AEROSPACE,
DEFENCE & RAILWAYS

Linear Dynamic System Analyses with Creo Simulate

– Theory & Application Examples, Capabilities, Limitations –

9th SAXSIM 2017, TU Chemnitz, 28.03.2017

Rev. 1.2 | 07.04.2017

Dr.-Ing. Roland Jakel

altran

At a glance: Altran, a global leader

20+

Countries

30,000+

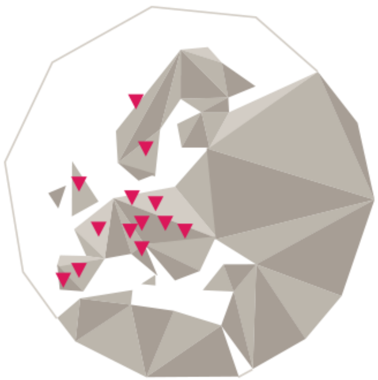
Innovation Makers

5

Industries

€ 2.120bn

2016 Revenues



EUROPE

—

Austria, Belgium, France, Germany, Italy, Luxembourg, Netherlands, Norway, Portugal, Spain, Sweden, Switzerland, United Kingdom

ASIA

—

China, India, Malaysia, Middle East

AMERICAS

—

Canada, United States of America, Mexico

What do we do? The industries

Our presence in the main business sectors enables us to partner with key players in the market:

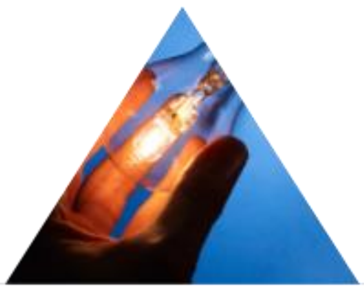
24%

AEROSPACE, DEFENCE & RAILWAY

EADS, AIRBUS (AN EADS COMPANY), DB, DIEHL Aircabin, Rolls-Royce, DLR, AEROLIA, Lufthansa Technik, Triumph Group, Inc., OHB SYSTEM, LIEBHERR, THALES

25%

ENERGY, INDUSTRY & LIFE SCIENCES



SIEMENS, AREVA, Shell, e.on, ZEISS, ABB, Dräger, VATTENFALL

23%

AUTOMOTIVE, INFRASTRUCTURE & TRANSPORTATION

Audi, BMW, VW, DAIMLER, Ford, BOSCH, Continental, ZF

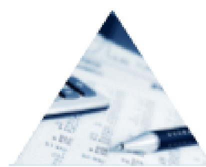
16%

TELECOMS & MEDIA

Deutsche Telekom, Telefónica, T-Systems, vodafone, KabelBW, NXP, groupm

12%

FINANCIAL SERVICES & GOVERNMENT



Deutsche Bank, COMMERZBANK, HypoVereinsbank (UniCredit Group), Volkswagen Bank

altran

Table of Contents

1. Introduction to dynamic analysis theory in Creo Simulate

1.1 Basic equations for dynamic analyses

Basic equation for dynamic systems

- Creo Simulate can only solve dynamic problems which can be described with help of the following linear differential equation (DEQ) of second order:

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

- Herein, we have: $[M]$=mass matrix, $[C]$=damping matrix, $[K]$=stiffness matrix, $\{F\}$=force vector, $\{x\}$=displacement vector and its derivatives with respect to time

Modal analysis as basis for all dynamic studies

- In order to determine the fundamental frequencies of a mechanical structure, first a modal analysis is performed before any subsequent dynamic studies are carried out
- The equation which is solved here is a special case of the above differential equation:

$$[M]\{\ddot{x}\}+[K]\{x\}=\{0\}$$

- Hence, for fundamental frequency determination in Creo Simulate, no damping $[C]$ is taken into account, so the real mechanical structures to be computed may only contain little damping to keep the error small
- Damping is taken into account only during subsequent dynamic analysis like shown on the next slide

altran

1. Introduction to dynamic analysis theory in Creo Simulate

1.2 Solution method coded

Solution Sequence:

- In Creo Simulate, the mentioned linear differential equation of second order,

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

  is not solved directly in physical coordinates, but in the following way:
- Before any dynamic analysis is performed in Simulate, the damping-free modal analysis, $[M]\{\ddot{x}\}+[K]\{x\}=\{0\}$, is carried out to obtain the modal base (eigenvector matrix) for the modal transformation
- The system is then transformed from physical space $\{x\}$ to modal space $\{\xi\}$ by replacing the physical coordinates with modal coordinates: $\{x\} = [\phi]\{\xi\}$
- Herein, $[\phi]$ is the eigenvector matrix, and $\{\xi\}$ modal coordinates; $[\phi]$ has a number of rows equal to the DOF in the model, and columns equal to the number of modes; $\{\xi\}$ has one column and rows equal to the number of modes
- In a subsequent dynamic analysis, in which modal damping $[C] = 2\beta\ [M]\omega$ and a forcing function is added, we have $[M]$, $[C]$ and $[K]$ as diagonal matrices now in modal coordinates!
- After the solution is performed, the solution is transformed back into physical space for post-processing

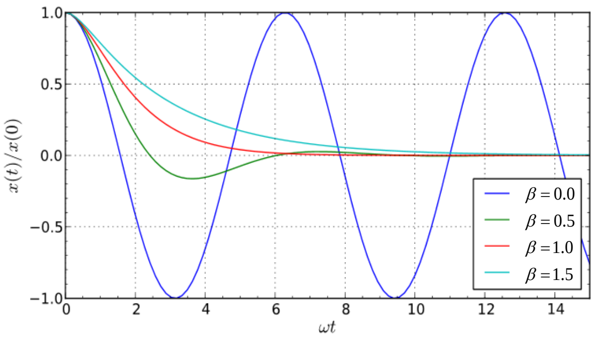
Remark: This solution method is used in many FEM codes for linear, small damped dynamic systems because of its computational efficiency (only diagonal matrices) and various practical advantages, e.g. different dynamic analysis types and damping values can be rapidly executed on base of the existing modal analysis!

1.3 Damping

Let's now look at the modal damping $[C] = 2\beta[M]\omega$ mentioned on the previous slide:

- For a simple, linear damped one-mass-oscillator (=harmonic oscillator with mass m, velocity proportional damping constant c and spring stiffness k), the damping ratio β (in German "Lehrsches Dämpfungsmaß") is

$$\beta = \frac{c}{2m\omega_0} = \frac{c}{2\sqrt{mk}} = \frac{c}{c_{crit}}$$



- Herein, c_{crit} is the so called "critical damping", leading to the aperiodic limit case $\beta = 1$, in which the oscillator just does not overshoot (red curve right)

- The damping ratio β is often expressed in % (like in Simulate), so we have
  $\beta > 100\,\%$: very strong damping (creeping case)
  $\beta = 100\,\%$: aperiodic limit case (critical damping = no overshooting)
  $\beta = 50\,\%$: max. damping supported in Creo Simulate (green curve in the diagram)
  $\beta = 1...4\,\%$: typical values used for many small damped, real mechanical structures
  $\beta = 0\,\%$: no damping

- Remember:
  The undamped and damped fundamental angular frequencies are related as follows:

$$\omega_0 = 2\pi f_0 = \sqrt{\frac{k}{m}}; \quad \omega = \omega_0\sqrt{1-\beta^2}$$

1.3 Damping

There are many methods to measure damping, like e.g.:

- Logarithmic decrement δ (relative damping):

$$\delta = \ln\frac{x_1}{x_2} = \frac{1}{n}\ln\frac{x_1}{x_{n+1}}$$

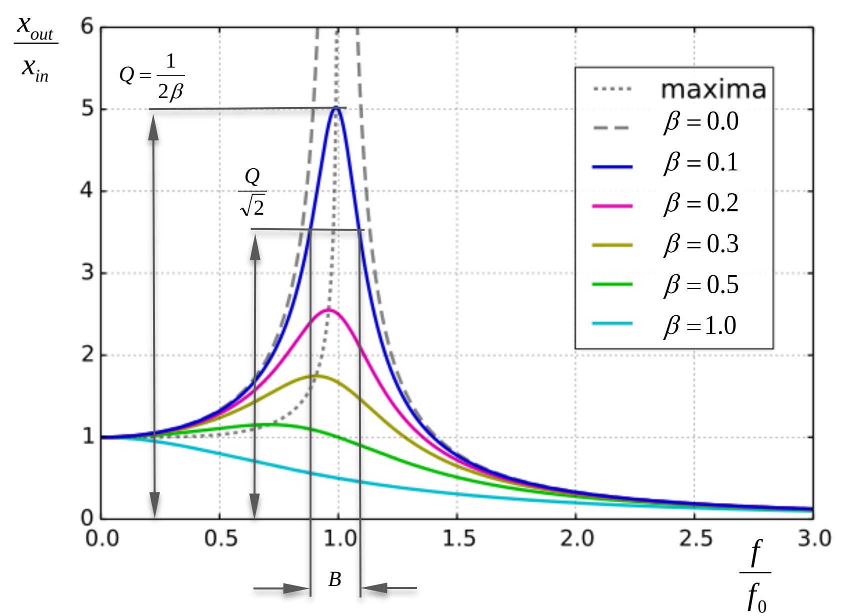
- δ can simply be obtained from the decrease of amplitudes x_i in an experiment evaluated in the time domain where the structure is dying out over the time with its natural frequency
- δ can then be transferred into β with help of the following equation (for small β):

$$\delta = \frac{2\pi\beta}{\sqrt{1-\beta}} \approx 2\pi\beta$$

- Another method for an experiment evaluated in the frequency domain is measuring the bandwidth B (in German "Halbleistungsbandbreite") or the magnification factor Q of the oscillator (for $\beta << 1$):

$$Q \approx \frac{1}{2\beta} = \frac{f_0}{B} \Leftrightarrow \beta = \frac{1}{2Q} = \frac{B}{2f_0}$$

- This is exemplarily depicted right for $\beta = 0.1$



1. Introduction to dynamic analysis theory in Creo Simulate

1.4 Limitations of the solution coded in Creo Simulate

Limitations of the solution method

- We have only a linear system (all matrices are constant), that means no nonlinearities can be taken into account like
  - Contact
    (therefore unknown force-vs-time curves of impact problems cannot be computed, but have to be assumed and then applied as external force function vs. time [4])
  - Change of constraints
    (all dynamic analyses use the constraints defined in the modal analysis!)
  - Nonlinear material
  - Nonlinear damping (e.g. from friction, hydraulic devices,…)
- Only modal damping can be applied to keep the damping matrix diagonal and therefore run times short (this damping is called in German language “Bequem-lichkeitshypothese“ – “hypothesis of comfort”)
- A severe limitation is that no discrete damper, not even a linear one, is supported (discrete linear springs are supported in dynamic analysis!)
- A discrete linear damper can only be approximated, this means those mode shapes which are damped by a discrete damper may be taken into account with a higher, individual modal damping
- Therefore, in Creo Simulate the damping can be applied in three ways:
  - Constant for the complete frequency domain
  - As function of frequency
  - For individual modes

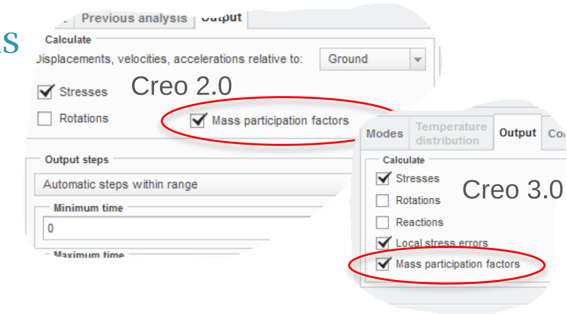
1. Introduction to dynamic analysis theory in Creo Simulate

1.5 Result quality assurance when performing dynamic analysis

- The solution method is of approximative nature even for ideal linear structures:
  - Continuum mechanical structures have an infinite number of natural modes, but for computation only a finite number of modes can be taken into account by the FEM code
  - Therefore, the modal base is cut after a certain number of modes, but in theory, the exact solution can only be obtained by superposing all modes to the total response!
  - Depending on this number of modes taken (or better not taken) into account, the analysis results may become pretty inaccurate!
- Therefore, it is in the responsibility of the user to assure that a sufficient number of modes is taken into account to obtain results of the required accuracy!
- There are a couple of methods how the result quality can be assured:
  - Compare the results with an analytical solution (if existing!)
  - Repeat the dynamic analysis with an increasing number of modes and see if the results converge (typically done in analyses with force excitation)
  - Only for analyses with base excitation: Check if the sum of the effective masses $m_{i,eff}$ of all modes taken into account is close to the total mass of the structure
  - A rule of thumb is to take into account all modes with eigen frequencies until at least the double value of the excitation frequency, but often even this may not be sufficient (sometimes the author had to use >4x the max. excitation frequency)
  - Check that for an excitation frequency of Zero Hz, the results match the results of a separately performed linear static analysis undertaken with the same model!
Note: There will always be a difference at the location of force introduction!
  - …

1. Introduction to dynamic analysis theory in Creo Simulate

1.5 Result quality assurance when performing dynamic analysis



Participation factors and effective masses:

- For all dynamic analyses with <u>base point excitation</u>, the code allows to compute mass participation factors and effective masses:
  - ➢ The effective mass $m_{i\ eff}$, multiplied with the base point acceleration, reflects the share this mode has to the total base point reaction force!
  - ➢ Note the effective mass of the mode depends on the excitation direction!
  - ➢ The <u>sum of the squares of the participation factors</u> is the total absolute mass of the structure!
- A Simulate example output for a 2-mass oscillator with a total mass of 2 kg: (=0.002 t; analysis was performed in mm, t, s unit system!)

Mode	frequency	part. factor	eff. mass	tot. mass
1	5.445276e+01	4.351645e-02	94.7%	94.7%
2	1.425516e+02	1.027535e-02	5.3%	100.0%

$m_{1,eff} = (4.351645e{-}02)^2\ t = 1.894\ kg = 94.7\ \%$

$m_{2,eff} = (1.027535e{-}02)^2\ t = 0.106\ kg = 5.3\ \%$

$\text{Total mass} = m_{1,eff} + m_{2,eff} = 2.000\ kg = 100.0\ \%$

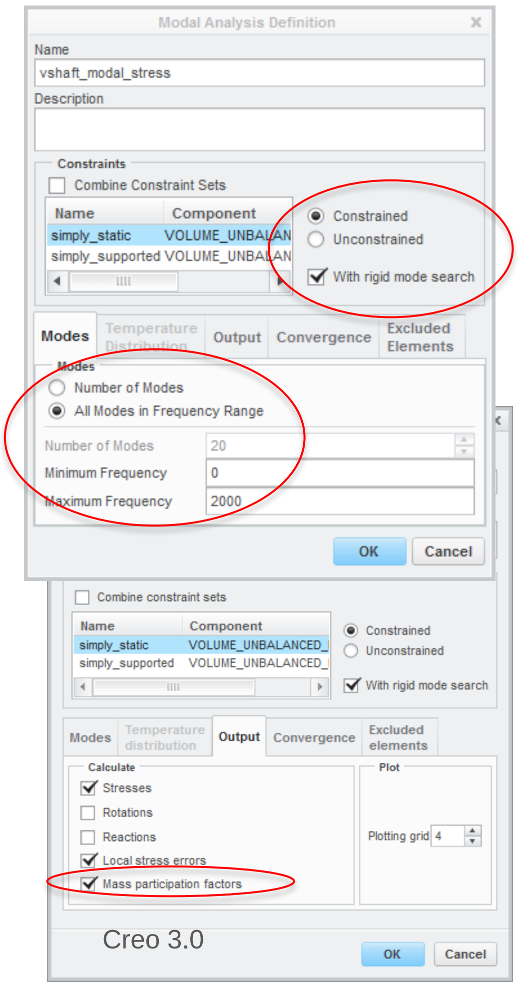
- Note: Modes with an effective mass of Zero cannot be excited over the base points (=the interface the structure is mounted to), but of course they may be by another external force directly acting on certain points of the structure!
- Effective masses are therefore just of importance for base point excited structures and not for force excited structures!
- Since Creo Simulate 3.0, the mass participation factors can also be requested in a modal analysis (output very comfortably for all three translations and rotations!)

2. Modal analysis

2.1 Standard modal analysis

2.1.1 Introduction

- Like mentioned in chapter 1, the modal analysis just solves
  $$[M]\{\ddot{x}\}+[K]\{x\}=\{0\}$$
- Note again damping $[C]\{\dot{x}\}$ is not taken into account in the Simulate modal analysis, so fundamental frequencies may appear in reality at slightly lower frequencies than predicted
- For typical damping around 1-4 %, this influence is negli-gible, but for the max. modal damping supported in the subsequent dynamic analysis (50 %), it may be up to 13.4%:
  $$\omega=\omega_0\sqrt{1-0.5^2}=\omega_0\cdot 0.866$$
- The user has the following choices to request modes:
  - Number of modes (always starting at Zero Hz)
  - Al modes in frequency range (with arbitrary min. and max. frequency)
- The code supports constrained and unconstrained ("free-free") modal analysis with rigid mode search
- Displacements (mode shapes) are always output as result; optional are stresses, rotations (for beams and shells), local stress errors, and new in Creo 3.0 mass participation factors
- Until Creo 2.0, mass participation factors could only be requested in subsequent dynamic analysis with base point excitation

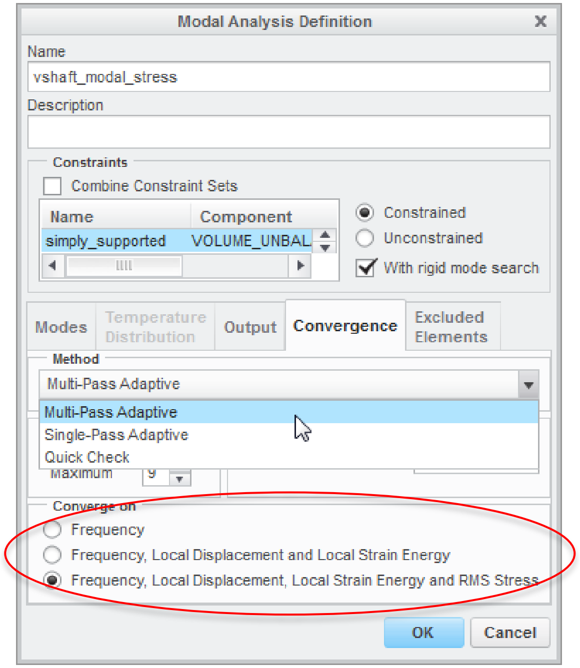


2. Modal analysis

2.1 Standard modal analysis

2.1.1 Introduction

- All known convergence methods are supported, with the exception of multi-pass adaptive convergence on measures (as supported in static analysis)
- Note the plotting grid must be set in the modal analysis, too, and cannot be changed in the subsequent dynamic analysis!

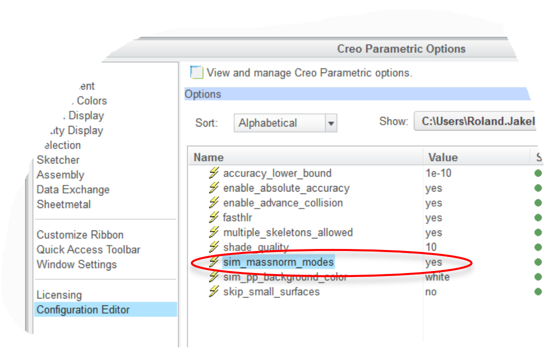


Mode shape output:

- Per default, the eigenvector displacements (mode shapes) are output unit normalized (=max. disp. magnitude scaled to 1), but the user may request mass normalization acc. to the equation

$$\{x_i\}^T \cdot [M] \cdot \{x_i\} = 1$$

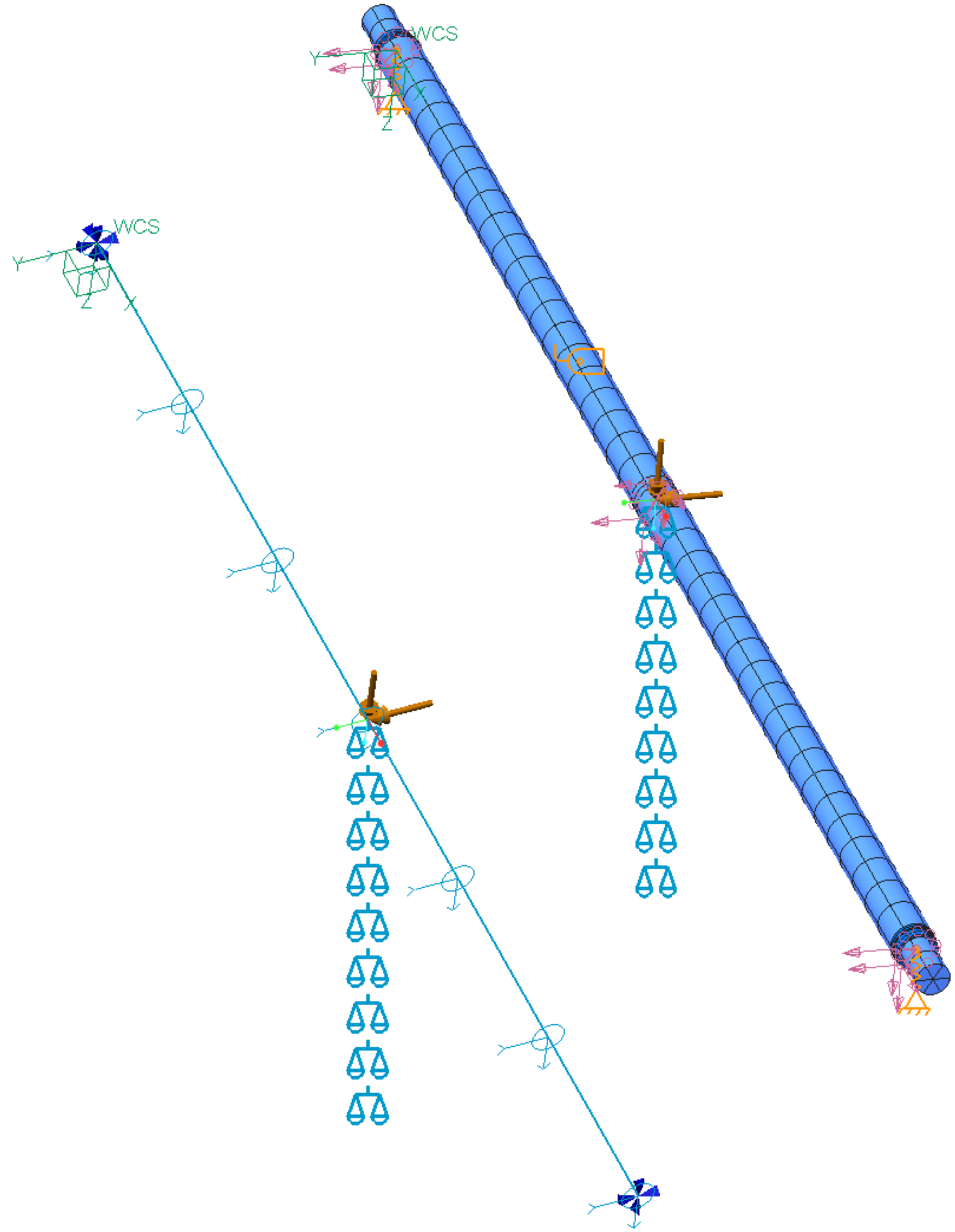
- This is sometimes advantageous since modal stress (if requested as result) is always output for mass normalized mode shapes, and for meaningful modal stress evaluation it does not make sense to use different normalizations for displacements and stresses (see chapter 3.2)



2. Modal analysis

2.1 Standard modal analysis

2.1.2 Example



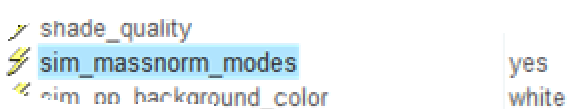
A long, slim drive shaft

- Steel shaft length 500 mm, diameter 13 mm (E=190 GPa, ν=0.3, ρ=7.85g/cm^3)
- Simply supported
- 1st fundamental frequency: $f_0 \approx 100$ Hz

Analysis as

1. simple 2 p-beams model (=much faster)
2. volume model with help of a mapped mesh (=better visualization of results)

- Request for mass-normalized displacement output:



2.1 Standard modal analysis

2.1.2 Example

Modal analysis results:

Beam model (2 p-beams only!): (CPU time 0.47 s) — Volume model (420 p-solids): (CPU time 150.45 s)

Mode	Frequency (Hz)	Convergence	Mode	Frequency (Hz)	
1	2.025479e-05	0.0%	1	1.639796e-03	Rigid body mode (shaft rotation)
2	1.003814e+02	0.0%	2	1.003679e+02	1st bending mode
3	1.003814e+02	0.0%	3	1.003679e+02	
4	4.005557e+02	0.0%	4	4.003401e+02	2nd bending mode
5	4.005557e+02	0.0%	5	4.003401e+02	
6	8.976563e+02	0.0%	6	8.965832e+02	3rd bending mode
7	8.976563e+02	0.0%	7	8.965832e+02	
8	1.587066e+03	0.3%	8	1.583717e+03	4th bending mode
9	1.587066e+03	0.3%	9	1.583717e+03	
10	2.459869e+03	0.0%	10	2.398748e+03	axial "pumping"
11	2.462784e+03	1.1%	11	2.454544e+03	5th bending mode
12	2.462784e+03	1.1%	12	2.454544e+03	
13	3.051092e+03	0.0%	13	2.980194e+03	1st torsional
14	3.517350e+03	1.6%	14	3.500247e+03	6th bending mode
15	3.517350e+03	1.6%	15	3.500247e+03	
16	4.764403e+03	12.0%	16	4.710566e+03	7th bending mode
17	4.764403e+03	12.0%	17	4.710566e+03	
18	6.102184e+03	0.0%	18	5.959154e+03	2nd torsional
19	6.312261e+03	16.9%	19	6.073706e+03	8th bending mode
20	6.312261e+03	45.0%	20	6.073706e+03	

