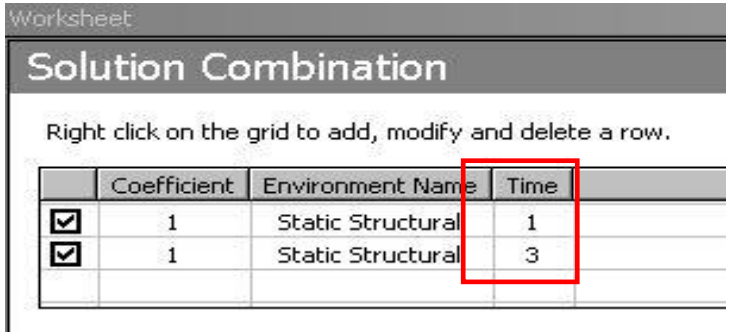
. . . Solution Combinations

Depending on the type of project, there are different ways to specify solution combinations.



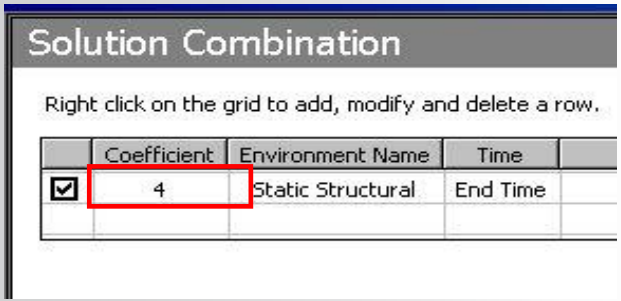


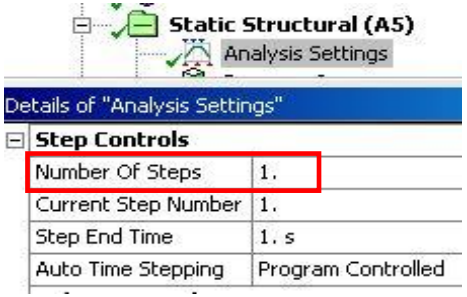
In this case a multistep structural analysis was performed. A combination is setup by combining results from different time points from the same environment.

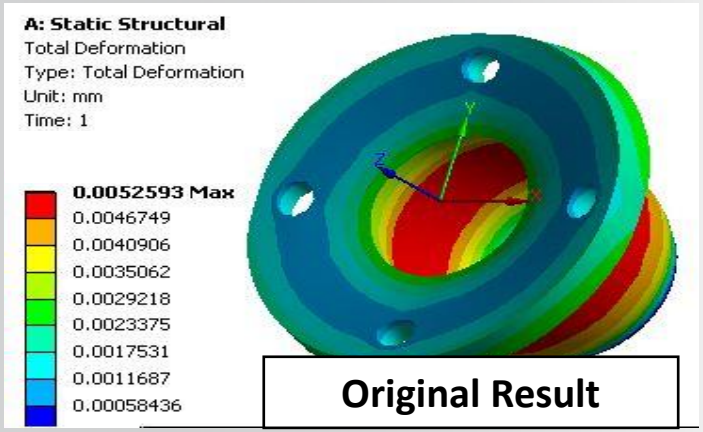


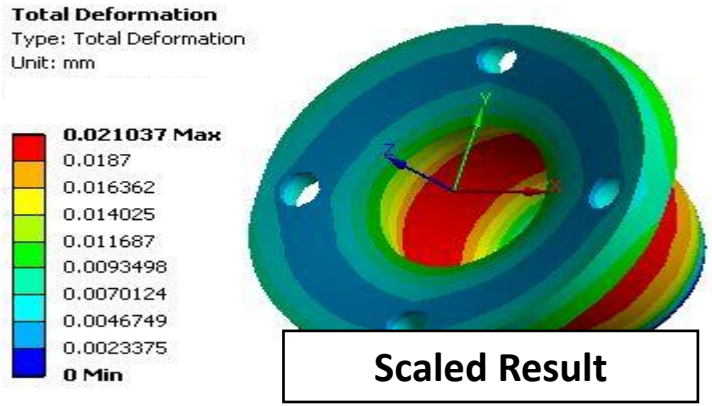
. . . Solution Combinations

A solution combination can be used to scale a single analysis as shown here.