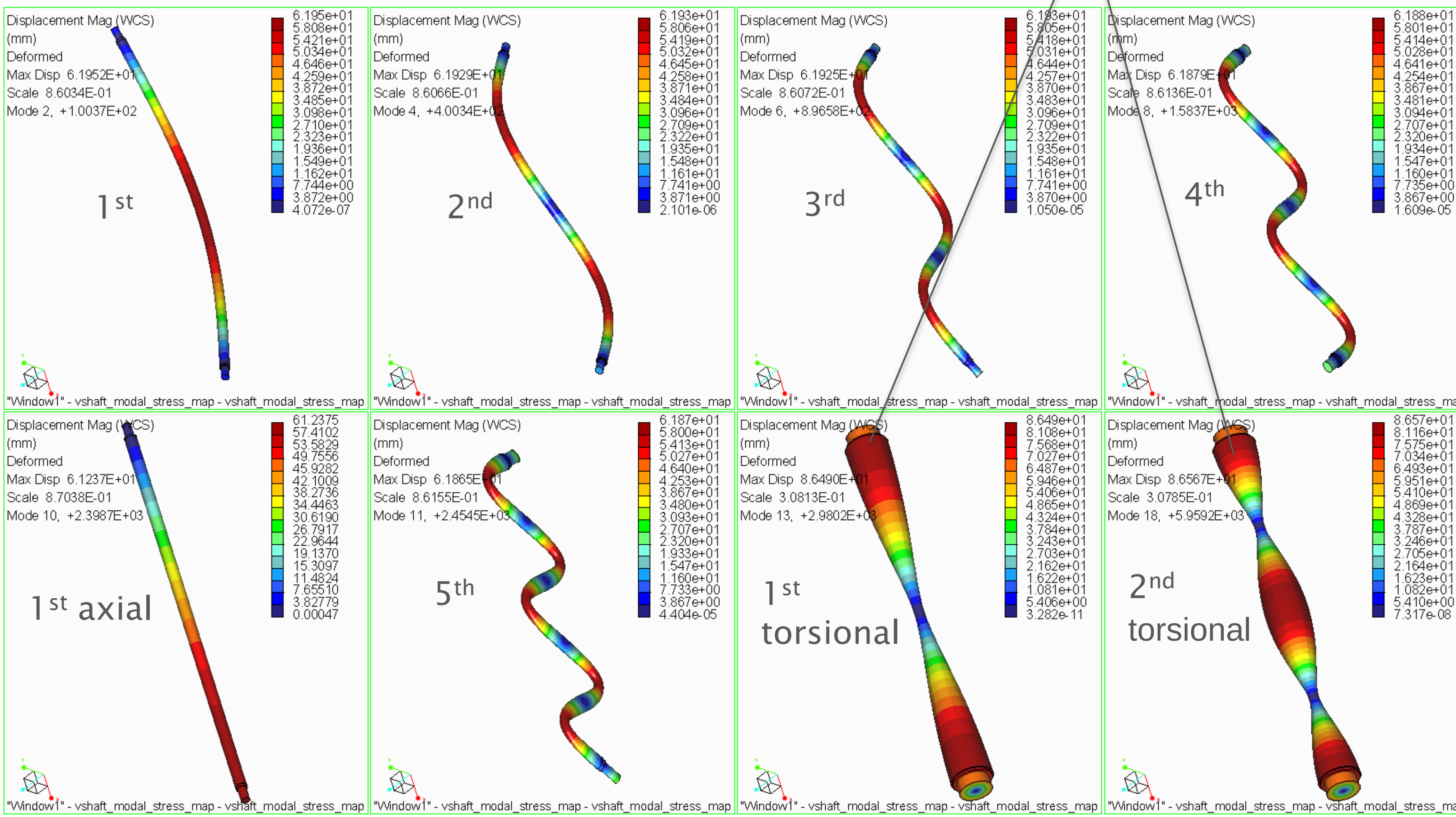
Note: Modes with same frequency, respectively, occur because of the rotational symmetric structure (bending may appear in any lateral direction)!

2. Modal analysis

2.1 Standard modal analysis
2.1.2 Example

Virtual shaft thickness increase under torque just because of linearized theory and displacement scaling!

<u>Mode shapes</u> (with mass normalized displacement output!)



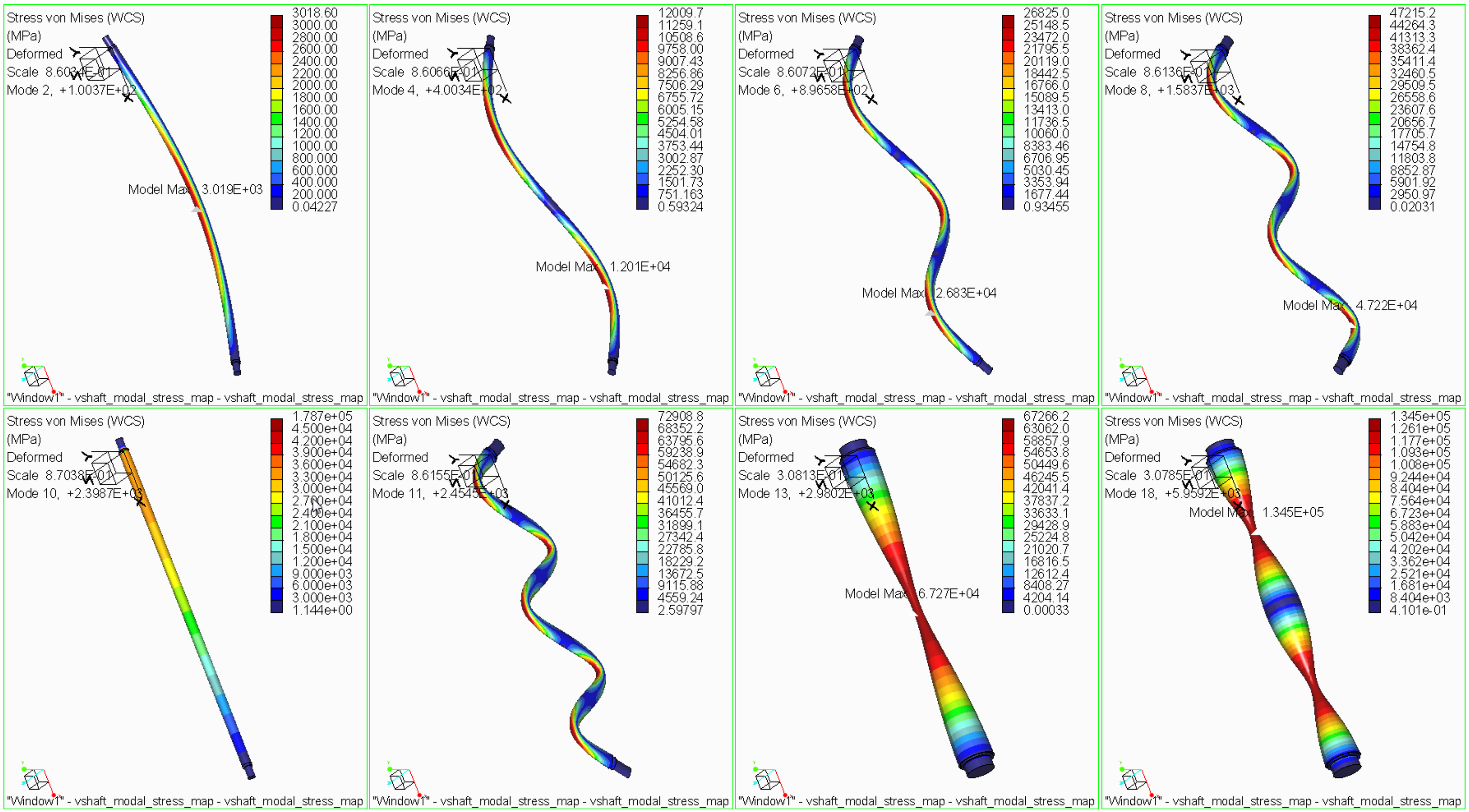
altran

2. Modal analysis

2.1 Standard modal analysis

2.1.2 Example

<u>Modal von Mises stress</u> (always computed for mass normalized displacement output!)



2. Modal analysis

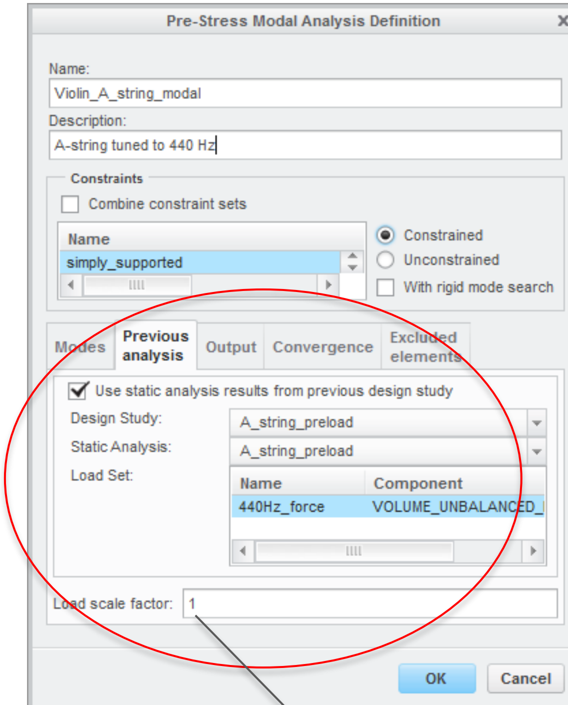
2.2 Modal analysis with prestress

2.2.1 Introduction

- Allows to take into account fundamental frequency changes from preloads:
  - ➢ Tensile stresses in slim structures increase the fundamental frequency (e.g. a music instrument string or a turbine blade under centrifugal loads)
  - ➢ Compression stresses in slim structures decrease the fundamental frequency
  - ➢ Bending preloads do not significantly change fundamental frequencies, since tensile and compressive stress loaded regions of the structure are balanced and compensate each other
- Basis of a modal analysis with prestress is a linear static analysis that defines the preloaded state created by the preload force $\{F_P\}$. From this preloaded state, the stress stiffness matrix $[K_\sigma]$ is computed for each integration point of each element
- The modal analysis with prestress then solves the following equation:

$$[M]\{\ddot{x}\}+\left([K]+\left[K_\sigma\left(\{F_p\}\right)\right]\right)\{x\}=\{0\}$$

- Note that unlike in a Creo Simulate static analysis with prestress, the static prestress cannot be combined with dynamic stress in subsequent dynamic analysis output!



Note the force applied in the previous static analysis defining the preloaded state can be optionally scaled, so it does not need to be re-run if another preload is applied!

2. Modal analysis

2.2 Modal analysis with prestress

2.2.2 Relationship with other analyses taking into account preloads

- Linear buckling analysis (“Eigenvalue buckling”): It is also based on a previous static analysis defining the preloaded state, but solves the equation:

  $$([K]+\lambda[K_\sigma(\{F_P\})])\{x\}=\{0\}$$

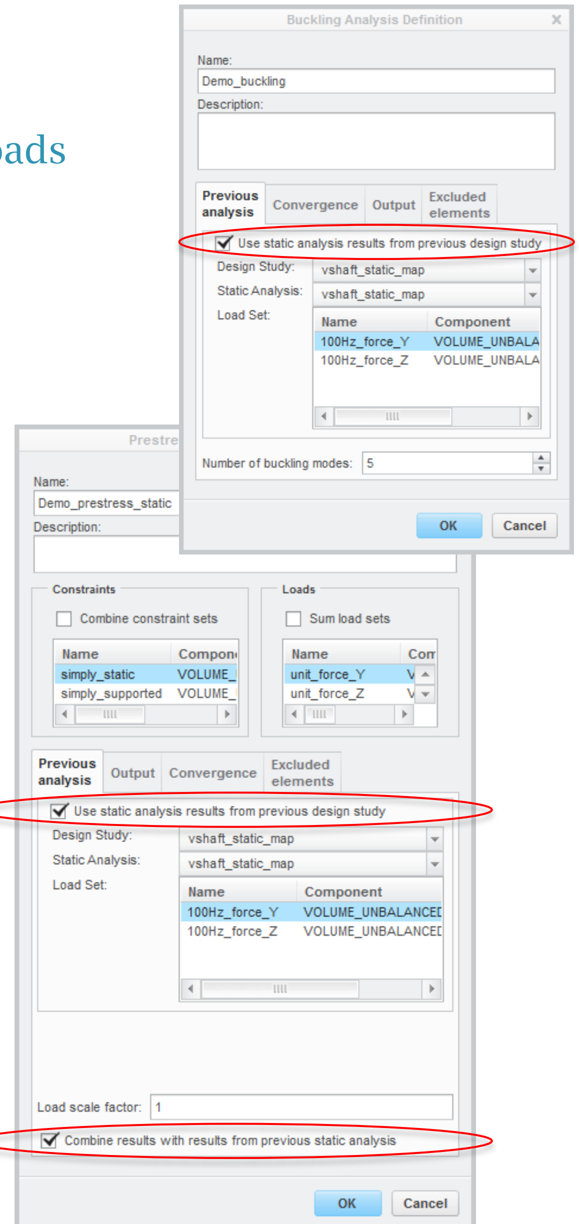
  - λ is also called the “buckling load factor” BLF
  - The linear buckling analysis does not take into account large displacements (it is assumed that the geometry is not significantly changing under load), so no tangential stiffness matrix $K_T = K + K_\sigma + K_L$ with K_L as stiffness matrix for large displacements is taken into account [5]

- Static analysis with prestress:

  $$([K]+[K_\sigma(\{F_P\})])\{x\}=\{F\}$$

  Like the modal analysis with prestress, this analysis takes into account weakening or stiffening effects from preloads!

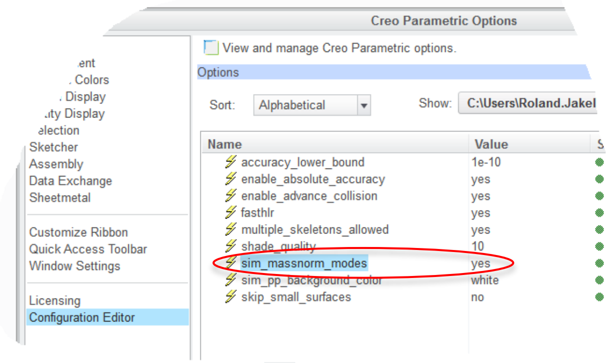
  - Both preload analyses may fail with the misleading error message “insufficiently constrained”, if the applied preloads are above the critical buckling load!
  - Static analysis with prestress outputs raw stresses and no superconverged stresses [9], so unlike in all other analyses in the postprocessor you can smooth these stresses (unsmoothed raw stress output allows to check for meshing quality)



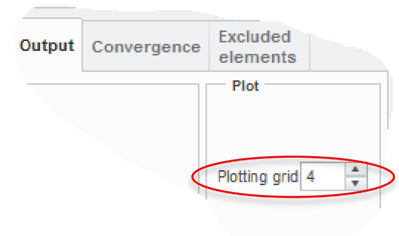
altran

2. Modal analysis

2.3 Hints for application



- Mass normalization can be requested via config.pro–option “sim_massnorm_modes” or with the engine command line option “–massnorm”



- With the “plotting grid” setting the RAM and hard disk resource consumptions can be highly influenced: E.g. for volume structures with no interest in a detailed stress computation in a later dynamic analysis, a plotting grid of 2 is sufficient!
(for beam models, a high plotting grid up to 10 should be used!)
- Also note requesting modal stress in the modal analysis needs a lot of RAM, so huge system models running into memory limits in the modal analysis may successfully run with deactivating the modal stress request
- If a static analysis fails with the error message “insufficiently constrained”, the rigid mode search in a constrained modal analysis can be successfully used for detecting the under constrained part or degree of freedom of the model (pretty useful for big system models with many parts/subassemblies).
Remark: If the modal analysis fails with the same error message even though, the reason is usually a free rotating point of the structure (e.g. a spring end point)!
- A modal analysis may also be used for checking mechanism modes, so for assuring a correct force flow in a static analysis!
- Measurement points should be defined before meshing (e.g. use hard point Auto GEM control or directly define the measure before meshing of the modal analysis model) – otherwise, the mesh cannot be reused in the subsequent dynamic analysis and the modal analysis has to be performed again!

3. Dynamic Analysis

3.1 Classification of the supported dynamic analysis types

- Depending on the forcing function {F} on the right side of the differential equation (DEQ), four different linear dynamic analysis types are supported in Creo Simulate:

	Periodic system (steady state)	Transient system
Exact excitation function (deter–ministic)	**Dynamic frequency analysis:** Only harmonic excitation $F = \hat{F}\cos(\omega t + \varphi)$ – neglecting transient effects – is present; the force is applied as function of frequency ($f = \frac{\omega}{2\pi}$) and all results are evaluated in the frequency domain	**Dynamic time analysis:** *) The excitation function is accurately known and applied as function of time; the results are evaluated in the time domain
Non–deter–ministic excitation	**Random response analysis:** The forcing function is obtained by statistical means (usually with help of measurements) and applied typically as acceleration density function (=acceleration density vs. frequency)	**Dynamic shock analysis:** **) The excitation force is typically a short random excitation (e.g. earthquake, pyrotechnic shock), for which an SRS (shock response spectrum) has to be computed as input for the FEM analysis. The analysis just offers an image of the "worst case" state with low computational effort!

*) Of course, in a dynamic time analysis also a harmonic excitation function can be applied, but unlike in dynamic frequency analysis, which just regards the particular solution of the DEQ, also the transient state will then be computed (homogeneous solution) before the steady state is reached

**) Strictly speaking, a dynamic shock analysis can be performed for any type of excitation for which an SRS can be obtained (e.g. also deterministic functions like half sine shocks, impulse functions, even harmonic excitation), but often it is performed for the mentioned transient examples

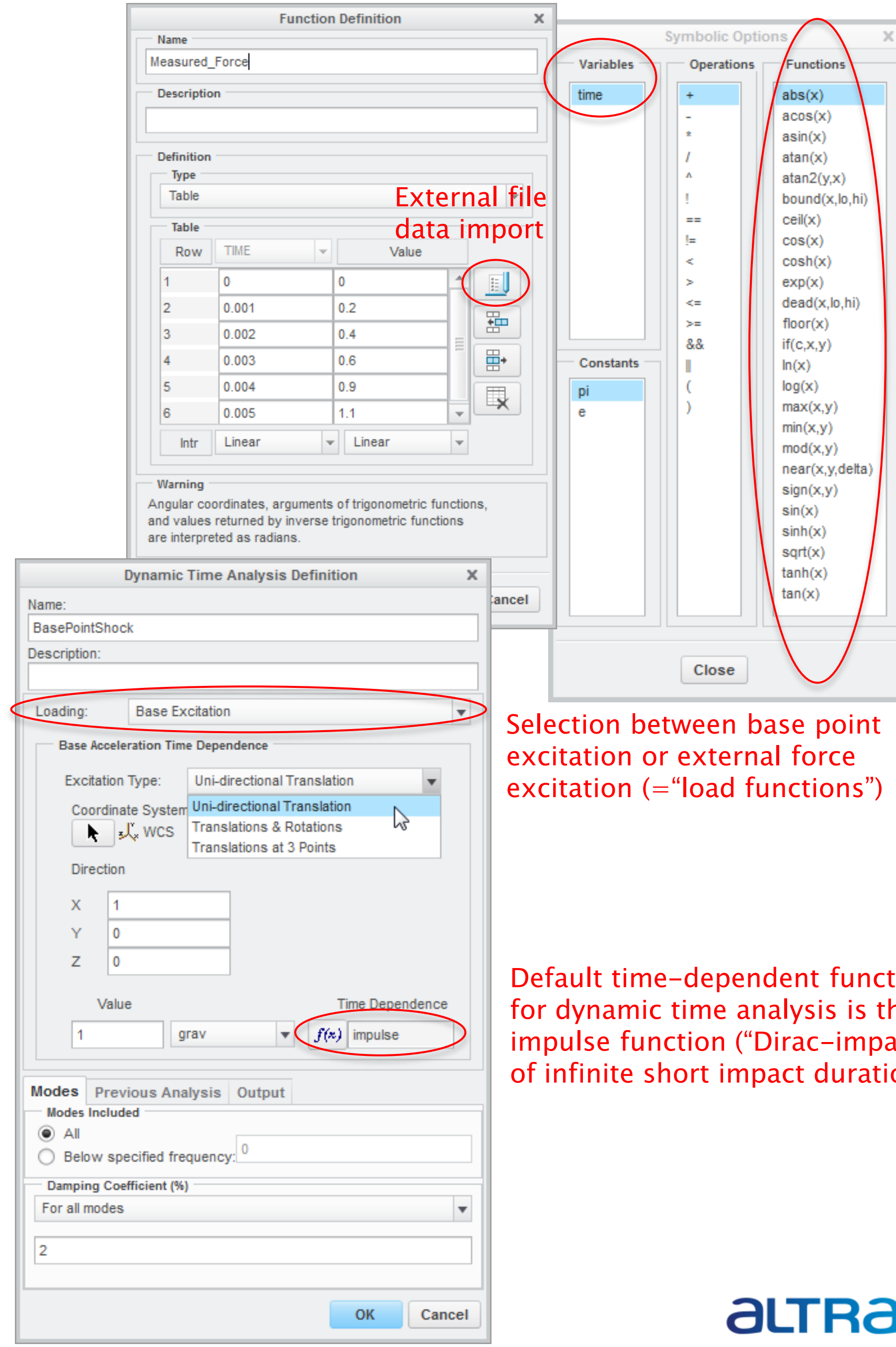
altran

3. Dynamic Analysis

3.2 Dynamic time analysis

3.2.1 Introduction

- The dynamic time analysis is the most universal and most simple to understand dynamic analysis in Creo Simulate, but also the computationally most intensive
- If other dynamic analyses fail for certain coding limitations, it is a good idea to try this analysis type!
- Nearly any arbitrary force vs. time function can be applied, either
  - ➢ in form of any analytic function like e.g. $F = \hat{F}\cos(\omega t + \varphi)$ (for all programmable functions, see right)
  - ➢ or as a tabular function (e.g. as file input from a given force-vs.-time measurement)
- Alternatively, any base point (interface) acceleration can be defined in a similar way



Selection between base point excitation or external force excitation (="load functions")

Default time-dependent function for dynamic time analysis is the impulse function ("Dirac-impact" of infinite short impact duration)

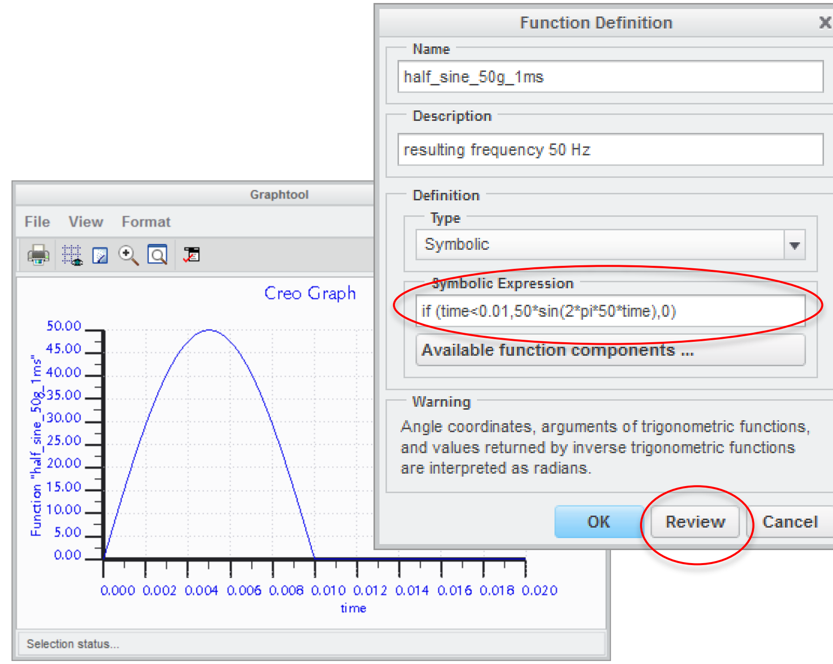
3. Dynamic Analysis

3.2 Dynamic time analysis

3.2.2 Examples

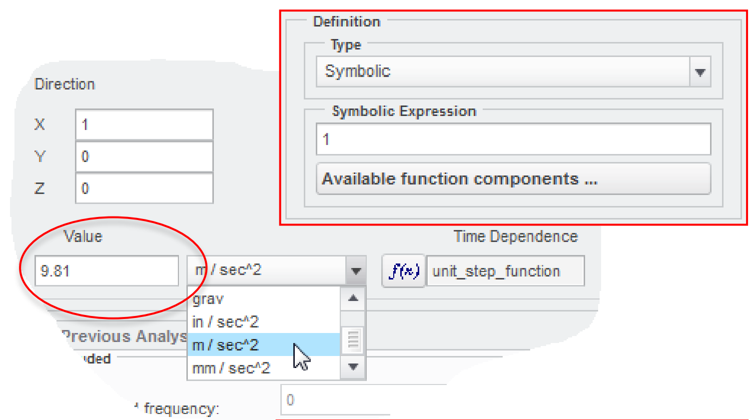
<u>A half sine base point acceleration shock</u>

- We want to apply a half sine wave shock with 50 g peak acceleration and 10 ms duration ($f = \frac{1}{T} = \frac{1}{20\ ms} = 50\ Hz$)
- The required analytic expression takes advantage of the "if"-function shown right
- Correct coding can be checked by graphical visualization of the function



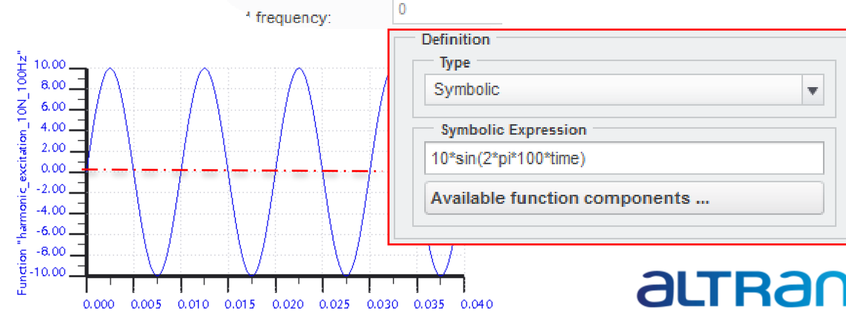
<u>A base point step (jump) function</u>

- We want to apply a 1 g step function for time t=0
- Very simple to code like shown right
- Always note the constant scale factor and units in the analysis definition dialogue!