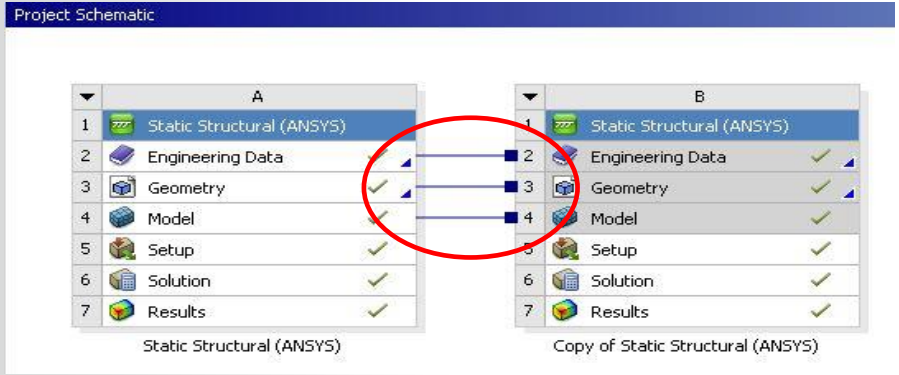


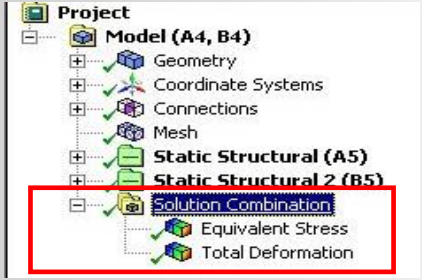


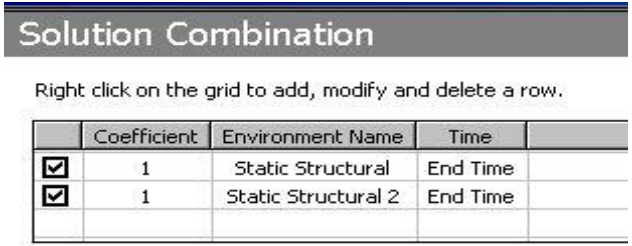


. . . Solution Combinations

If a project contains multiple analysis systems that share geometry and model cells their solutions can be combined as well.

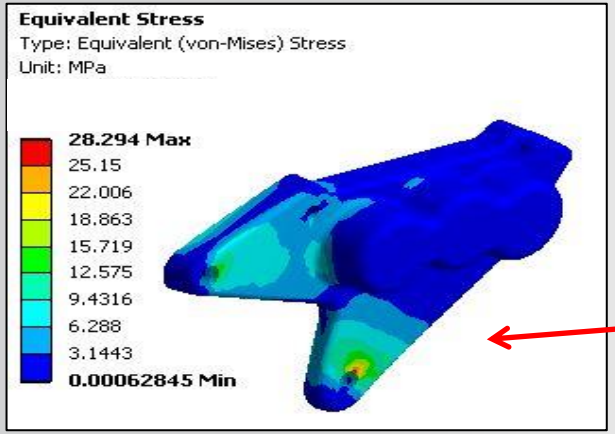




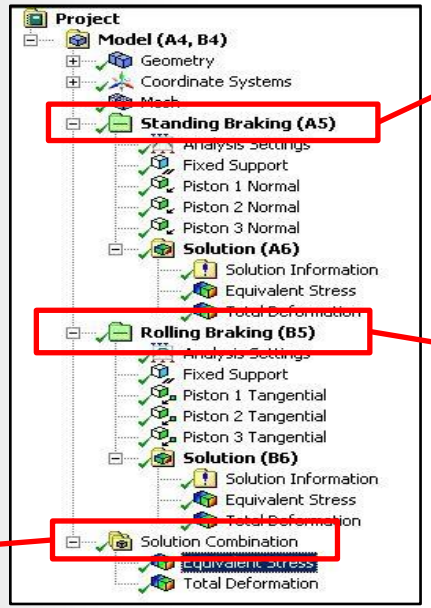


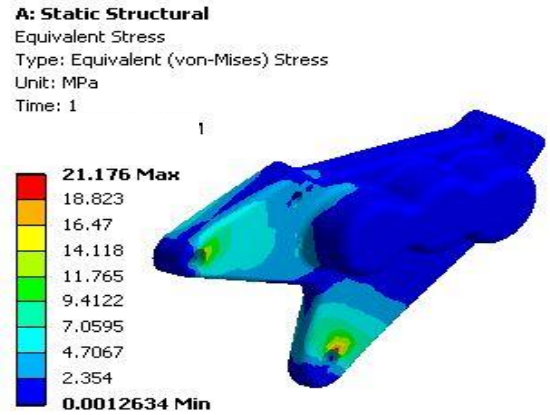
. . . Solution Combinations

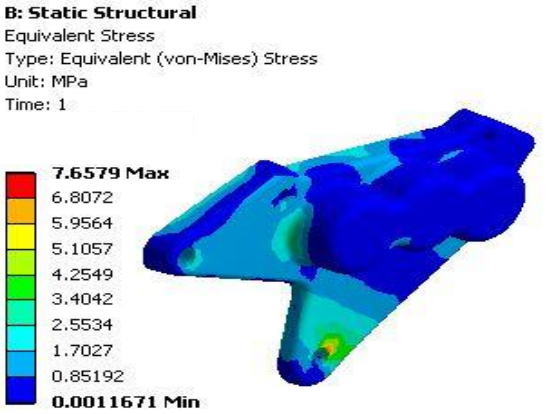
Example: a brake caliper is simulated in both standing and rolling configurations. After the 2 environments have solved a resulting combination shows the effect of both.



Solution Combination







G. Workshop 10.1

- **Workshop 10.1 – Multistep Analysis**