<u>A harmonic sine excitation</u>

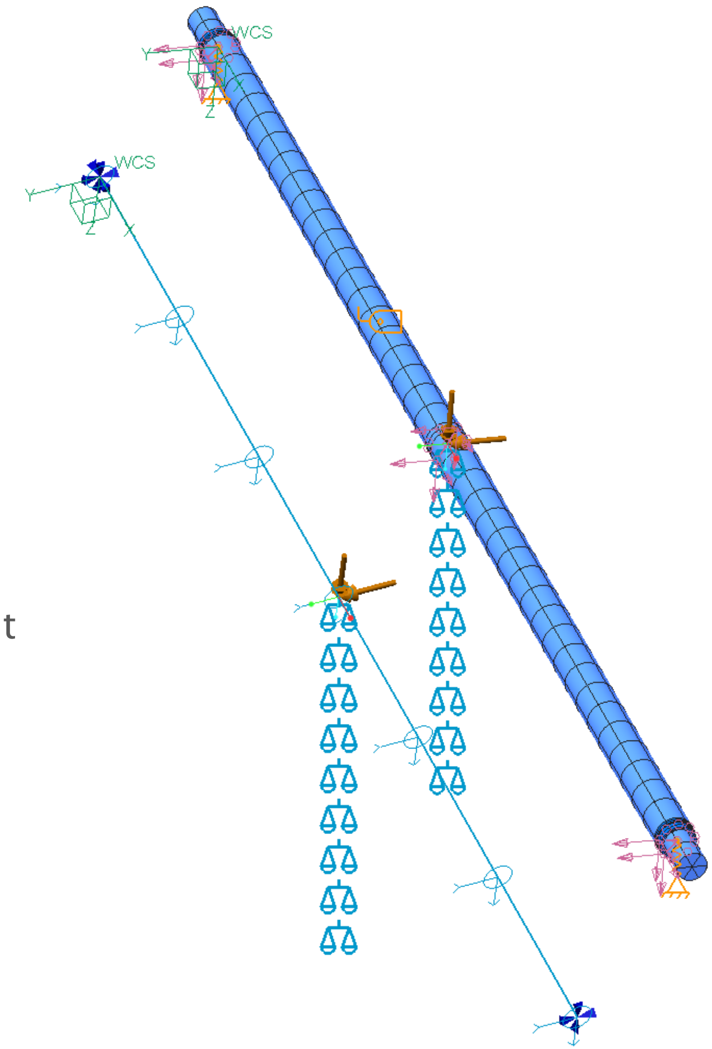
- We want to apply a force of 10 N with f=100 Hz, starting with 0 N at t=0



3. Dynamic Analysis

3.2 Dynamic time analysis

3.2.2 Examples



A long, slim drive shaft with unbalance operated at its resonance frequency (100.3 Hz)

- Example model of chapter 2.1.2
- Rotational speed 6000 rpm (100 Hz)
- Modal damping 2 %
- Static unbalance mass u=1 gram at unbalance radius 6.5 mm in the middle of the shaft (no dynamic unbalance assumed in the example, but this would be simple to simulate, too)

Questions of interest:

- How big is the displacement (shaft bending) under this operating condition?
- How big is the max. shaft acceleration due to vibration created by the unbalance?
- How long does it take until the shaft swings up?
- Are the stresses in the shaft still low enough so that it can be safely operated even though it is running in resonance?
- Is the foreseen balancing quality *G* sufficient, e.g. to obtain the required swinging velocity?

$$G = e\omega = \frac{U}{m}\omega = \frac{ur}{m}\omega = balancing\ quality \left[\frac{mm}{s}\right]$$

U = unbalance $u \cdot r$ [g mm]
u = unbalance mass
r = unbalance radius of u
m = total mass of the rotor
e = excentricity of m
ω = angular velocity

3. Dynamic Analysis

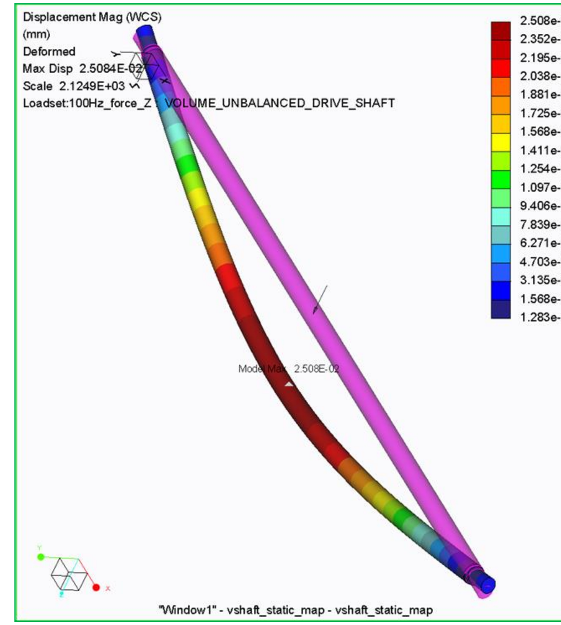
3.2 Dynamic time analysis

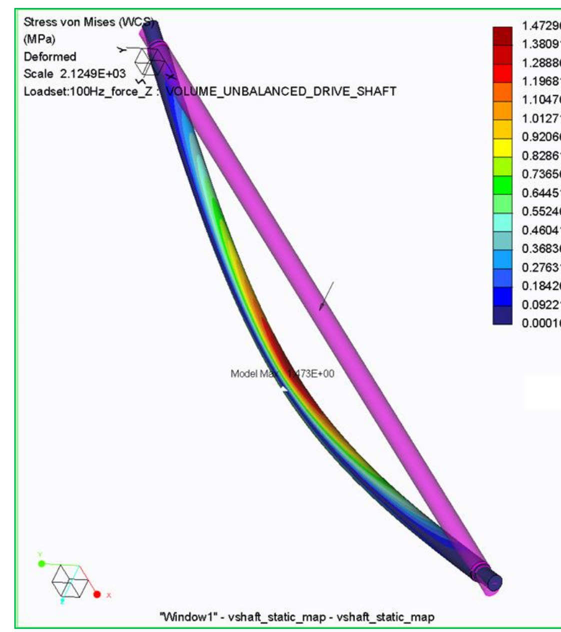
3.2.2 Examples

<u>Before we run the dynamic time analysis, let's see what we can do with some simple estimations and the information we already have:</u>

- Unbalance force: $F_u = u \cdot r \cdot \omega^2 = 2.566\ \mathrm{N}$
- Static deformation under this force: 0.025 mm (obtained in a simple, linear static analysis shown right or with help of a formulary)
- Magnification factor for the given damping: $Q = 1/2\beta = 25$
- Therefore the expected deformation at 6000 rpm (very close to the resonance peak) will be: $\approx$ 0.625 mm
- From the modal analysis of chapter 2.1.1 we know that the modal von Mises stress is 3019 MPa for a mass normalized displacement of 61.95 mm
- Since with the central unbalance force at 100 Hz we predominantly excite just the first mode, we can estimate the real stress by scaling the modal stress from mode 1 to the estimated displacement of 0.625 mm by rule of three: We obtain approx. 30 MPa in resonance from this!

$$\frac{x\ \ MPa}{3019\ MPa} = \frac{0.625\ mm}{61.95\ mm} \Rightarrow x \approx 30\ MPa$$





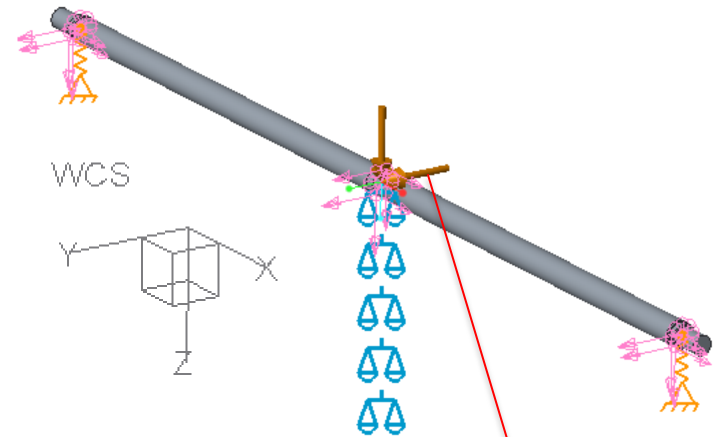
3. Dynamic Analysis

3.2 Dynamic time analysis

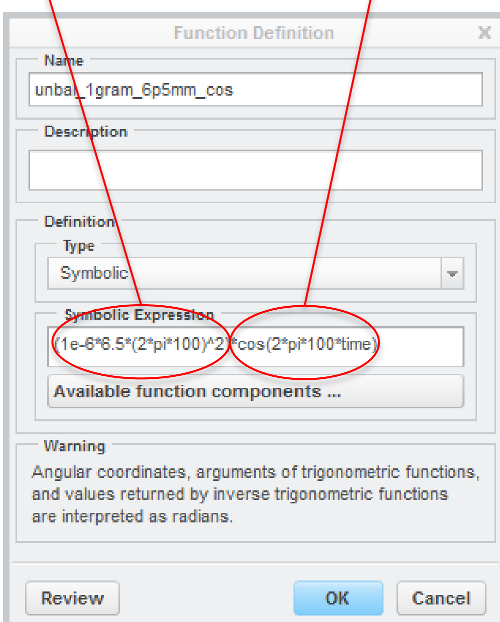
3.2.2 Examples

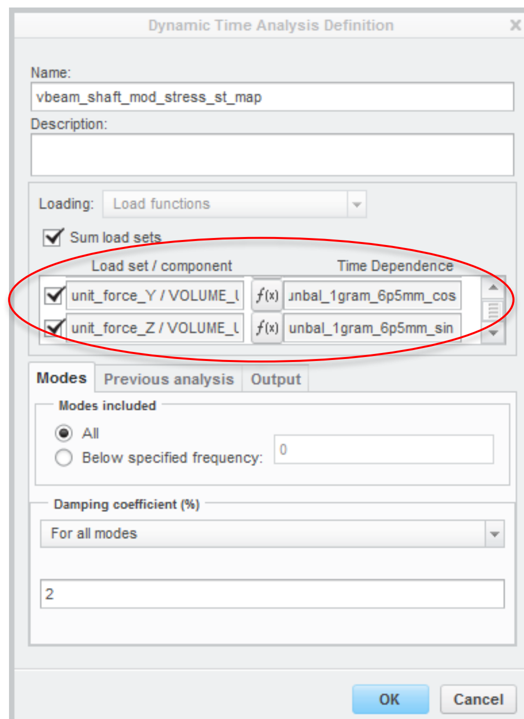
<u>Setting up the dynamic time analysis</u>

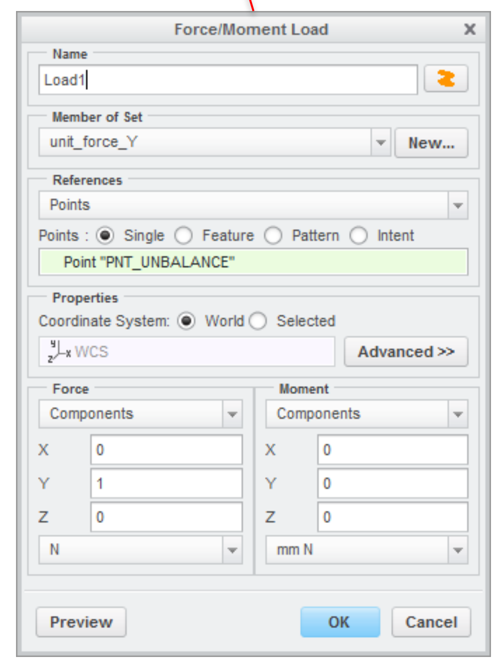
- The 100 Hz rotating unbalance is defined by using a sine and a cosine forcing function as time dependent functions for the two unit forces applied under 90 °, respectively



$F_u = u \cdot r \cdot \omega^2$ Cosine function for 100 Hz







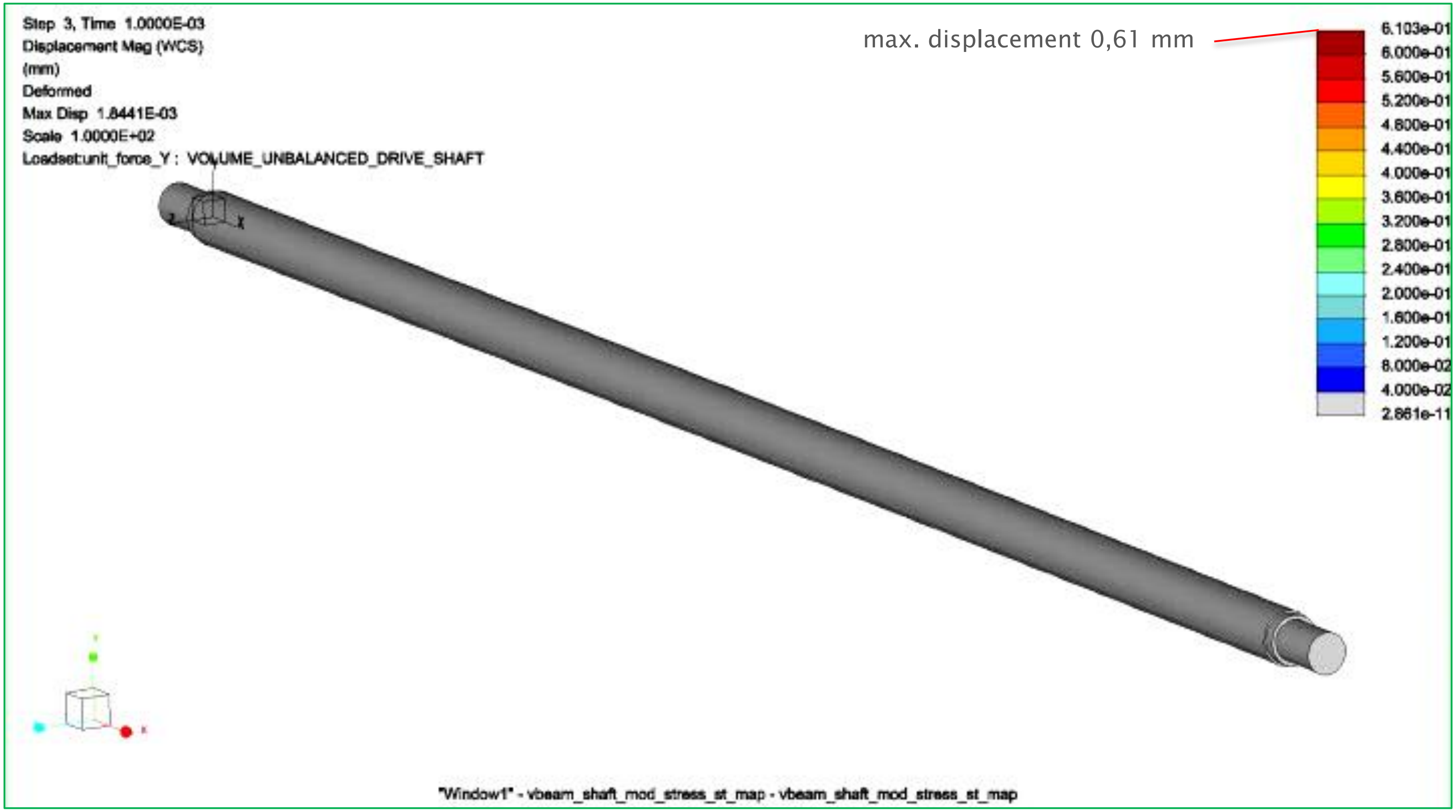
altran

3. Dynamic Analysis

3.2 Dynamic time analysis

3.2.2 Examples

Displacement animation [mm] of the swing up process (scale 100:1, 0.5 s duration)

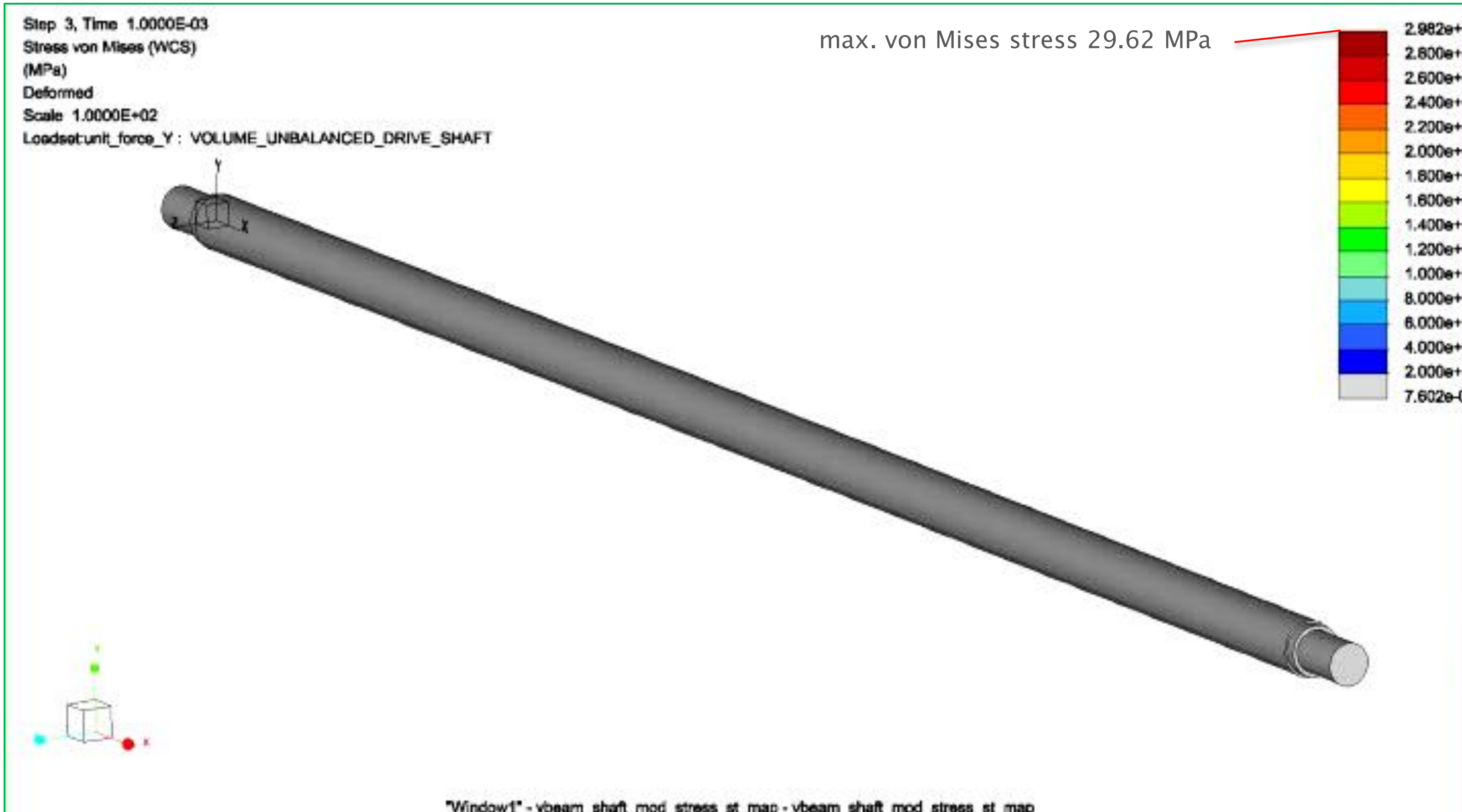


3. Dynamic Analysis

3.2 Dynamic time analysis

3.2.2 Examples

Von Mises stress animation [MPa] of the swing up process (scale 100:1, 0.5 s duration)

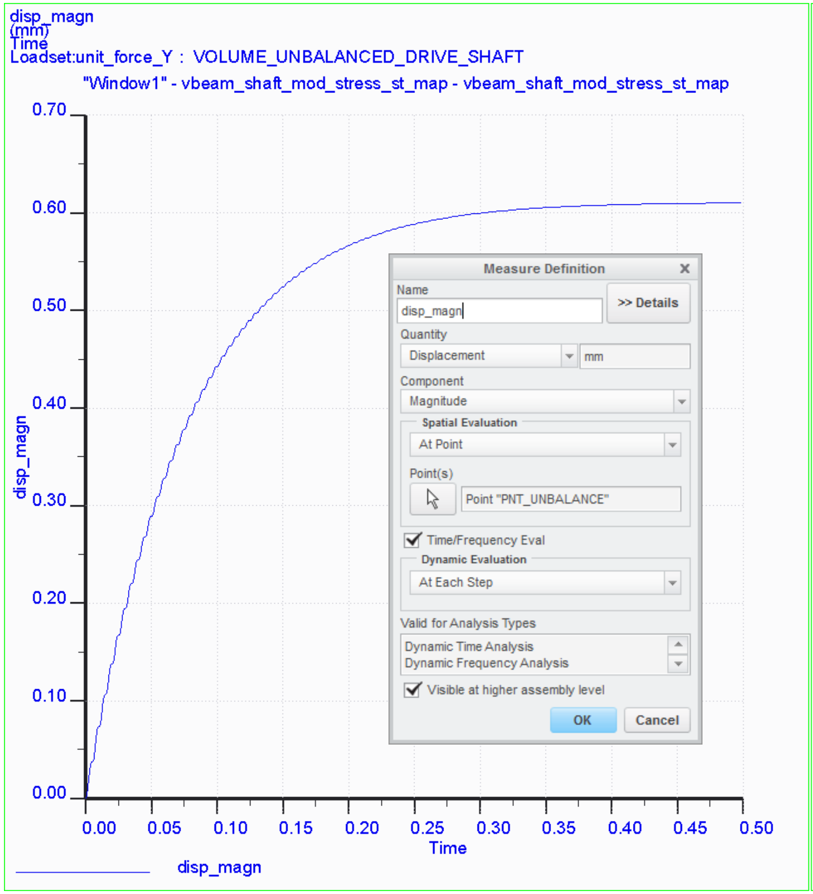


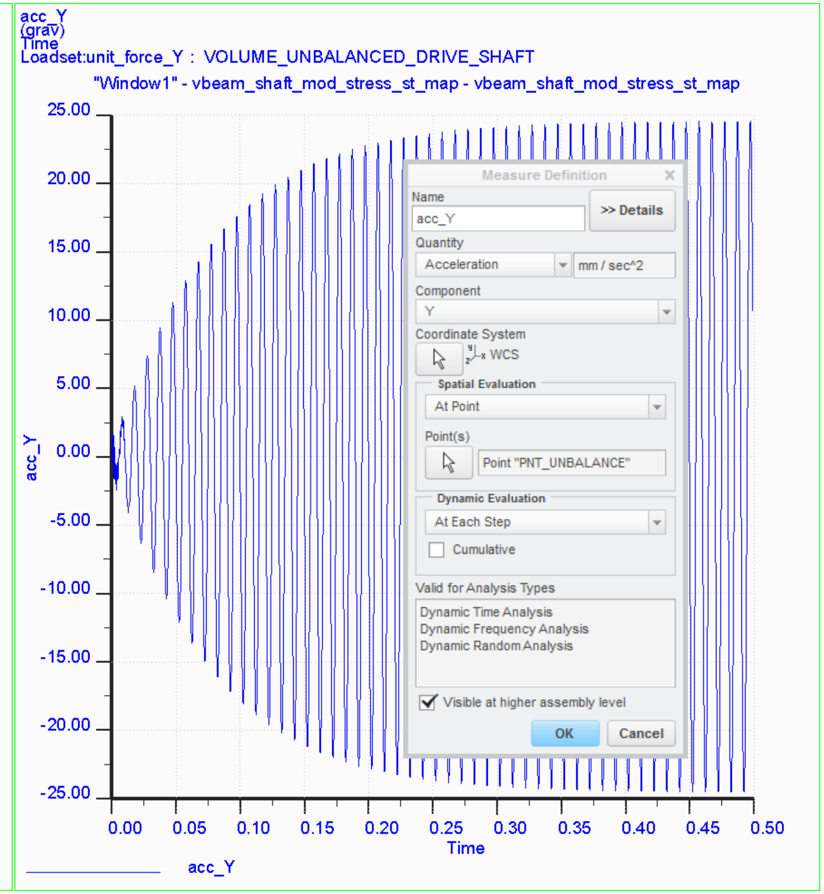
3. Dynamic Analysis

3.2 Dynamic time analysis

3.2.2 Examples

Measures for displacement magnitude [mm] (left) and Y-acceleration [g] (right)





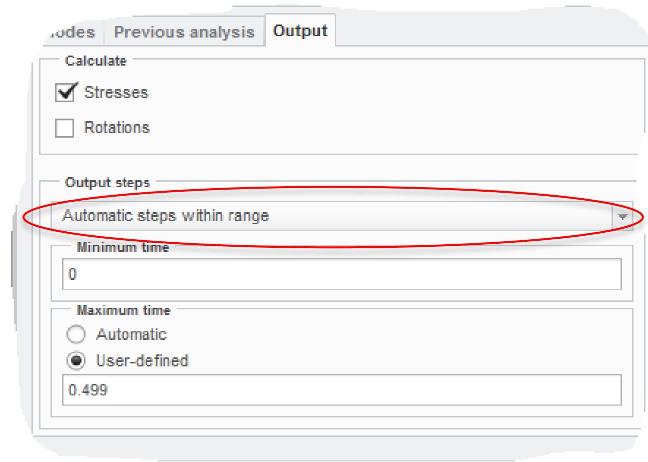
3. Dynamic Analysis

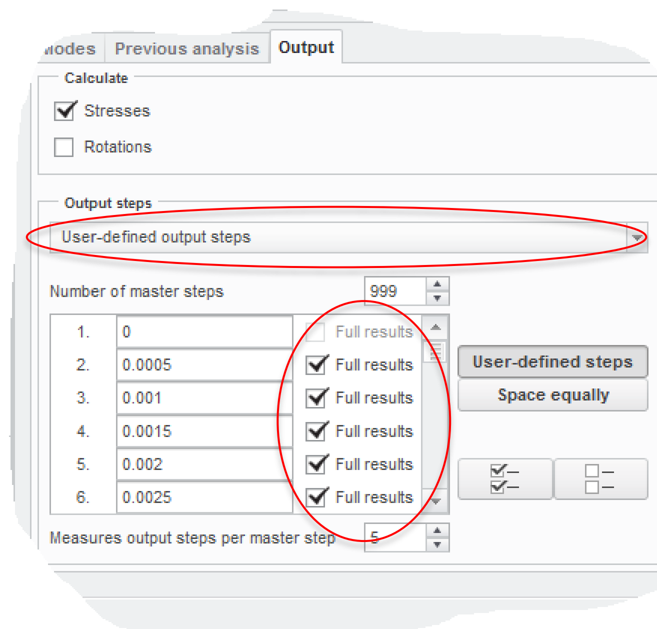
3.2 Dynamic time analysis

3.2.2 Examples

<u>Using the dynamic analysis output functionality</u>

- Dynamic analysis can typically be computed with automatic output or user defined output steps
    - ➢ For the first case, only measures are output and the code automatically assures a suitable (time or frequency) stepping
    - ➢ Since there are no system default measures for dynamic analysis, the user always has to define measures before a dynamic analysis with automatic steps (better already before the modal analysis)
    - ➢ For user-defined output, the user can select stepping and request full results (=colorful PP images) for all or only those steps of certain interest
- Often it is a good idea to run the first analysis with automatic output (and meaningful measures!) and then run a second analysis with user defined output for further evaluation
- We will subsequently explain this at the shaft example



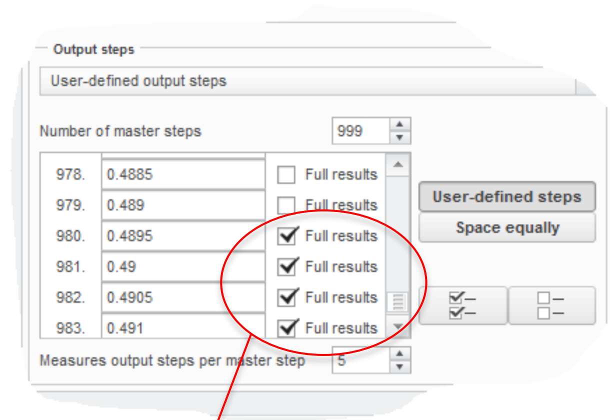


altran

3. Dynamic Analysis

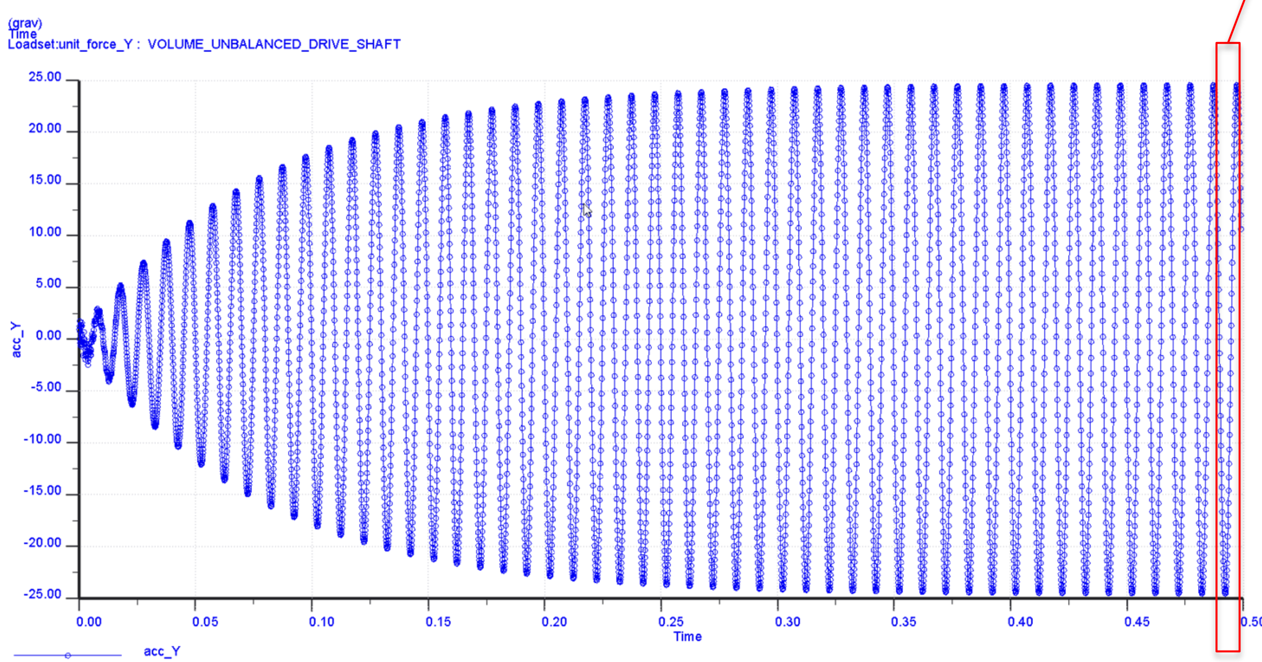
3.2 Dynamic time analysis

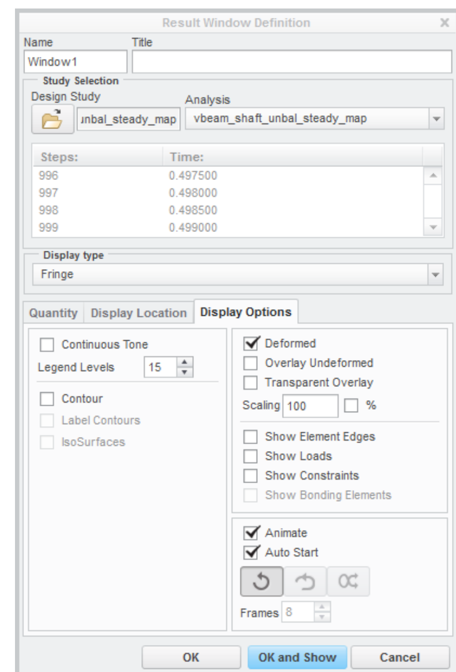
3.2.2 Examples



Animating only the particular solution of the DEQ for the unbalanced shaft in a dynamic time analysis

- The measurement output for e.g. the Y–acceleration shows that the system has practically reached its steady state at the end of the computed time span of 0.5 s
  - ➢ So we will request full output now just for the last period of the analysis, this can then be repeatedly displayed in the postprocessor





3. Dynamic Analysis

3.2 Dynamic time analysis

3.2.2 Examples

Von Mises stress animation [MPa] of the steady state (scale 100:1, one period of 0.01 s)

3. Dynamic Analysis

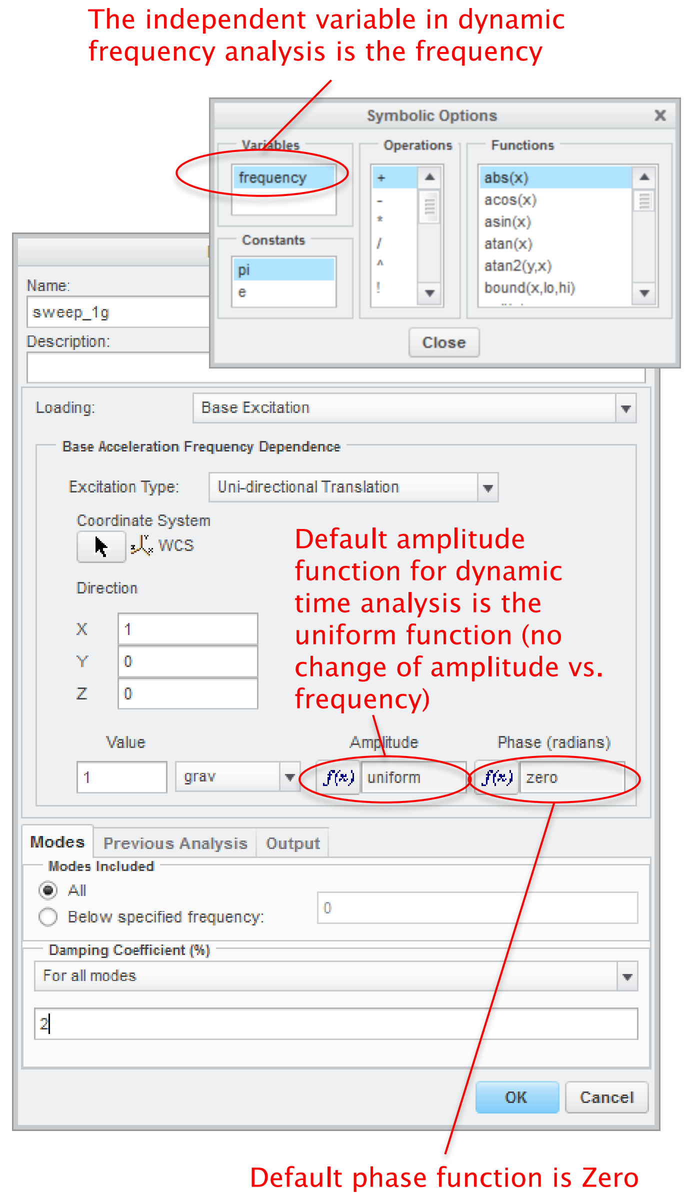
3.3 Dynamic frequency analysis

3.3.1 Introduction

- The dynamic frequency analysis computes the structure's responses to pure harmonic excitation (cosine/sine function with one frequency at the same time) and disregards any homogeneous solution (transient oscillation); just the steady state (particular solution) is taken into account
- So for a given excitation frequency $f = \omega/2\pi$, the exciting force (if force excitation is present) has the following form:

$$F(t) = F_{\max} \cos(\omega t + \varphi)$$

- The amplitude F_{max} and phase φ of the excitation vs. frequency function can be input by analytic functions or as tabular input (in analogy to the tools used in dynamic time analysis)
- Like in dynamic time analysis, in addition to load functions also base excitation is supported
- A typical application is a sine sweep test with a very low sweep rate (strictly speaking: an infinitely low sweep rate!)



altran

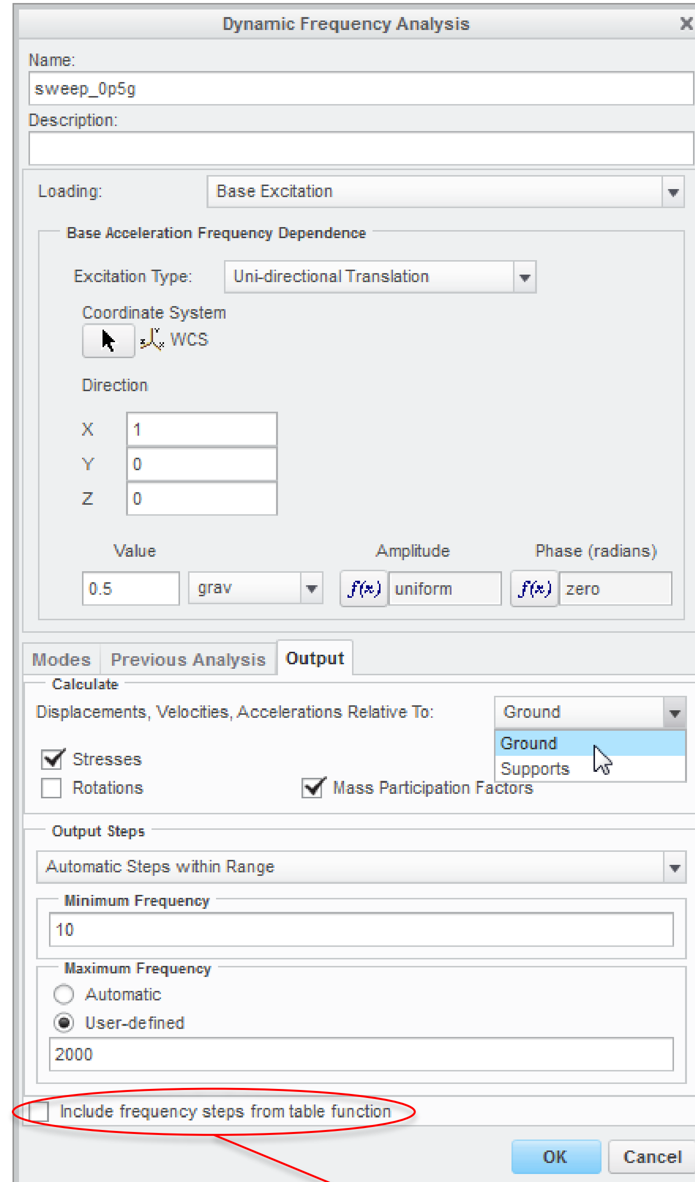
3. Dynamic Analysis

3.3 Dynamic frequency analysis

3.3.2 Examples

<u>0.5 g constant base point acceleration in X between 10 and 2000 Hz (sine sweep)</u>

- Very simple to define, see right
- Note the possibility to request output relative to “ground” or “supports”-option for base point excitation!



Important to activate for non-constant tabular input only, otherwise the automatic frequency stepping generator misses those frequencies!

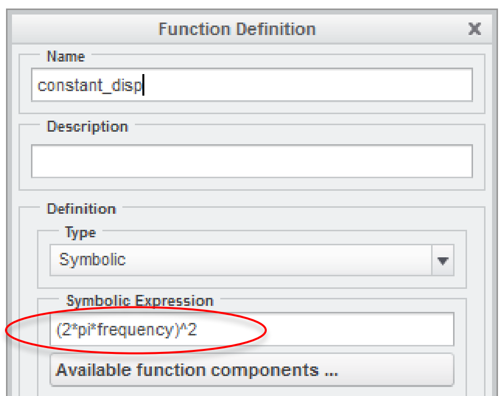
<u>10 mm frequency independent (constant) base point displacement amplitude</u>

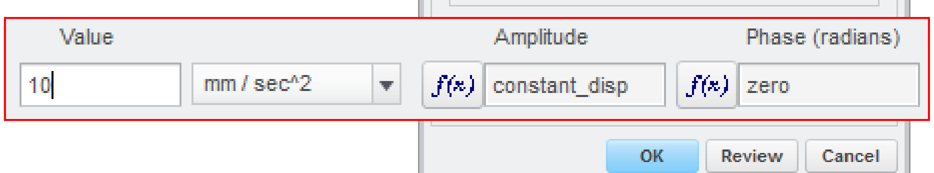
- We have to define a frequency dependent acceleration function for this as shown below

$$x = x_{\max} \sin(\omega t)$$

$$\dot{x} = x_{\max} \omega \cos(\omega t)$$

$$\ddot{x} = x_{\max} \omega^2 (-\sin(\omega t))$$



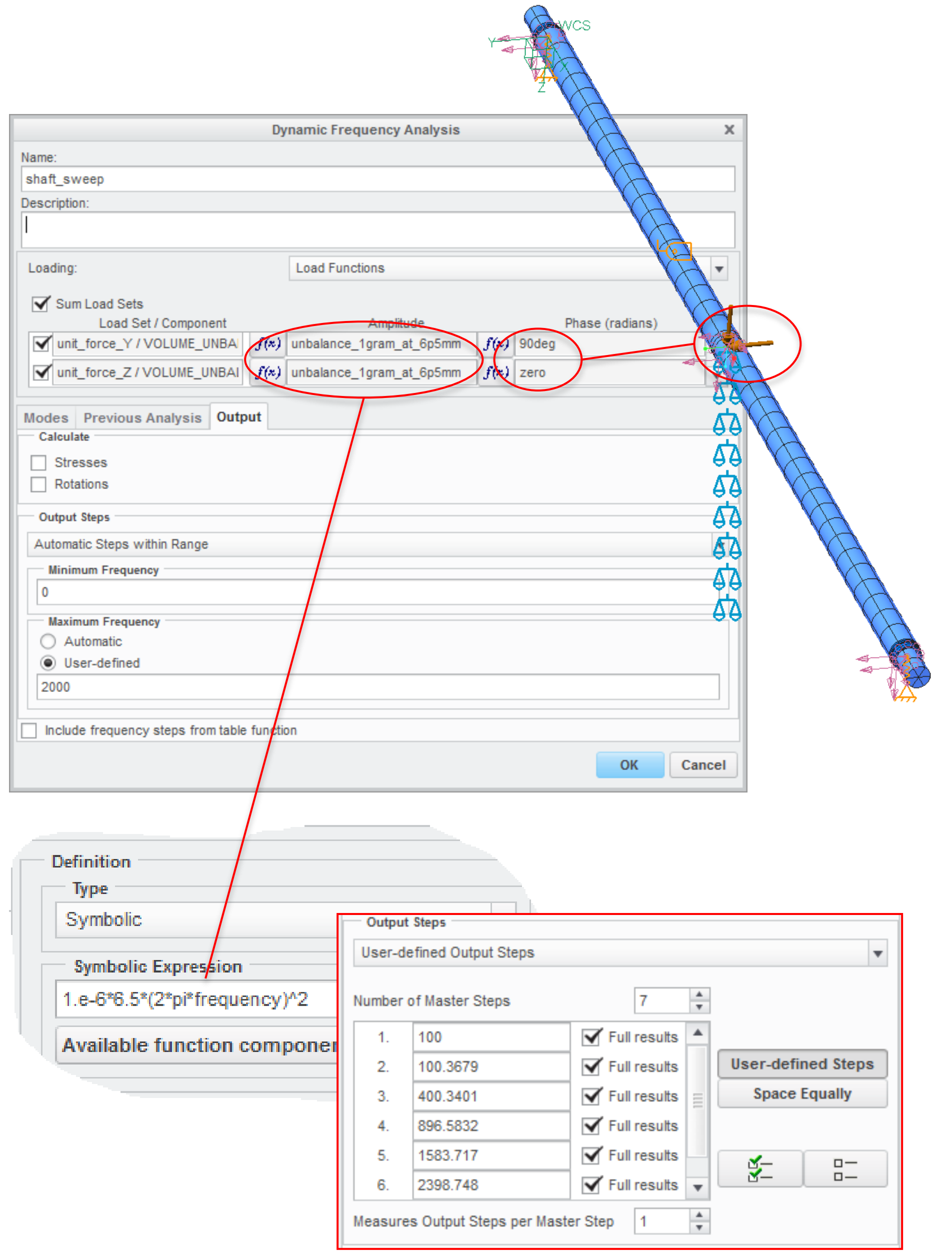


3. Dynamic Analysis

3.3 Dynamic frequency analysis
3.3.2 Examples

<u>A long, slim drive shaft with unbalance operated at its resonance frequency (100.3 Hz)</u>

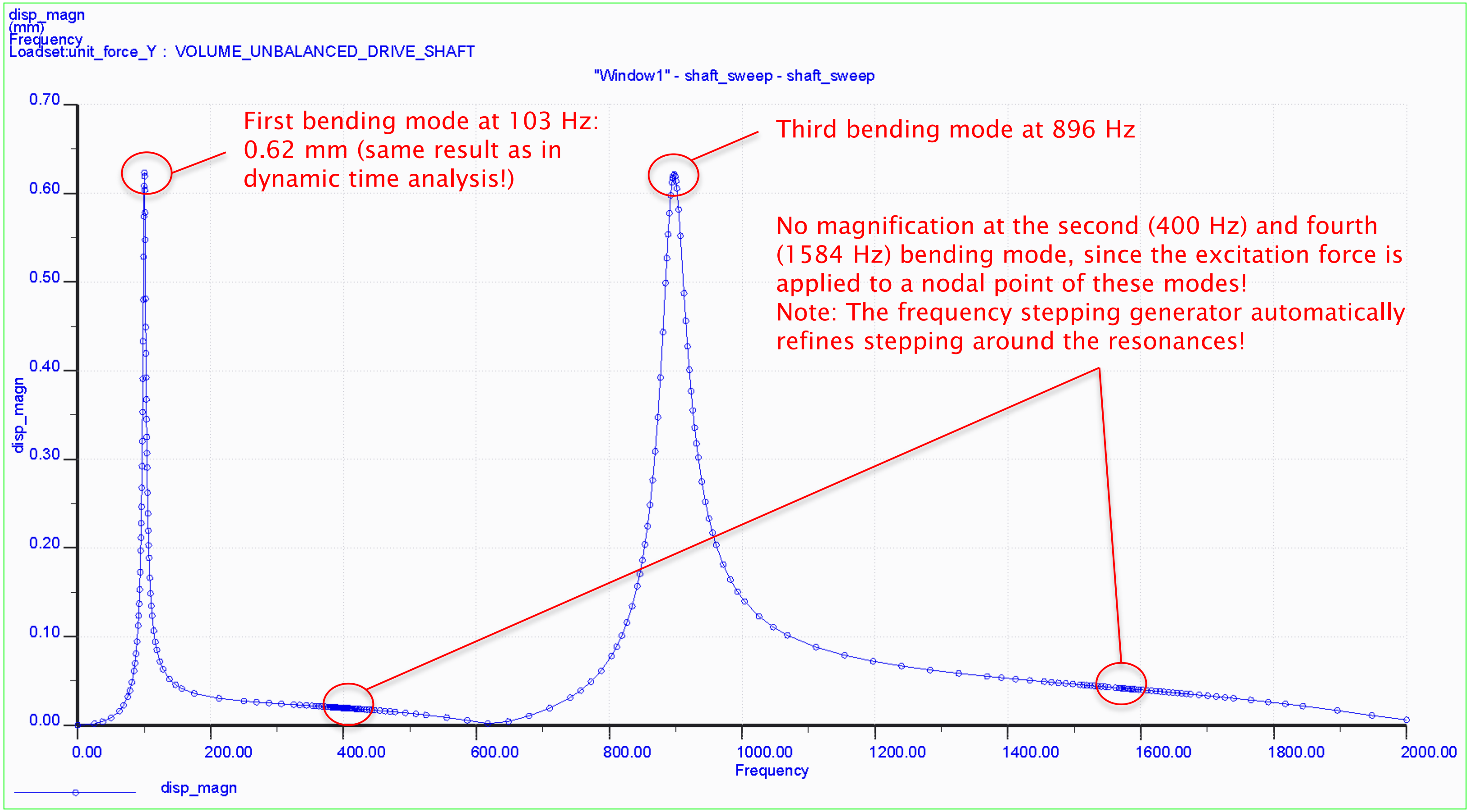
- Example model of chapters 2.1.2 and 3.2.2:
  - ➢ Nominal rotational speed 6000 rpm (100 Hz)
  - ➢ Modal damping 2 %
  - ➢ Static unbalance mass u=1 gram at an unbalance radius of 6.5 mm in the middle of the shaft
- In addition, we also want to know the response of the shaft vs. frequency between 0 and 2000 Hz
- Necessary analysis definition is shown on the right side
- Additional analysis with full output request at the fundamental frequencies for results animation within the postprocessor



altran

3.3 Dynamic frequency analysis
3.3.2 Examples

Frequency response curve for displacement magnitude of the central shaft point

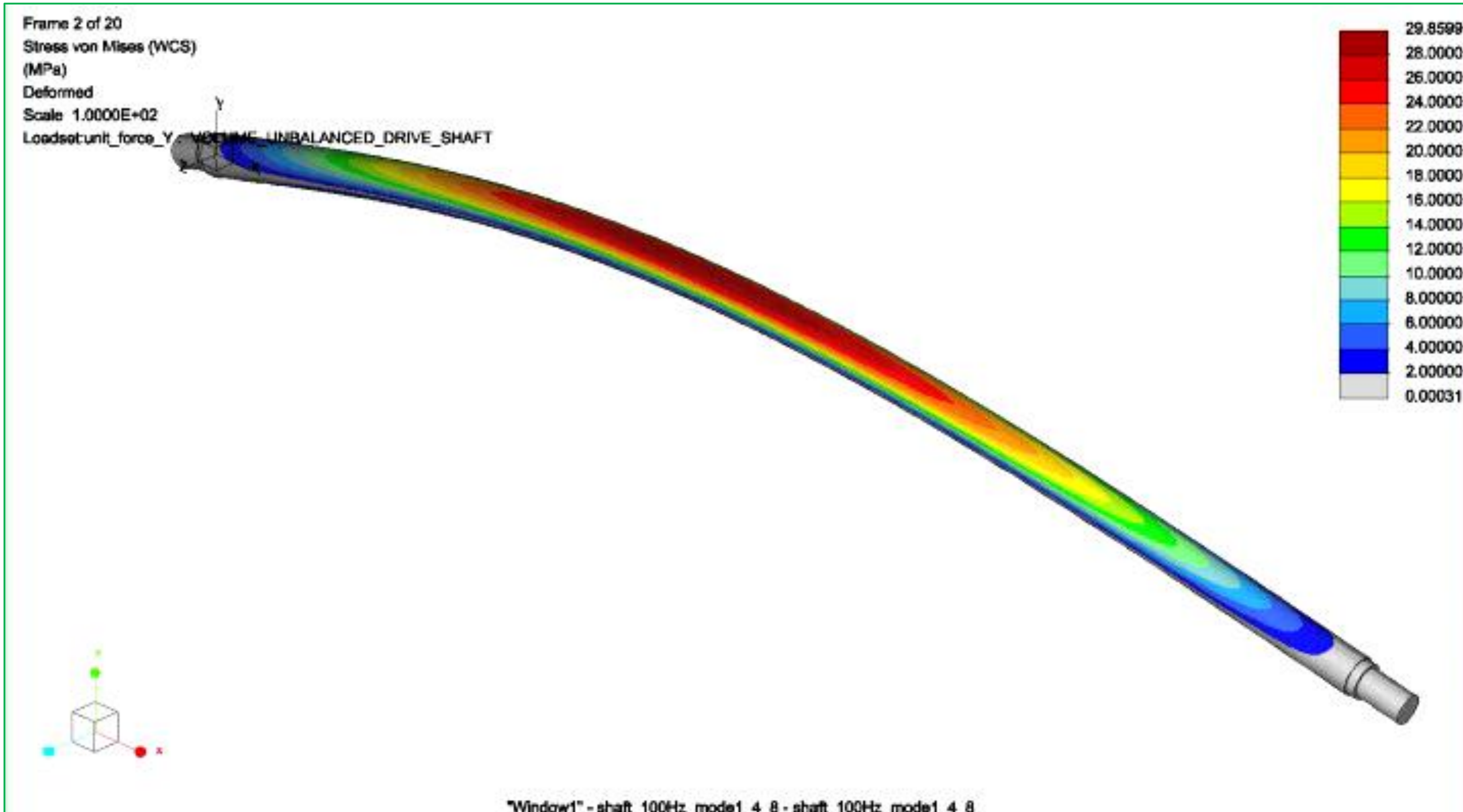


3. Dynamic Analysis

3.3 Dynamic frequency analysis

3.3.2 Examples

Von Mises stress animation [MPa] for 100 Hz (scale 100:1)



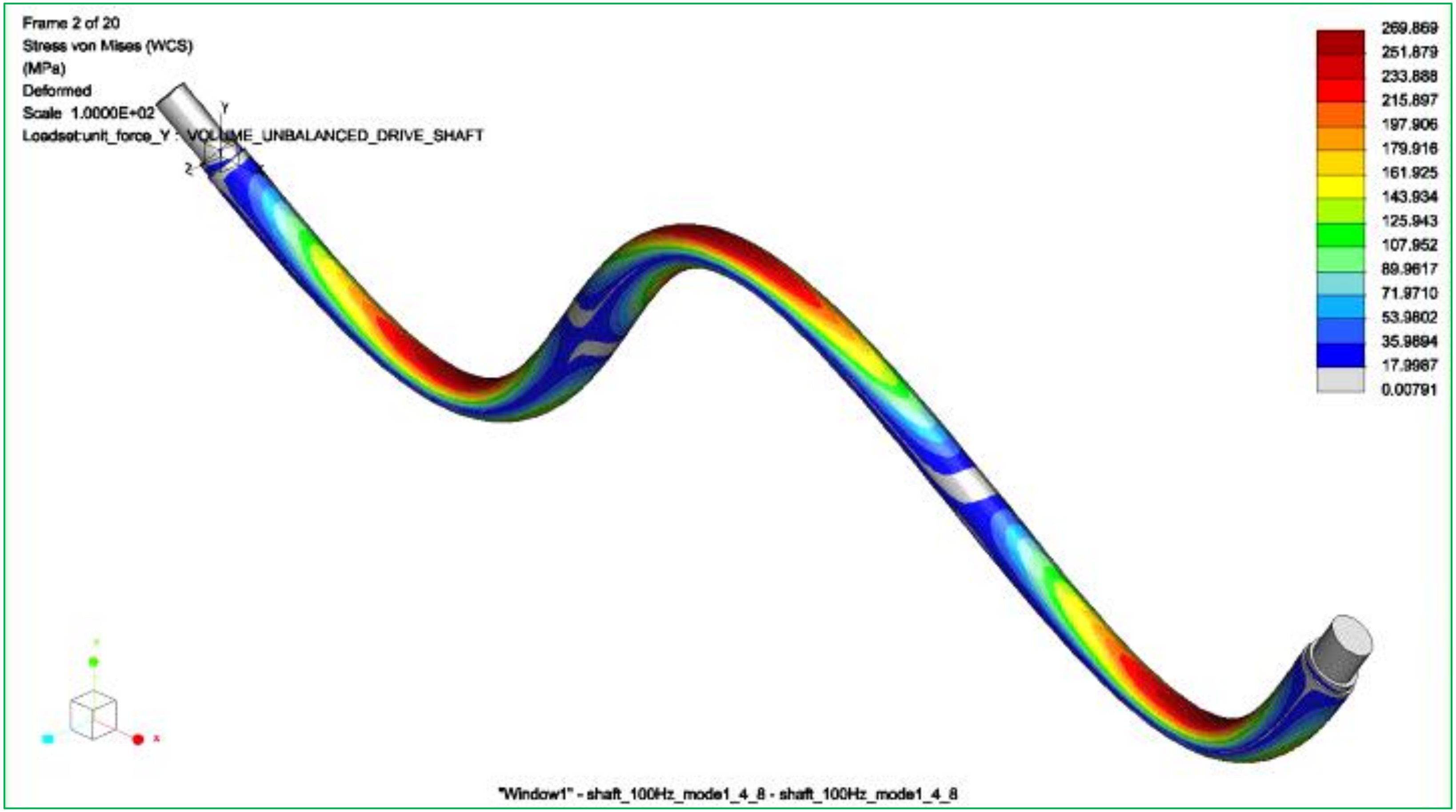
Comparison of elapsed times:
Dynamic frequency analysis requesting full output for 7 frequencies: 5.71 s
Dynamic time analysis with full output just for the steady state at 100 Hz: 224 s
Dynamic time analysis for the complete swinging up process at 100 Hz: 664 s

3. Dynamic Analysis

3.3 Dynamic frequency analysis

3.3.2 Examples

Von Mises stress animation [MPa] for 896 Hz (scale 100:1)

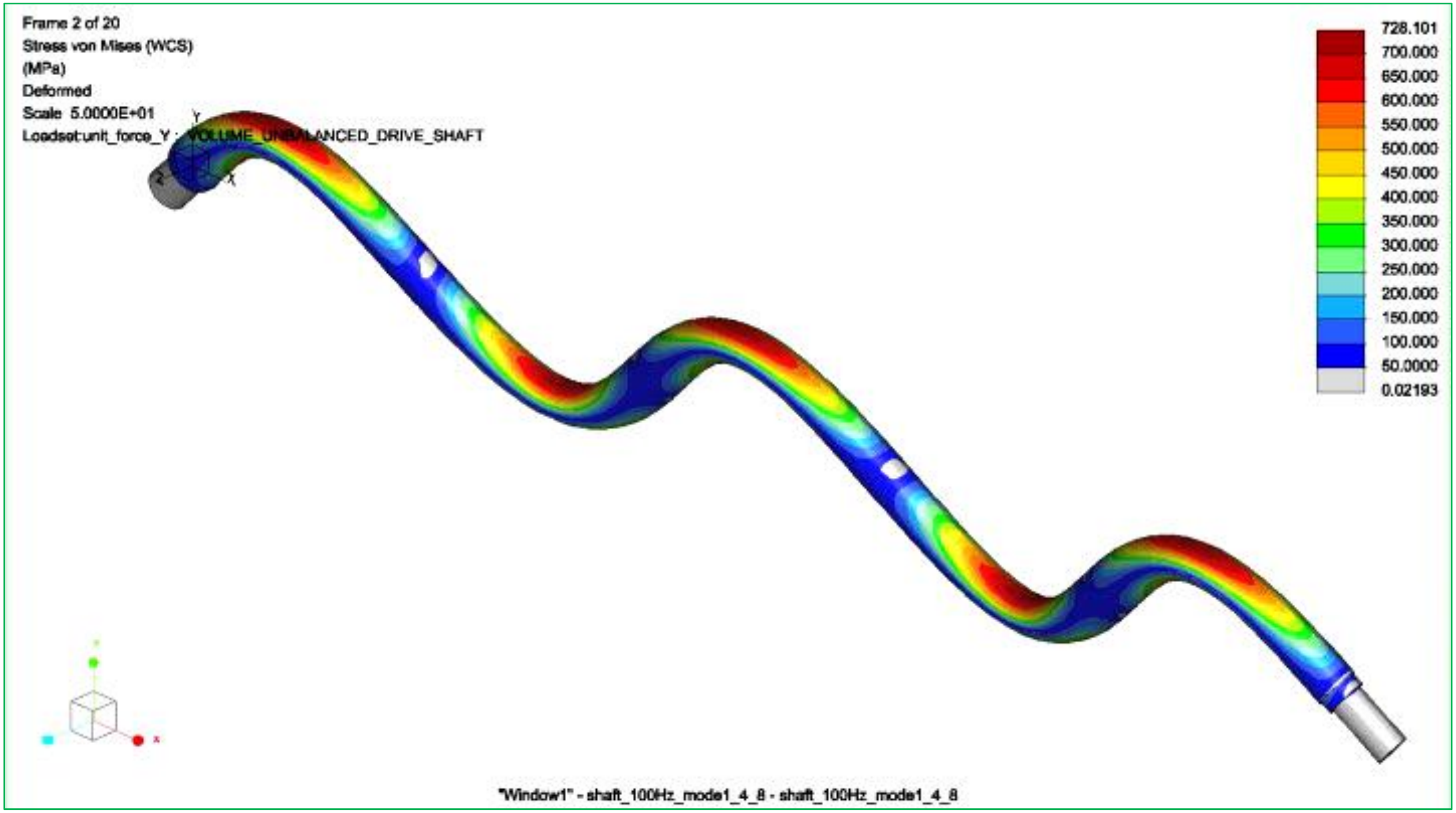


3. Dynamic Analysis

3.3 Dynamic frequency analysis

3.3.2 Examples

Von Mises stress animation [MPa] for 2454 Hz (scale 50:1)



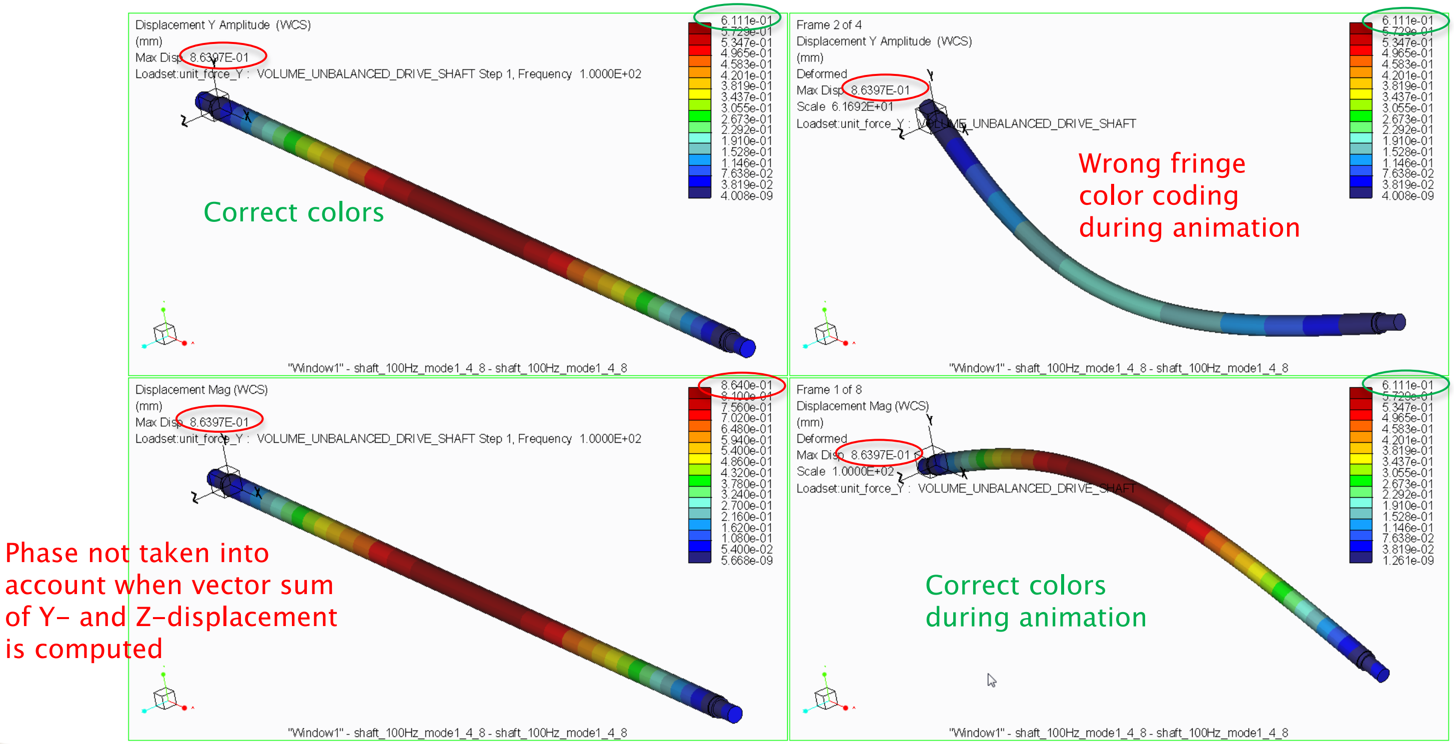
3. Dynamic Analysis

3.3 Dynamic frequency analysis

3.3.3 Remarks for application

Note SPR 2875703 when evaluating dynamic frequency results in the postprocessor:

- Unfortunately, it may happen that the phase result is not taken into account correctly in the postprocessor. In latest tests the following occurred:
  - Displacement, velocity and acceleration *magnitude* fringe plots just give correct results when animated, but wrong when not!
  - Displacement, velocity and acceleration *component* fringe plots just give correct results when *not* animated (but never the “Max. Disp.” value display as shown below)
- Be also careful when evaluating other results (phases…)

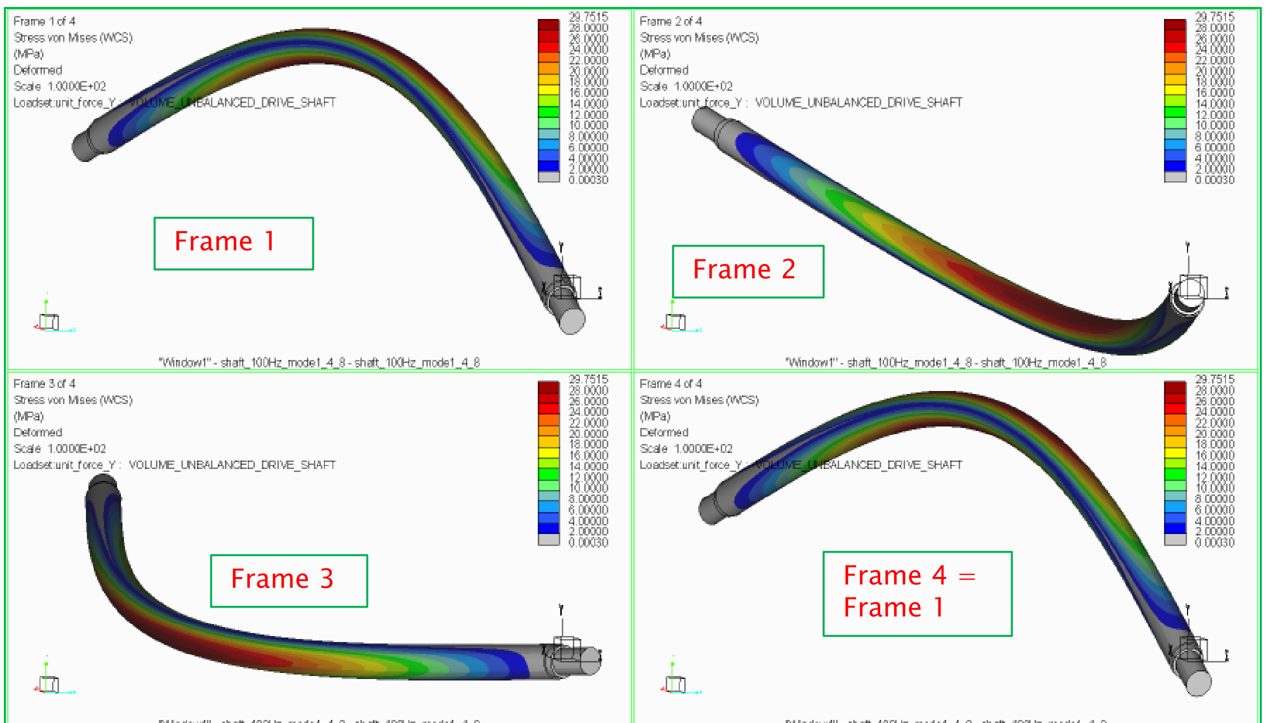


3. Dynamic Analysis

3.3 Dynamic frequency analysis
3.3.3 Remarks for application

Note SPR 2875703 when evaluating dynamic frequency results in the postprocessor:

- Furthermore, to obtain a "smooth" animation, use as many frames n as possible (>20), since the PP erroneously divides the deformed shape by n−1 (=first and last frame have identical shape; this error can be reduced by using many frames!)
- In general, do not activate *deformed shape* without activating *animation* for dynamic frequency analysis results evaluation (this results in a meaningless shape just taking into account the amplitude maximum in positive coordinate direction, but no phase)

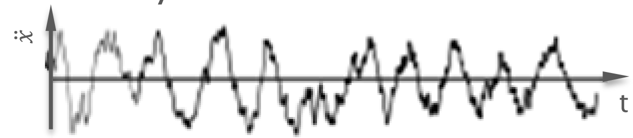


3. Dynamic Analysis

3.4 Random response analysis

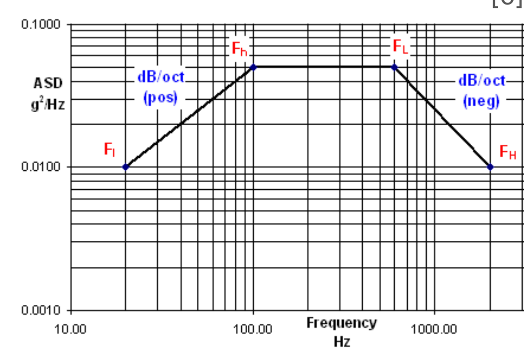
3.4.1 Introduction

- Until now, we have just looked at excitations where the excitation of the structure can be predicted for any time t with help of a deterministic mathematical function
- In reality, we have also excitations for which an accurate input prediction in a deterministic sense cannot be done. Examples for this may be
  - Jet or rocket engine noise
  - Turbulent fluid flow
  - Ground acceleration during an earthquake



- Such excitations are described with statistical methods, and usually a lot of measurements of the excitation in the time domain must be done, evaluated and edited in order to obtain a reliable acceleration or force spectral density function
  - For more details, refer to suitable technical literature
- With this random response spectrum finally on hand, the code computes in this analysis type the answer of the mechanical structure when subjected to this excitation
- In probably >90% of the application cases this will be an acceleration spectral density function introduced into the structure's base points, exemplarily shown right (note also force excitation is supported in Simulate)
- In the following chapter we will therefore exemplarily describe how this works with help of a simple one-mass-resonator, for which an analytical solution exists

[6]



altran

3. Dynamic Analysis

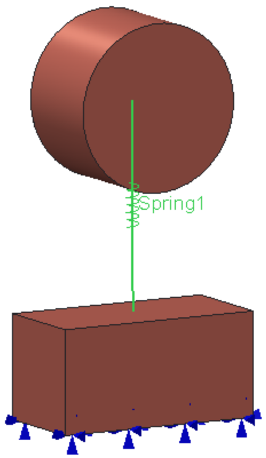
3.4 Random response analysis

3.4.2 Examples

Base point excitation of a one-mass-oscillator with white noise:

- Given is a white noise acceleration density spectrum with a constant acceleration density of W_{in}=0.2 g^2/Hz for 1-2000 Hz
- The effective input acceleration is the square root below the acceleration density curve, also called RMS- (root mean square) or 1σ-acceleration:

$$g_{in,RMS} = \sqrt{0.2(2000-1)} = 19.995 g_{RMS}$$



- With help of the Miles formula (valid for ideal white noise with an infinite frequency span from 0 Hz to ∞ Hz!)

$$g_{out,RMS} = \sqrt{\frac{\pi}{2} \cdot Q \cdot f_0 \cdot W_{in}}$$

we obtain for a resonator with Q=25 (β=2 %) and a fundamental frequency of 700 Hz the 1σ-output acceleration:

$$g_{out,RMS} = \sqrt{\frac{\pi}{2} \cdot 25 \cdot 700Hz \cdot 0.2 \frac{g^2}{Hz}} = 74.15 g_{RMS}$$

- If the momentary values of the output are Gauss distributed, the expected peak value is usually defined as the 3σ value, which means that only 0.3 % of the momentary values of the output acceleration are greater than 222,5 g_{RMS}, and that for 99,7 % of the time the acceleration is below!
- This 3σ-value is typically used for estimation of the max. resonator acceleration and therefore for evaluating the risk of forced rupture, even though 0.3 % of the peak values are not covered: Practical experience shows this usually works fine!

altran

3. Dynamic Analysis

3.4 Random response analysis

3.4.2 Examples

Base point excitation of a one-mass-oscillator with white noise:

- Oscillator properties: $m = 4\,kg$, $K = 77377.7\,N/mm$
- Fundamental undamped frequency:

$$f_0 = \frac{1}{2\pi}\sqrt{\frac{K}{m}} = 700\text{ Hz}$$

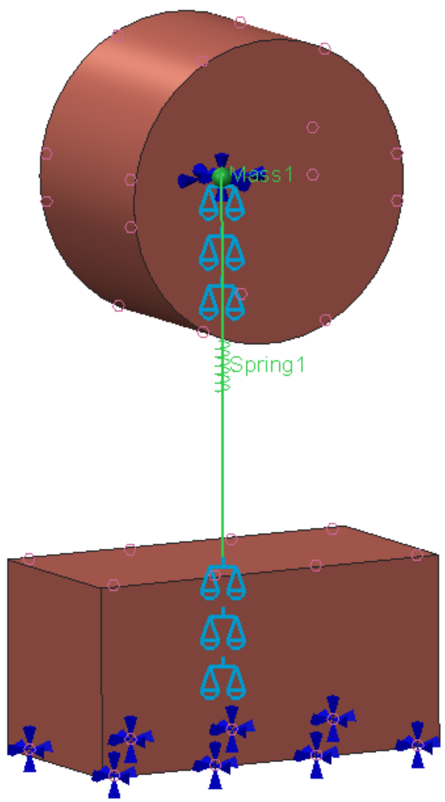
- With 2% damping: $f = f_0\sqrt{1-\beta^2} = 699.86\,Hz$

- Note: The bandwidth B for harmonic excitation (see chapter 1.3) and the "effective bandwidth" B_{eff} for white noise excitation are related as follows:

$$B_{eff} = \frac{\pi}{2}B = \frac{\pi}{2}\frac{f_0}{Q}$$

- So we obtain for harmonic excitation $B = 28\,Hz$ and for white noise excitation $B_{eff} = 44\,Hz$

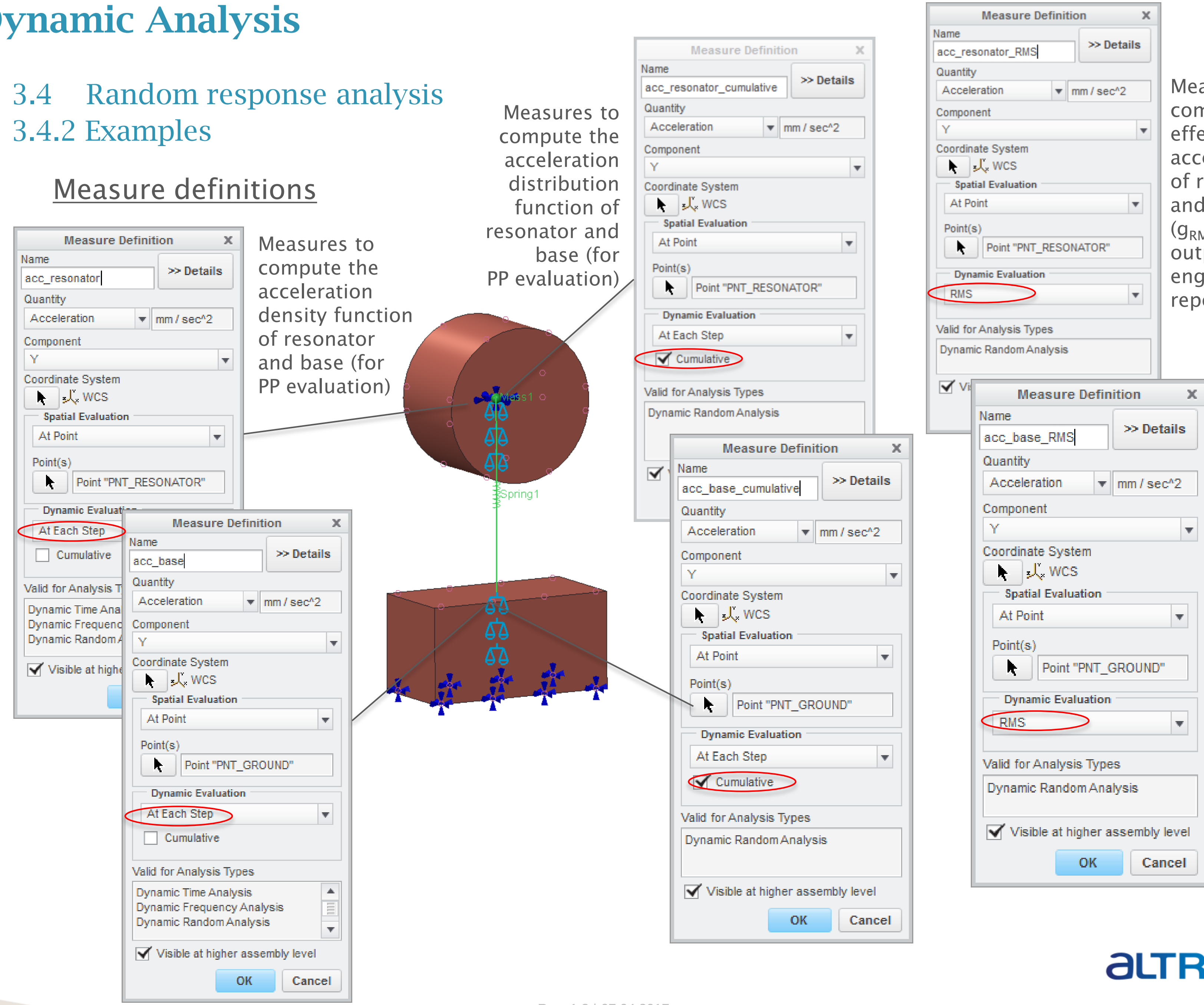
$$Q \approx \frac{1}{2\beta} = \frac{f_0}{B} \Leftrightarrow \beta = \frac{1}{2Q} = \frac{B}{2f_0}$$



3. Dynamic Analysis

3.4 Random response analysis
3.4.2 Examples

Measure definitions



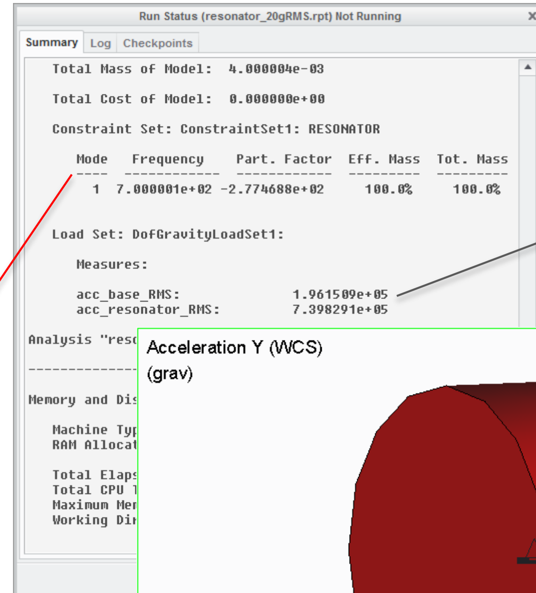
altran

3. Dynamic Analysis

3.4 Random response analysis

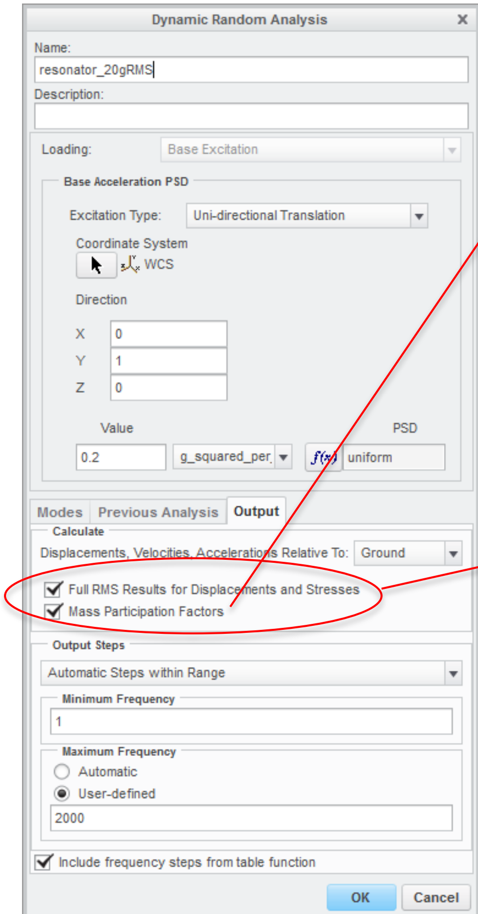
3.4.2 Examples

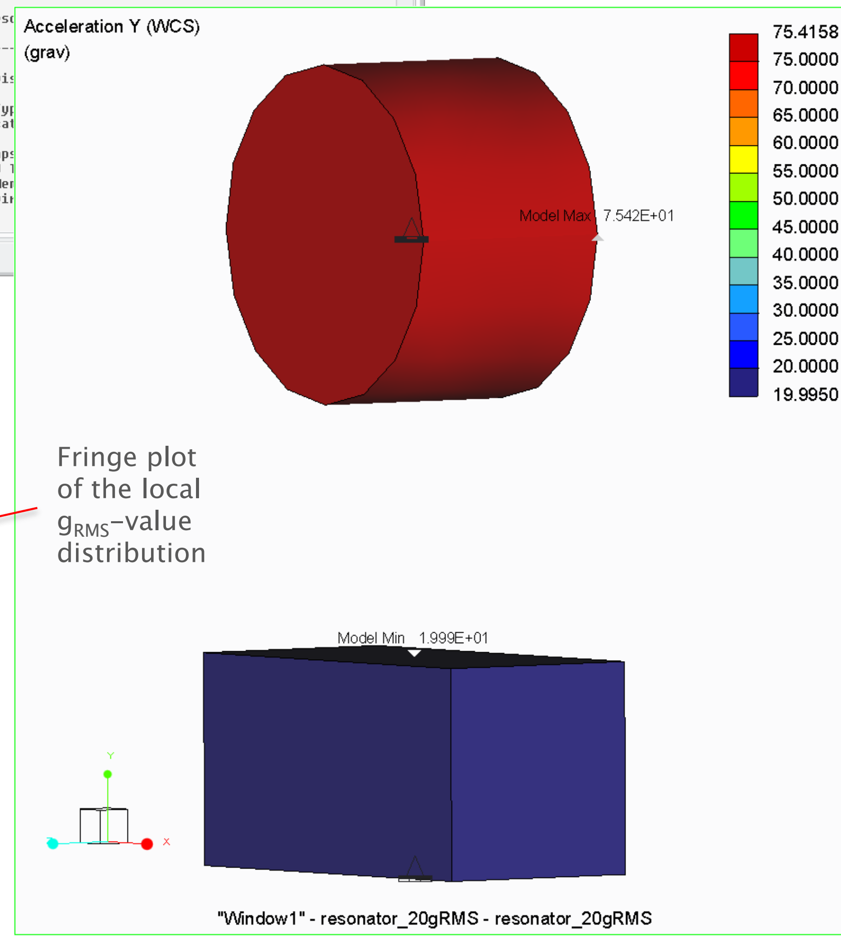
<u>Fringe plot and report file results</u>



g_{RMS} measure output in the engine report:

$196150.9\ mm/s^2 = 19.995\ g$
$739829.1\ mm/s^2 = 75.4158\ g$





Fringe plot of the local g_{RMS}-value distribution

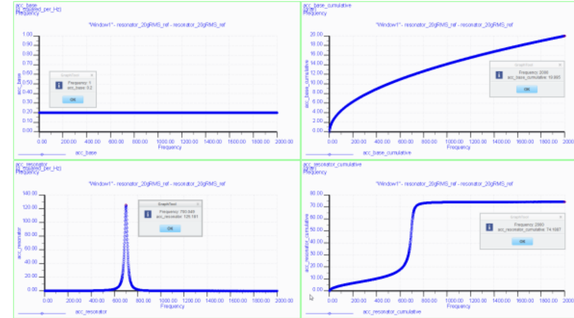
altran

3. Dynamic Analysis

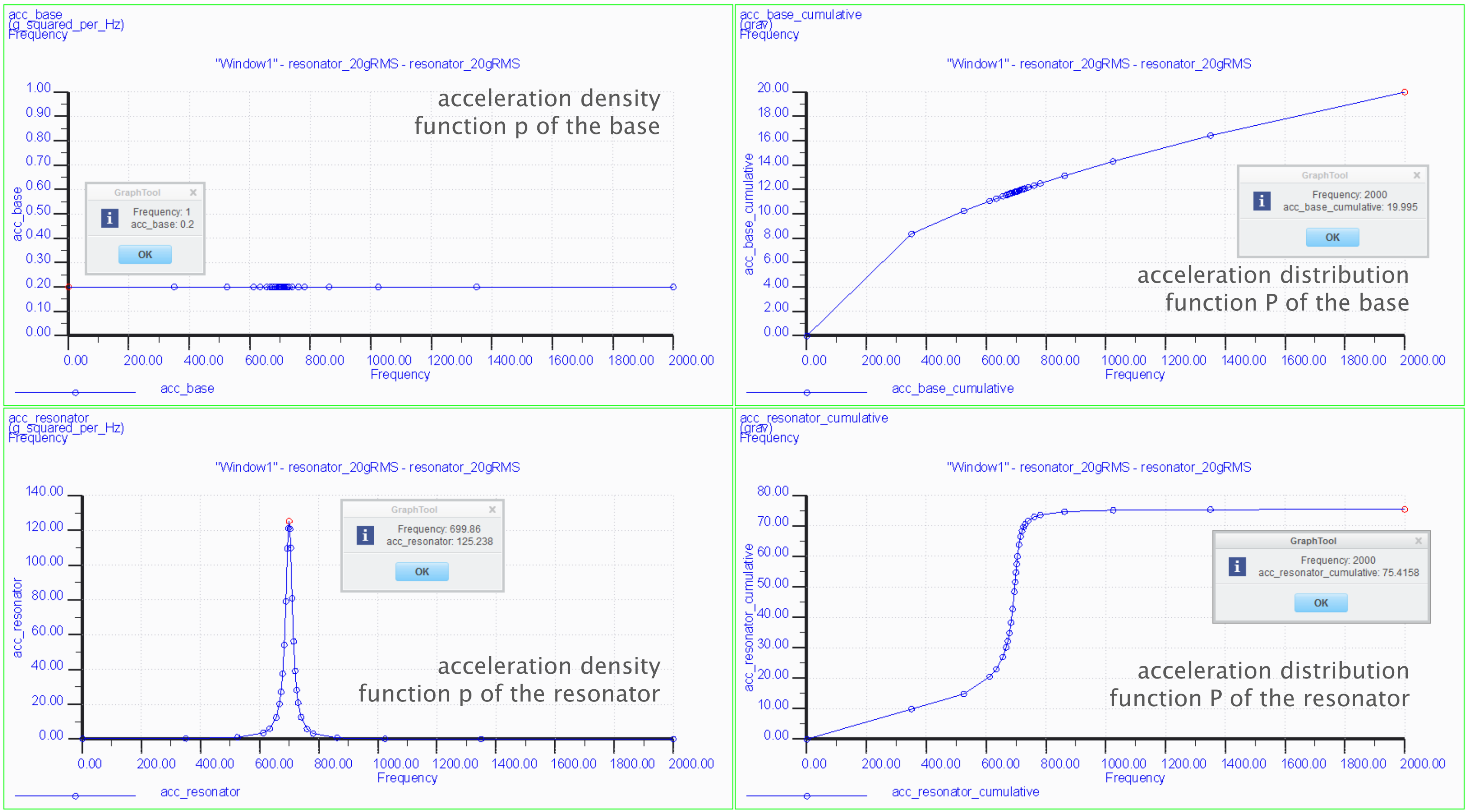
3.4 Random response analysis

3.4.2 Examples

Refined output
with manual
stepping



Measure results for automatic frequency stepping



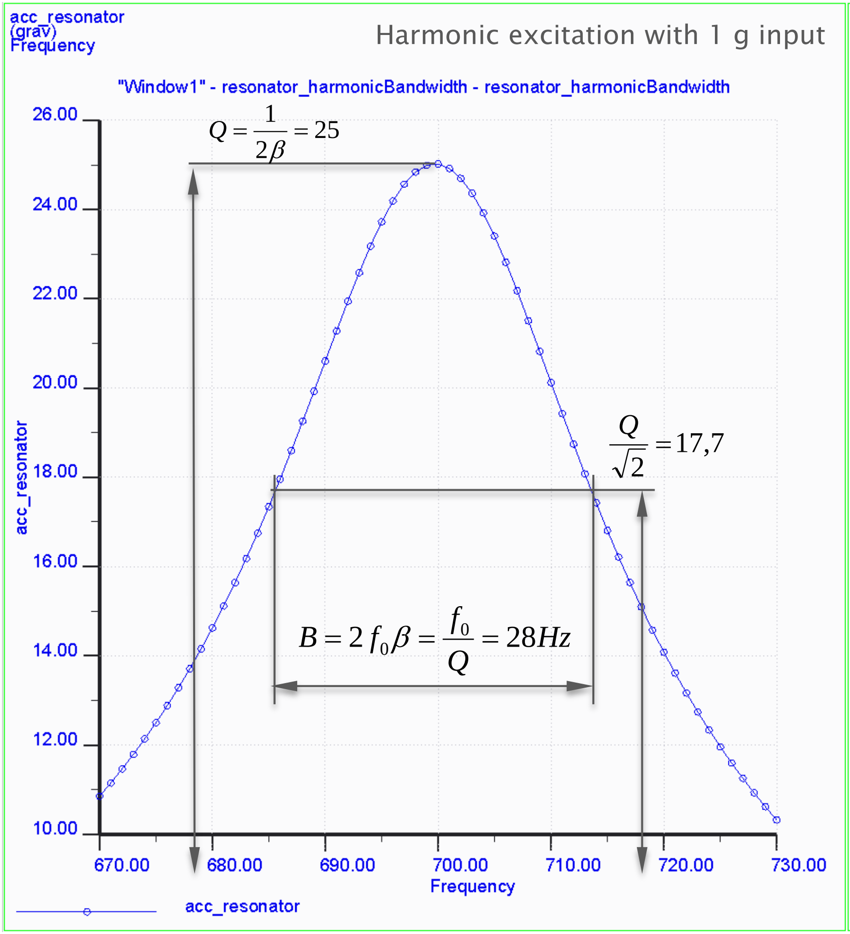
3. Dynamic Analysis

3.4 Random response analysis
3.4.2 Examples

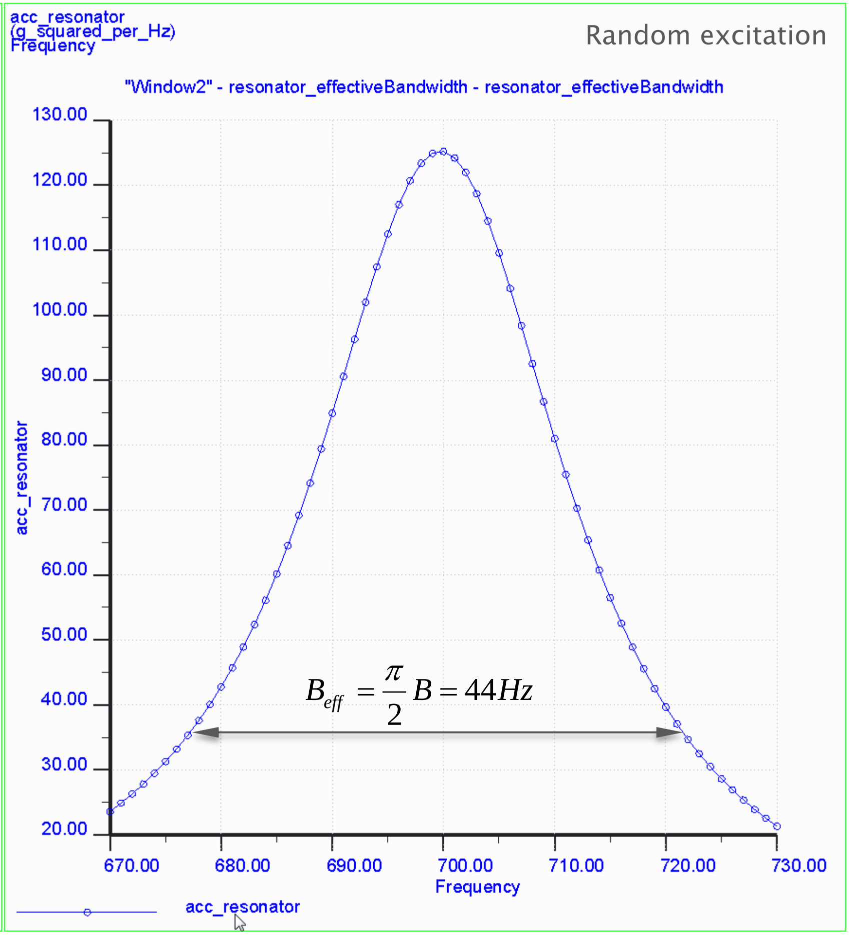
Bandwidth B: The power to maintain the oscillation grows with the square of the amplitude and is at the border of the bandwidth approximately half of the max. resonance power

Bandwidth results in comparison to harmonic excitation

Harmonic excitation with 1 g input



Random excitation

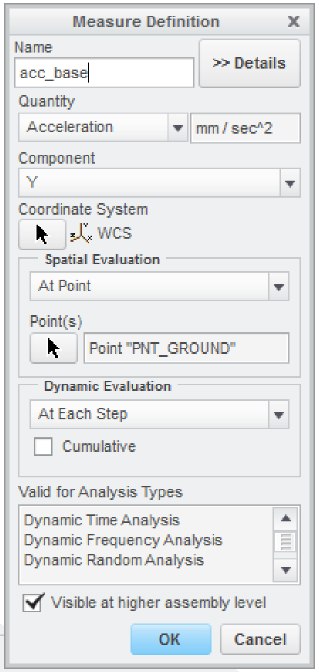


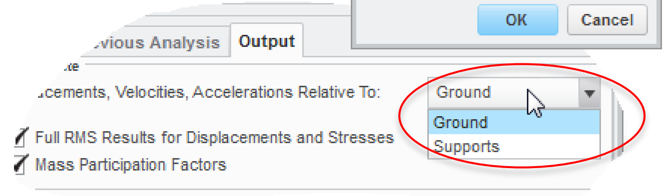
3. Dynamic Analysis

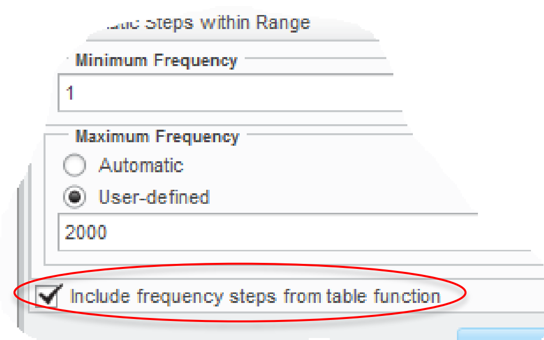
3.4 Random response analysis

3.4.3 Remarks for application

- Always define suitable measures at a base point to check if the given input acceleration density was correctly applied!
- Take care of output settings (relative to ground or supports)!
  Note: Output relative to supports means that the results are expressed in what an observer sitting on a base point (=support) <u>sees when looking at the excited structure</u>, but not what he <u>feels</u>!
- The automatic frequency stepping generator may be fooled out if the lower frequency bound is requested to be 0 Hz, better use e.g. 1 Hz. Otherwise, it may happen that a finer stepping around the natural mode(s) does not correctly take place!
- Therefore, always check stepping of the output acceleration around the resonance frequencies with a suitable measure
- Do not forget to toggle on “Use frequency steps from table function” for non-uniform random spectra to accurately capture function values at these frequencies





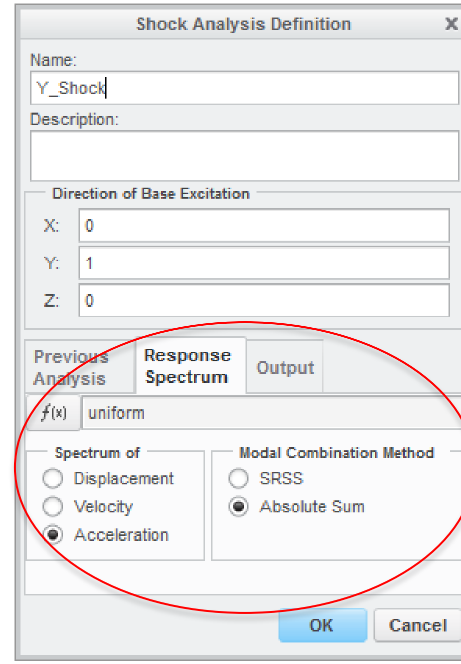


3. Dynamic Analysis

3.5 Dynamic shock analysis

3.5.1 Introduction

- Dynamic shock analysis significantly differs from the other three supported dynamic analysis types: At first sight the user can observe that neither force excitation, nor any modal damping β can be defined!
- The reason is that this analysis type only computes a single image of the expected “worst-case” structural response with help of a given SRS (Shock Response Spectrum), in which damping already is included (this analysis type does not provide any animations like e.g. dynamic time analysis!)
- An SRS is defined as the maximum response of an SDOF (Single Degree Of Freedom) system with a given damping β and variable natural frequency to base point excitation.
- So first, prior to the dynamic shock analysis, an SRS has to be computed by the user for the existing base point excitation (described on the next slide)



What is the advantage of dynamic shock analysis?

- Very low computational resources required (=very quick), since the DEQ does not need to be solved again!
- The evaluation of the shock spectra for different excitation functions allows to judge immediately which excitation creates the worst-case loads!

Remark: Dynamic shock analysis was proposed for the first time in 1932 in the doctoral thesis of the Belgian-American scientist Maurice Anthony Biot

3. Dynamic Analysis

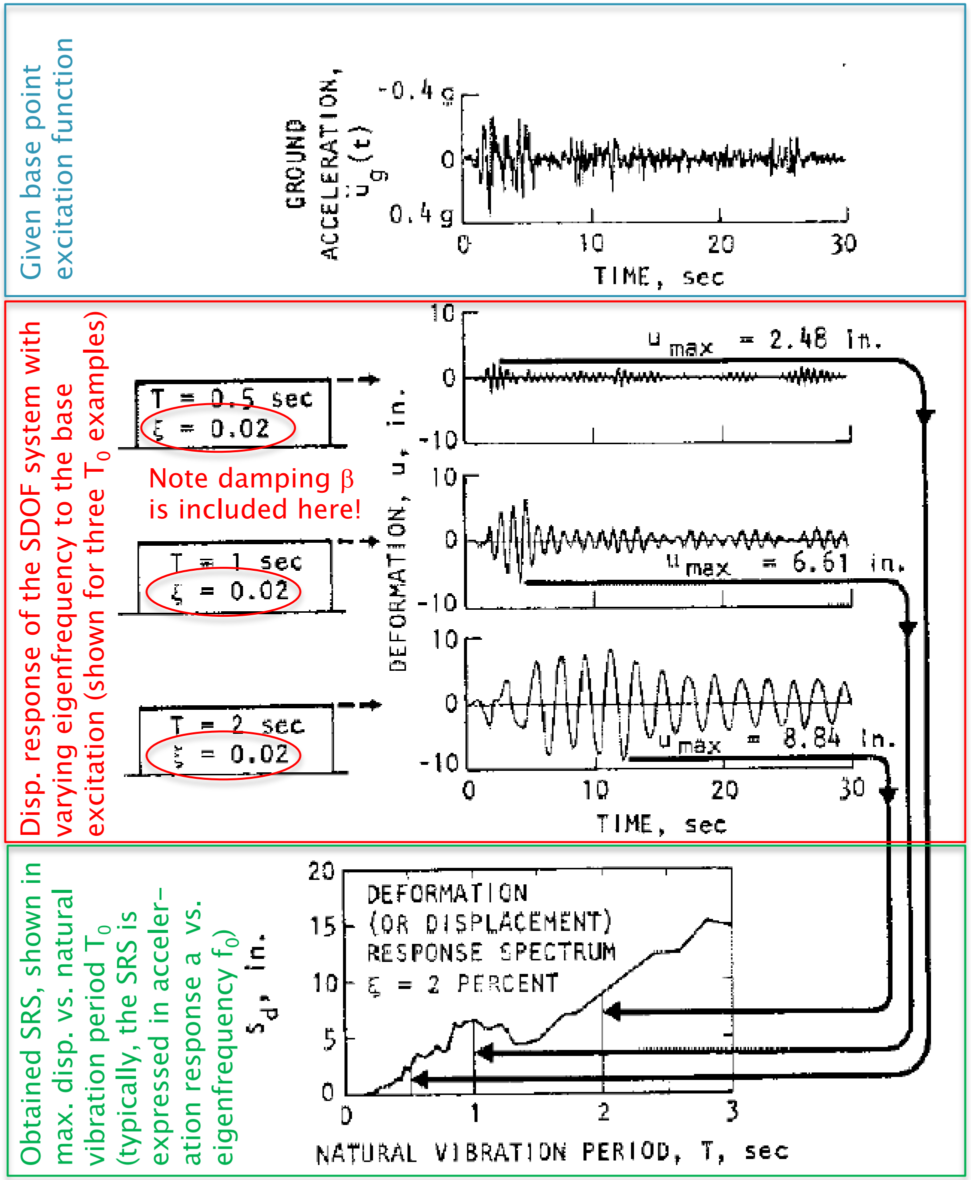
3.5 Dynamic shock analysis

3.5.1 Introduction

- The process to generate an SRS for a given base point excitation is depicted in the right image
- The SRS can be expressed in displacements d (shown right), “pseudo-velocity” v or “pseudo-acceleration” a vs. T_0 or better, as required in Simulate, vs. f_0
- These spectra are linked by the so called “spectra response relation”:

$$s_a = \omega s_v = \omega^2 s_d$$

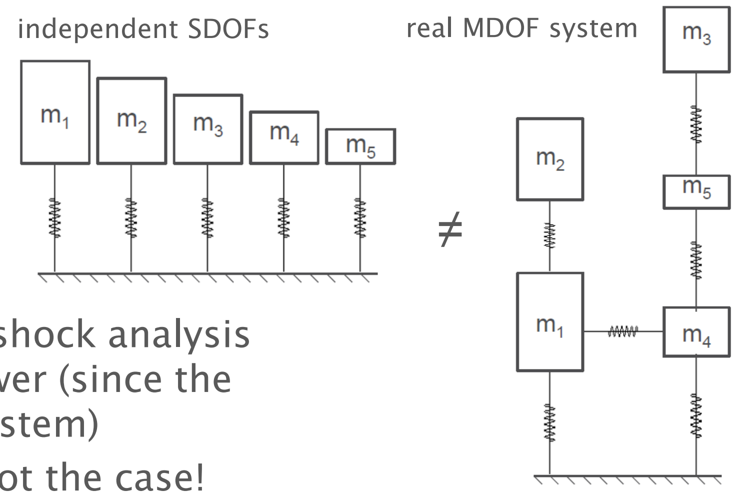
- Important: When computing the SRS, note the different required output references [3]:
  - Displacements: relative supports
  - (Pseudo-)acceleration: rel. ground
  - Pseudo-velocity: has to be computed by the spectra response relation from s_d or s_a



3. Dynamic Analysis

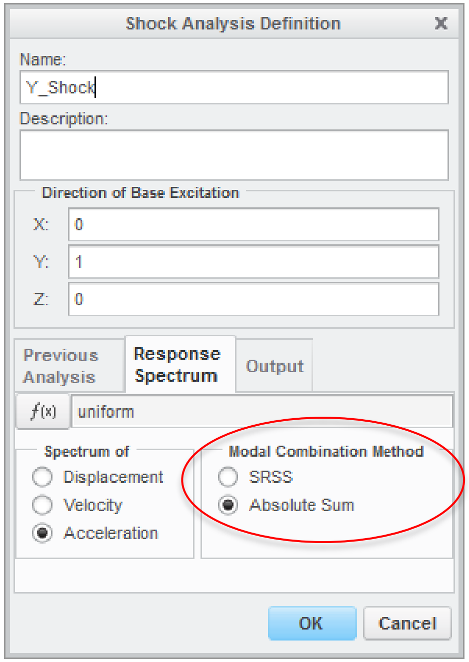
3.5 Dynamic shock analysis

3.5.1 Introduction



<u>Modal superposition method</u>

- For independent SDOF-systems, the dynamic shock analysis always delivers per definition an accurate answer (since the response spectrum was created with such a system)
- Unfortunately, for real MDOF systems this is not the case!
- Indeed, the dynamic response of a linear elastic MDOF system can be expressed as linear combination of its fundamental modes, but the information <u>at which time</u> the modes answer with their maximum, respectively, is lost during the creation of the shock response spectrum
- So two pragmatic (empirical) approaches are offered in Simulate to compute the maximum structural response:
  - ➢ The modes are simply combined with their absolute sum: → very conservative approach, since in reality the modes do not answer with their maximum at the same time
  - ➢ "Square Root of the Sum of the Squares" (averaged superposition, expected to be more realistic)
- For this superposition, the individual max. modal response magnitude is read out from the shock response spectrum, and of course the individual mass participation factors are taken into account (base point excitation!)
- For more details about dynamic shock analysis, see [1-3]



3. Dynamic Analysis

3.5 Dynamic shock analysis

3.5.2 Examples

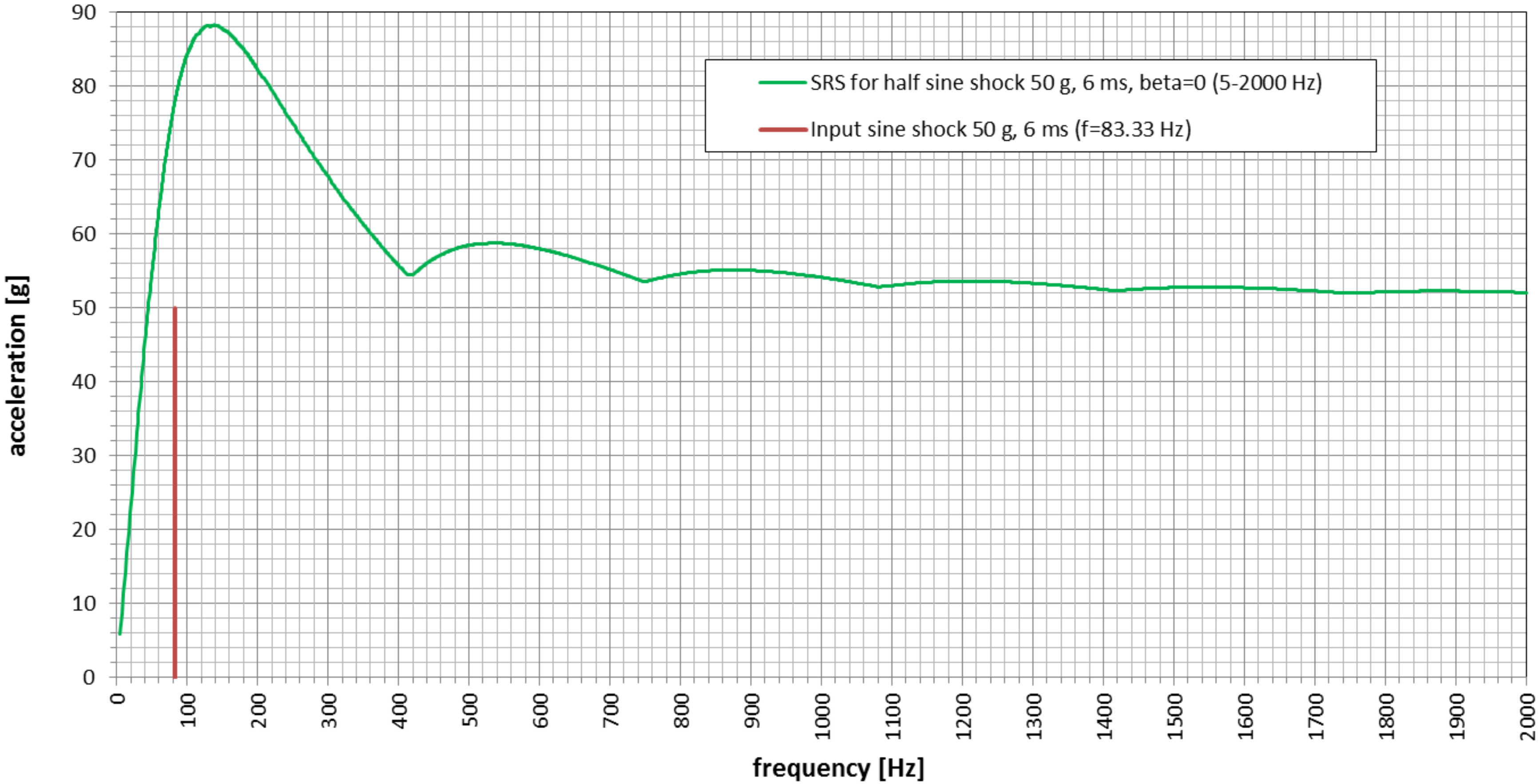
- A shock response spectrum can be obtained in several different ways, e.g.
  - From Literature (typically normalized diagrams are given)
  - By rules and standards (e.g. for civil or military engineering)
  - With help of a Creo simulate global sensitivity study
  - With Mathcad as shown in [7], [8]
  - By other suitable software
  - …
- In [3], a method is shown how to use the Simulate global sensitivity study for this:
  - Create a simple SDOF model with just a point mass and a discrete spring
  - Enter the base excitation function of interest in a dynamic time analysis, also enter the damping present in the real structure you want to compute afterwards
  - Define a measure for the max. system response over the complete analysis time (use max. displacement output rel. supports, or max. acceleration relative ground, but do not use a max. velocity response measure; see also [3], p. 23!)
  - Define a property parameter to vary the fundamental frequency of the SDOF system (i.e. spring stiffness or mass)
  - Sweep this parameter in a global sensitivity study referencing the dynamic time analysis in a way that the complete frequency span is covered for which you need the SRS. Note: For practical reasons, you may need more than one analysis with different step width and edge frequencies for this, since at lower frequencies you have less sample points ($f \sim \sqrt{K}$)
  - After the study, draw the max. displacement or acceleration response measure vs. SDOF frequency (MS EXCEL) – this is the SRS you need!

3. Dynamic Analysis

3.5 Dynamic shock analysis

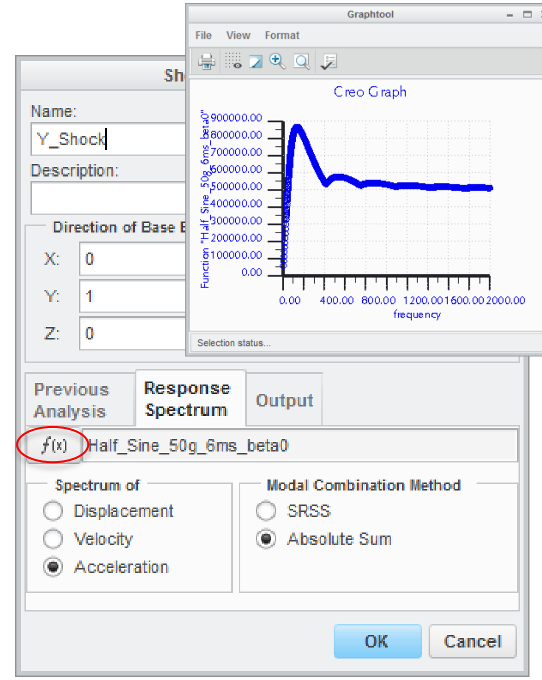
3.5.2 Examples

- By following this procedure, you should obtain the following shock response spectrum for an acceleration half sine shock of 50 g, 6 ms duration (→ f=83.3 Hz) and no damping, see [3]

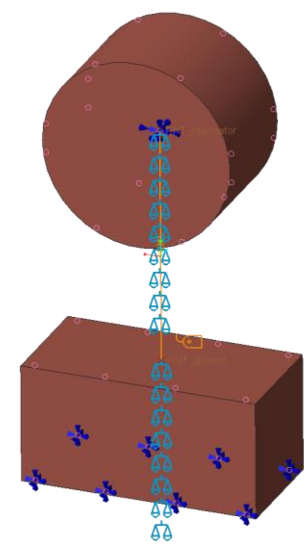


3. Dynamic Analysis

3.5 Dynamic shock analysis
3.5.2 Examples



- Now, take this spectrum and import it into the response spectrum form sheet of the dynamic shock analysis definition dialogue
- Run the dynamic shock analysis e.g. with the 700 Hz-SDOF-system used in chapter 3.4
- A max. relative displacement result of 0.028 mm will be computed (see next side left). Note: Displacement output in a Simulate shock response analysis is always given relative to supports, never relative to ground!
- Run two dynamic time analysis for cross-checking: One relative to ground and one relative to supports, with the measures shown below – you should obtain similar results (units mm, t, s)
- For an SDOF-system, you can simply compute the max. acceleration out of the shock analysis displacement result with help of the spectra response relation (see next slide)



```
Measures:

acc_base_max:          4.905000e+05
acc_resonator_max:     5.413873e+05
disp_base_max:         2.615889e+00
disp_resonator_max:    2.595781e+00
vel_base_max:          1.520799e+03
vel_resonator_max:     1.542662e+03

Analysis "HalfSine50g6ms_ground" Completed
```

```
Measures:

acc_base_max:          0.000000e+00
acc_resonator_max:     6.589017e+04
disp_base_max:         0.000000e+00
disp_resonator_max:   -2.798673e-02
vel_base_max:          0.000000e+00
vel_resonator_max:    -2.596859e+01

Analysis "HalfSine50g6ms_supports" Completed
```

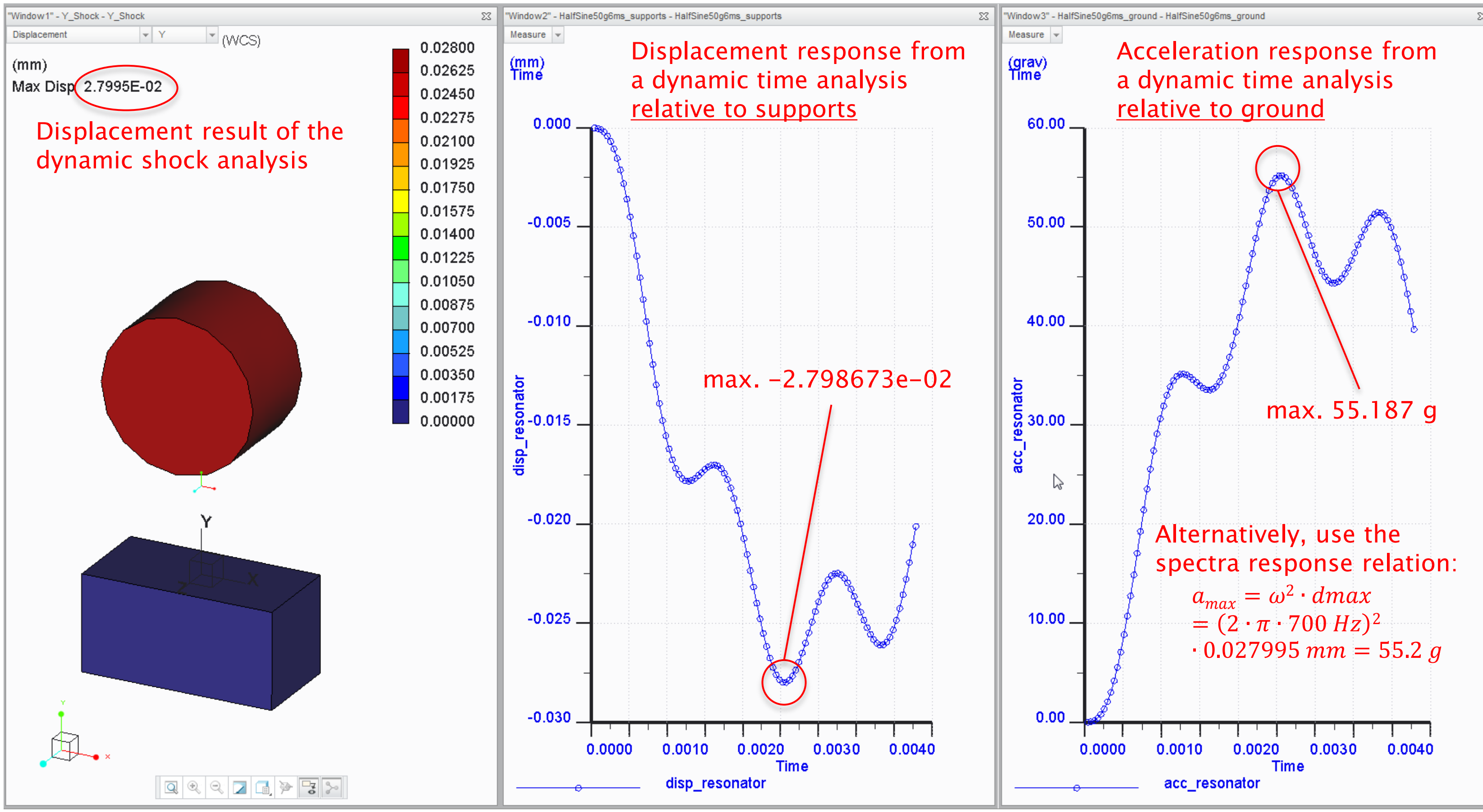
Remark: Using the spectra response relation for velocities may deliver a very inaccurate result compared to the real rel. velocity, see also [3]: $v_{max} = \omega \cdot dmax = (2 \cdot \pi \cdot 700Hz) \cdot 0.027995\, mm = 123.1\, \frac{mm}{s} \gg 25.97\, \frac{mm}{s}$

3. Dynamic Analysis

3.5 Dynamic shock analysis

3.5.2 Examples

- Response comparison of the 700 Hz SDOF-resonator for the different analysis types (50 g, 6 ms half sine shock, β=0)

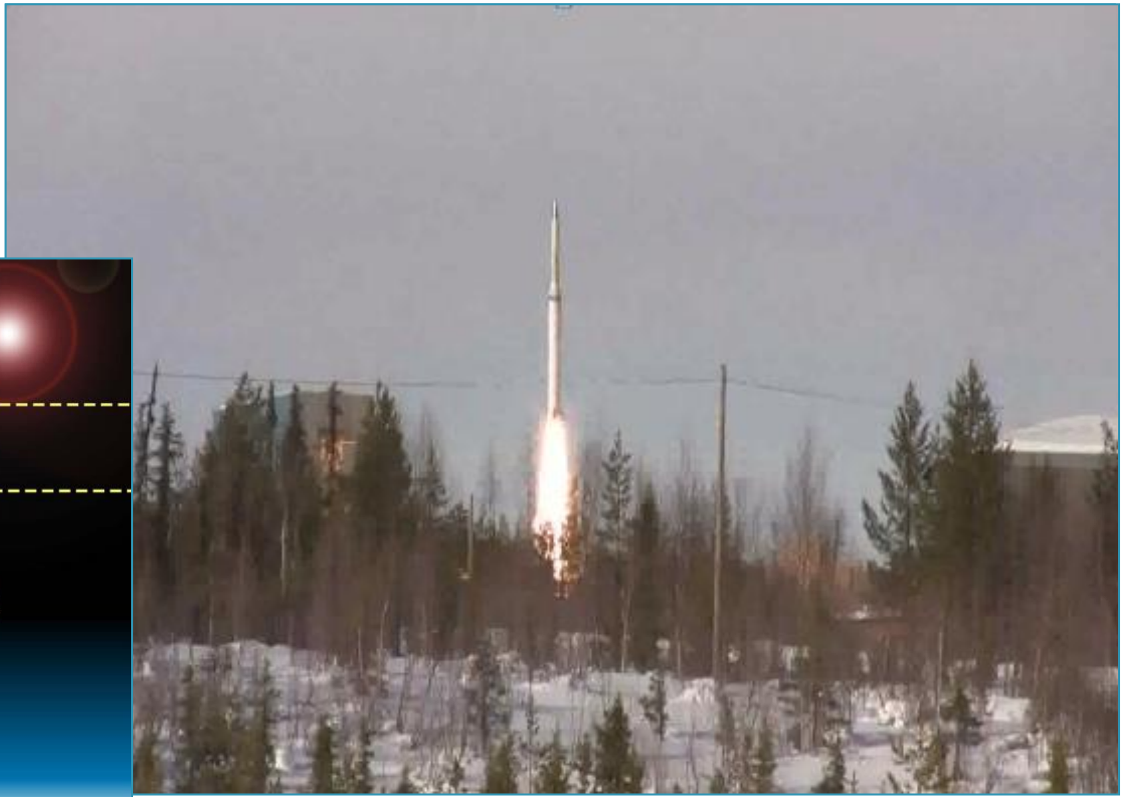


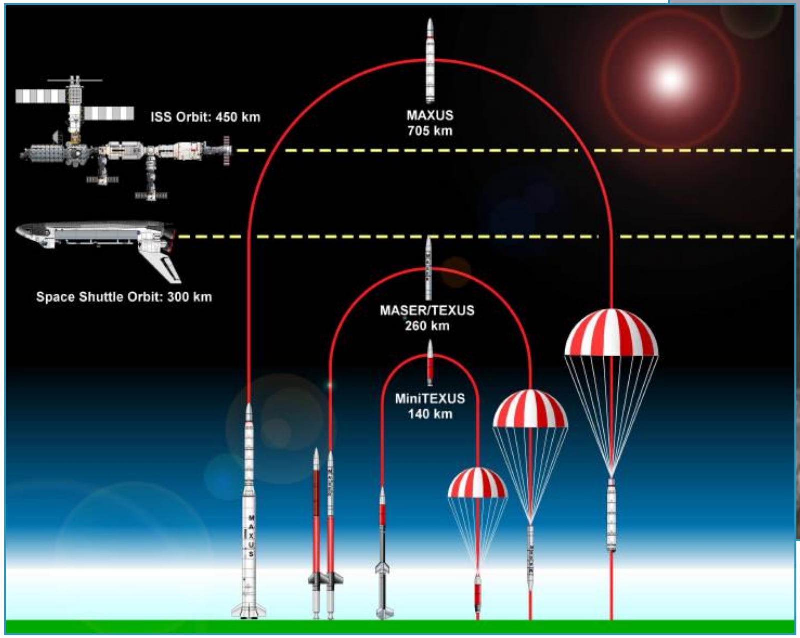
3. Dynamic Analysis

3.6 Examples for dynamic analyses of big system models

- For Zero-G experiments, Airbus DS in Bremen develops various payloads for the ESA MAXUS and TEXUS sounding rockets
- Such experiment platforms are subjected to very high dynamic loads during launch

Movie of a MAXUS rocket launch (Kiruna, Sweden)



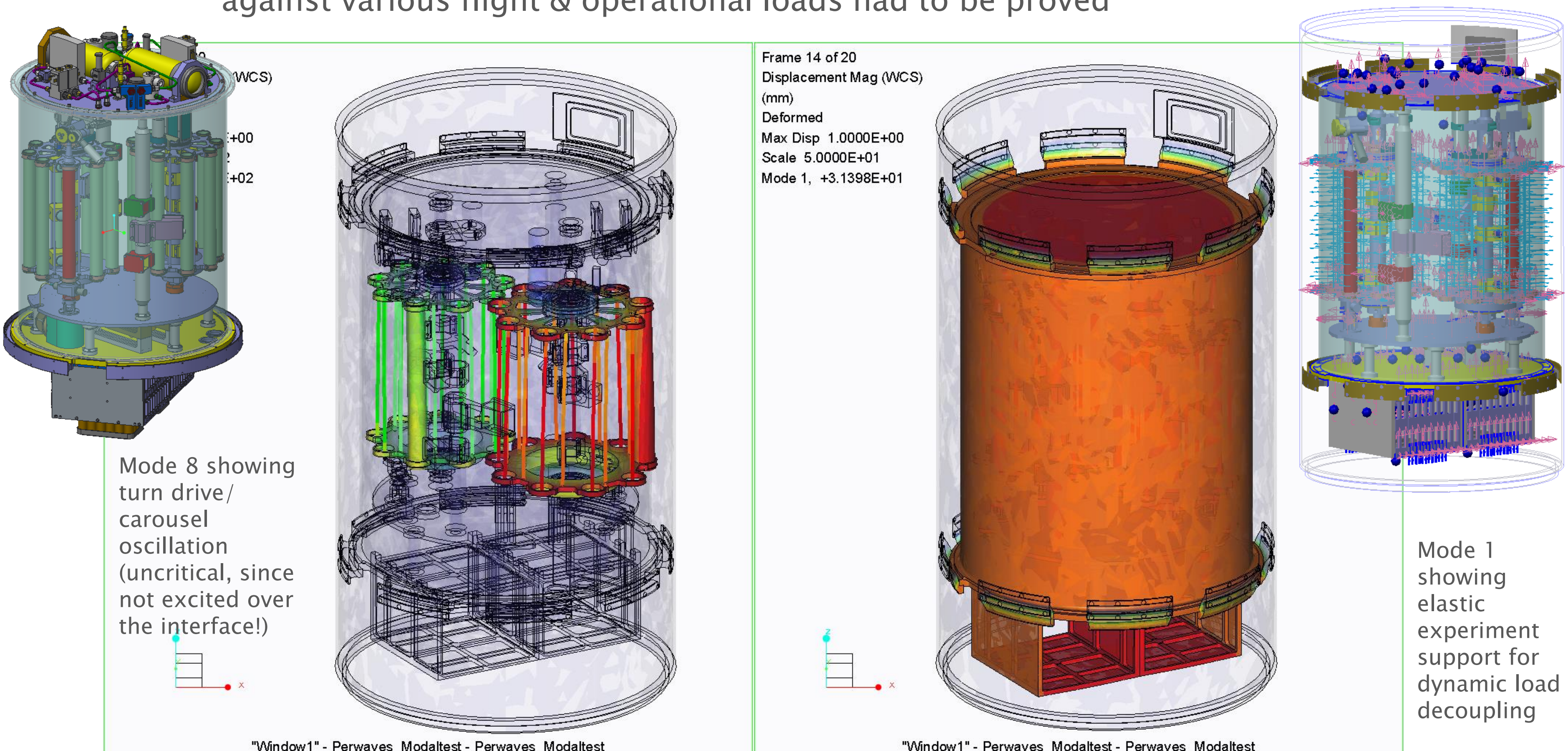


MAXUS & TEXUS ballistic sounding missiles simplified flight profile

3. Dynamic Analysis

3.6 Examples for dynamic analyses of big system models

- PERWAVES: An experiment that examines combustion processes in Zero G environment
- This experiment contains mechanisms and several fragile glass tubes: Their resistance against various flight & operational loads had to be proved



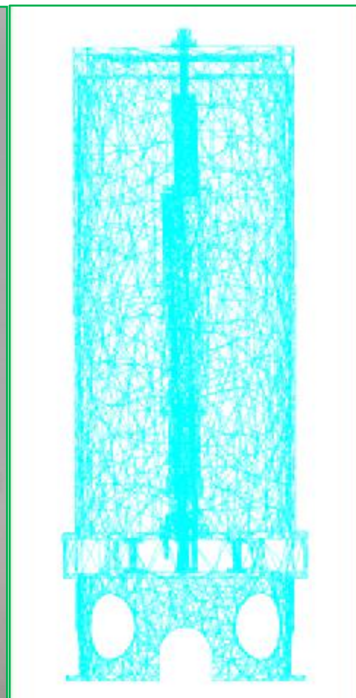
Mode 8 showing turn drive/ carousel oscillation (uncritical, since not excited over the interface!)

Mode 1 showing elastic experiment support for dynamic load decoupling

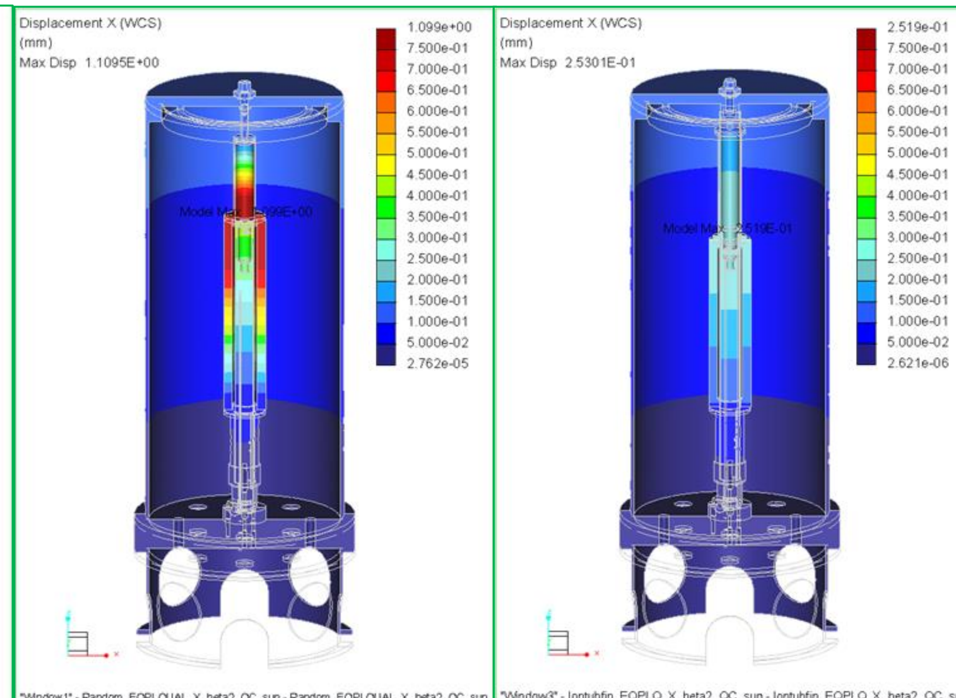
3. Dynamic Analysis

3.6 Examples for dynamic analyses of big system models

- GRADECET – A high temperature furnace that melts metallic specimens at 1700 °C and recrystallizes them convection free in Zero-G environment on MAXUS flights
- Improvements had to be found and analyzed to reduce dynamic random loading within the heating section



Exemplary harmonic excitation of the first mode of the tantalum “finger”, covering a ceramic tube around the specimen

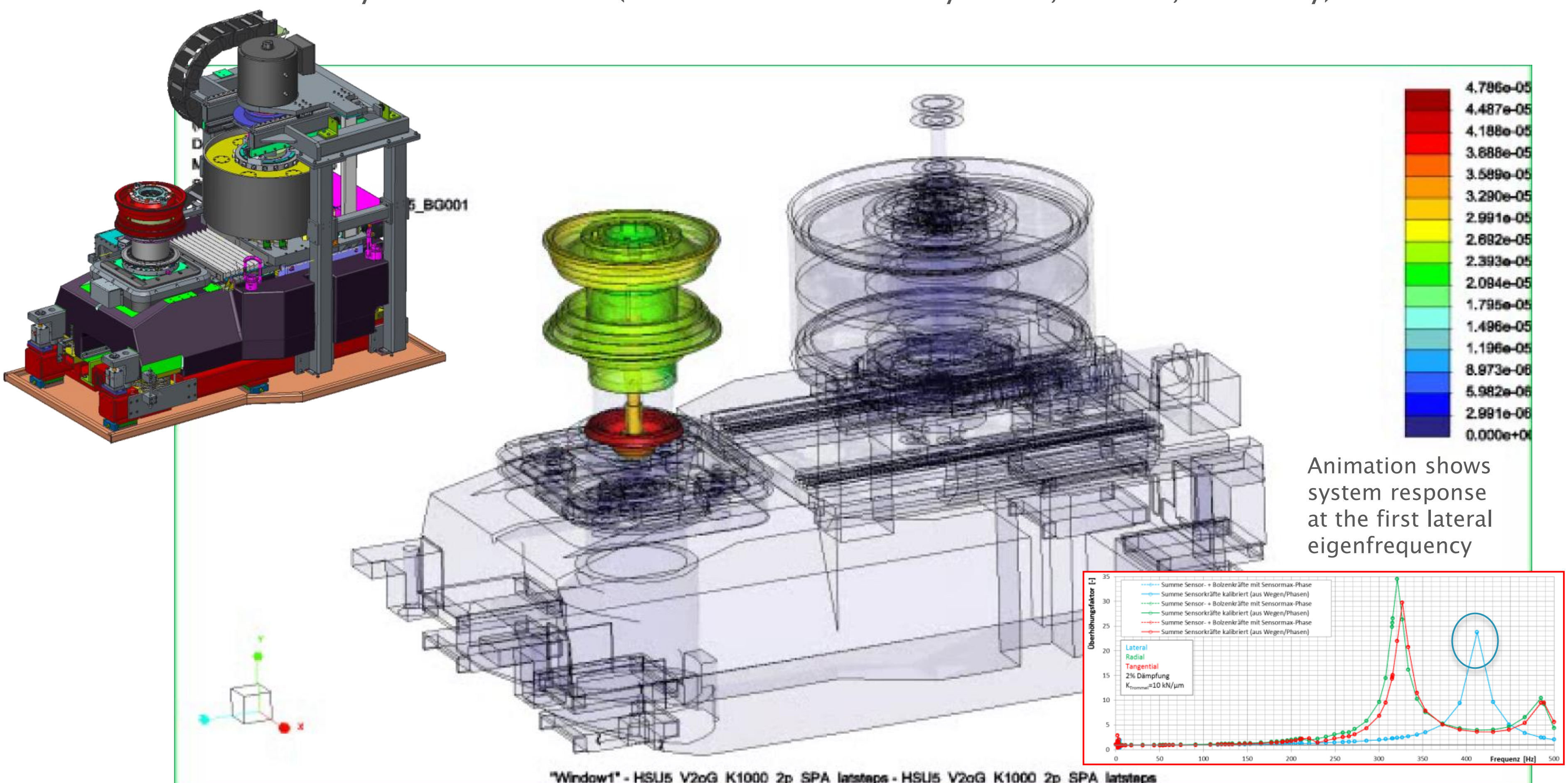


1σ-displacement response of initial (left) and improved design (right)

3. Dynamic Analysis

3.6 Examples for dynamic analyses of big system models

- Computation of the force transfer functions of a tire test rig for high speed uniformity measurements (HSU 5 from ZF Test Systems, Passau, Germany)

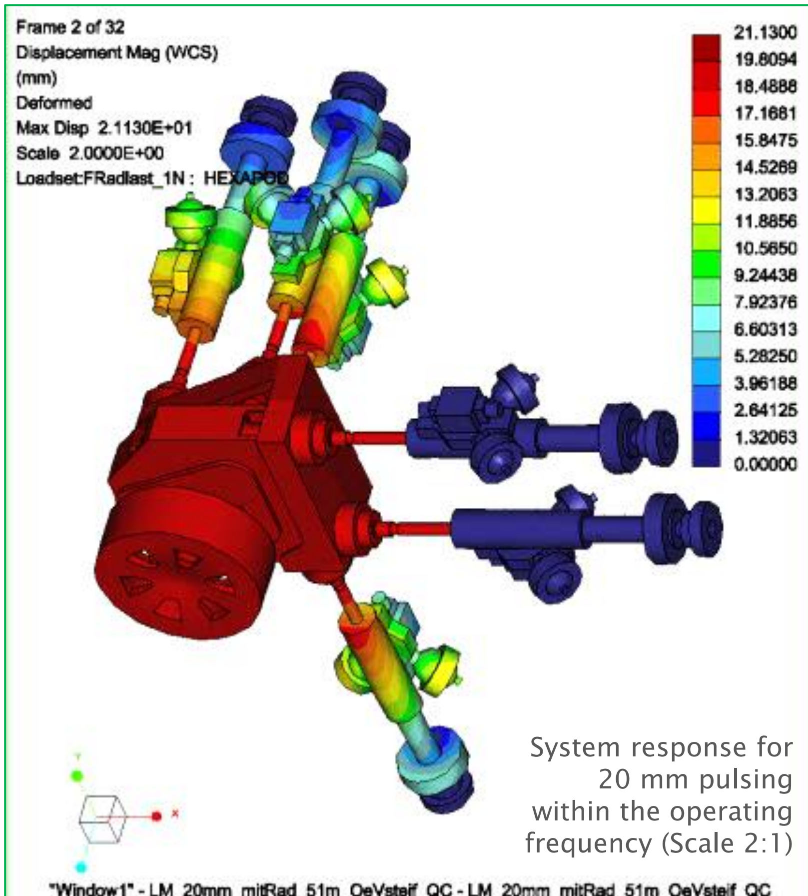


altran

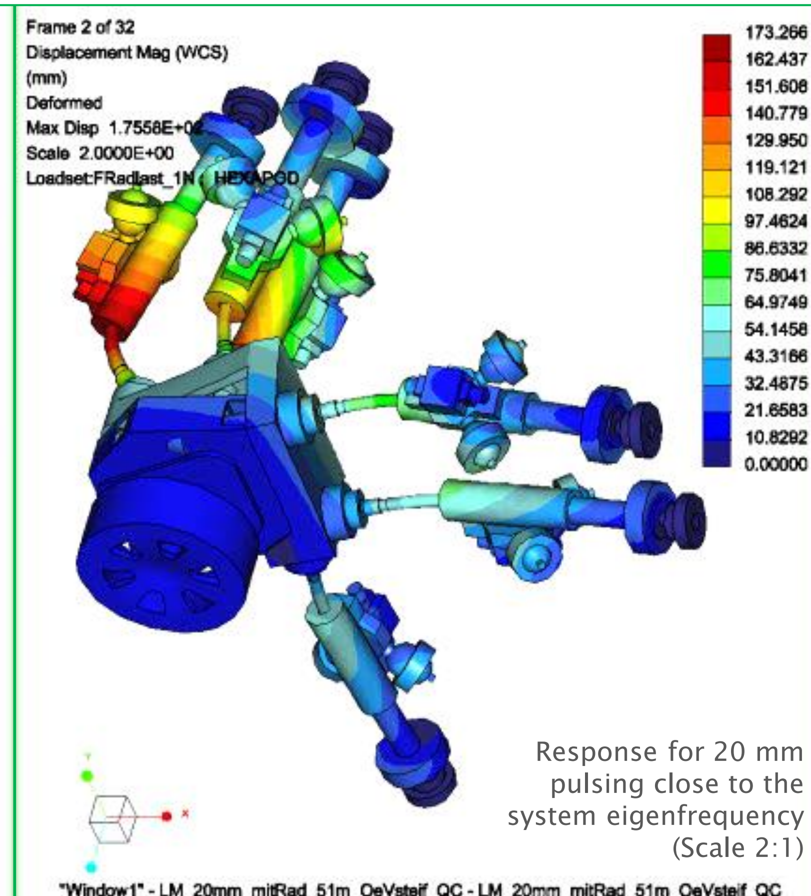
3. Dynamic Analysis

3.6 Examples for dynamic analyses of big system models

- Behavior of a new ZF highly-dynamic half axis test rig in hexapod architecture
  - One goal of the examination was to estimate the response function for harmonic pulsing of the two tire load cylinders with an amplitude of 20 mm
  - This was done with help of the so called “seismic mass” concept



System response for 20 mm pulsing within the operating frequency (Scale 2:1)

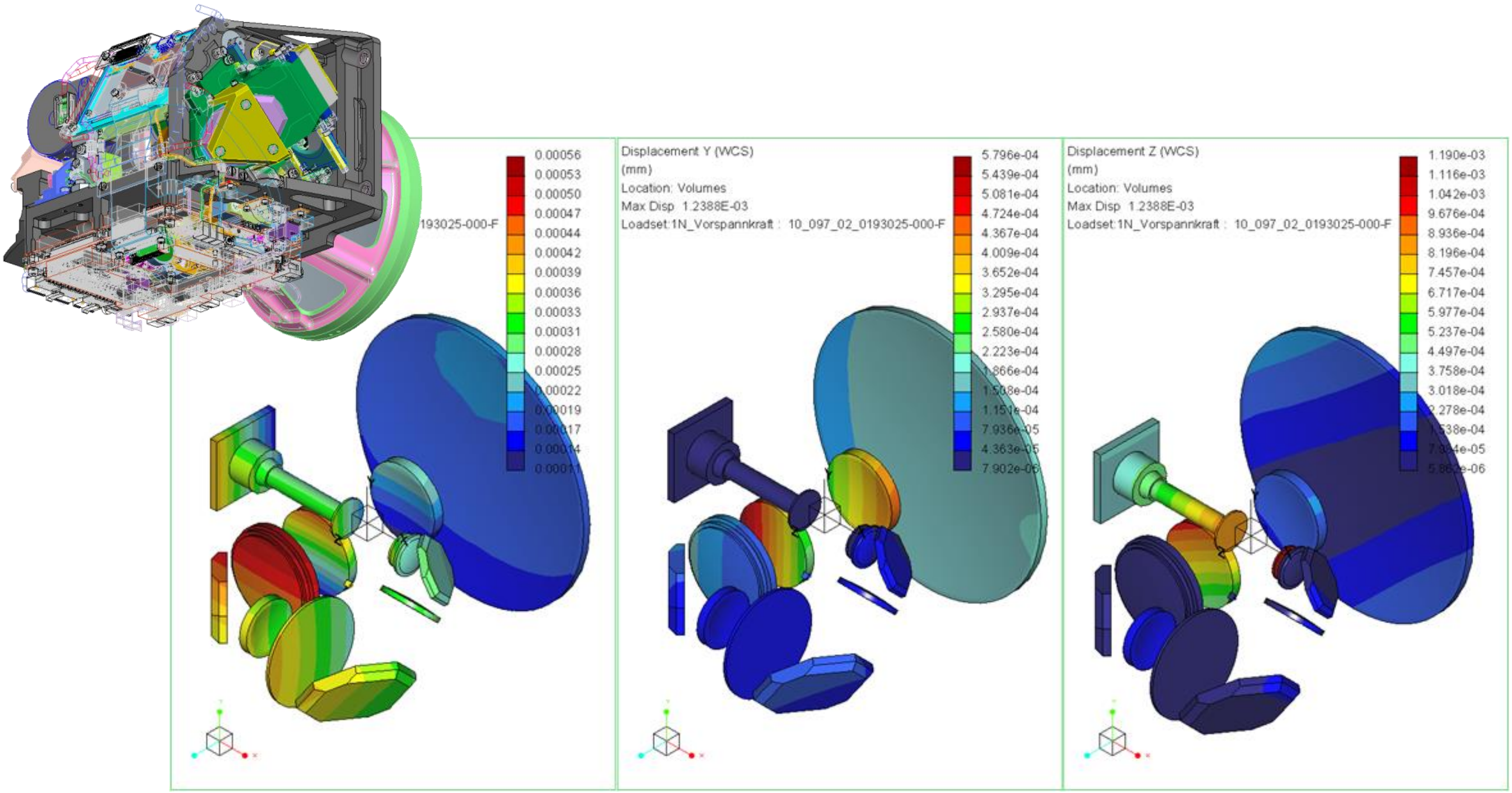


Response for 20 mm pulsing close to the system eigenfrequency (Scale 2:1)

3. Dynamic Analysis

3.6 Examples for dynamic analyses of big system models

- Random response analysis of a thermal imaging system to compute optical surface displacements under specified acceleration spectral density (Hensoldt, Oberkochen)

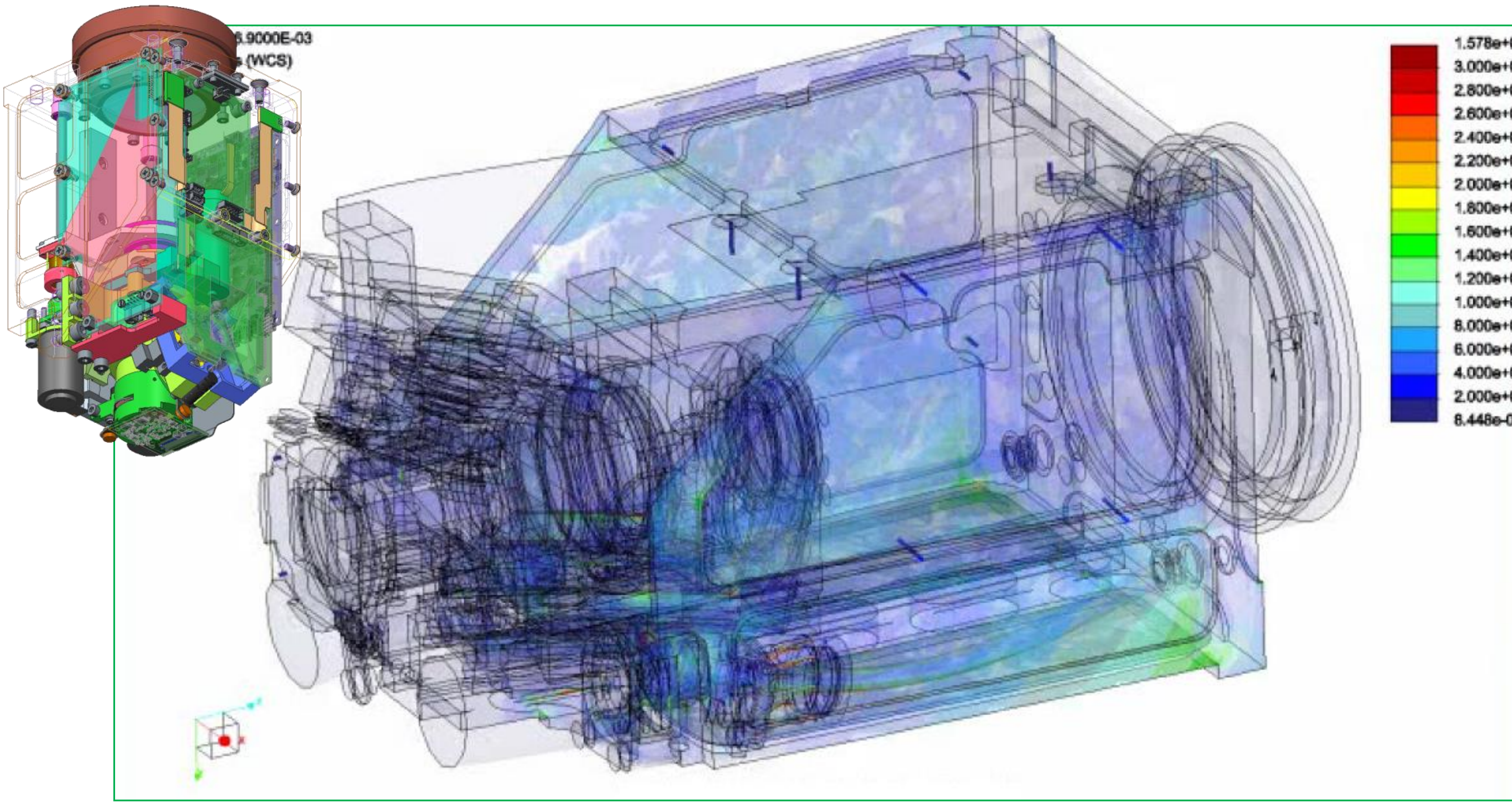


3. Dynamic Analysis

HENSOLDT
Detect and Protect.

3.6 Examples for dynamic analyses of big system models

- Shock response behavior of an imaging system for visible and short wave infrared light (Hensoldt, Oberkochen)

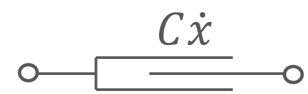


altran

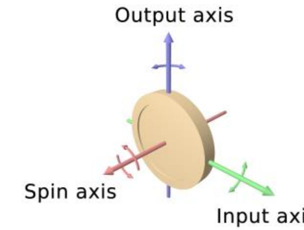
4. Feedback to the software developer PTC

4.1 Missing functionality & enhancement requests to PTC for dynamic analysis

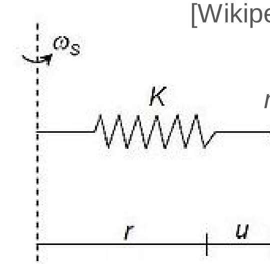
- Missing discrete damper:
  - ➢ add a linear, discrete damper that connects two points at least with a simple translational and/or rotational damping constant C



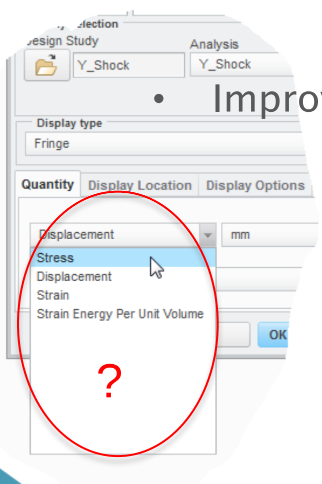
- Missing support for rotating machinery:
  - ➢ Support gyroscopic effects in modal analysis: Whereas for slim rotors (e.g. the shaft example) this effect can be neglected, for massive, disk-shaped rotors (fly-, gearwheels) it is of fundamental importance for correct prediction of their rotordynamic behavior!
  - ➢ Support centrifugal softening (also called "spin softening"): This effect leads to a decrease of fundamental frequencies since the elastic stiffness K has to be replaced by the effective stiffness: $K_{eff} = K - \omega_s{}^2 m$ (can be obtained by balancing spring and centrifugal force). This is of importance if the radial displacement u under centrifugal force cannot be neglected compared to the radius r (e.g. for fast spinning turbine blades). In case $K < \omega_s{}^2 m$: Instability appears!

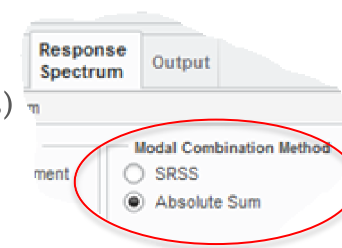


[Wikipedia]



- Improved functionality for dynamic shock analysis:
  - ➢ Support more modal combination methods from literature, like e.g. 1st mode absolute, all higher modes as SRSS
  - ➢ Support not only displacement, but also velocity and especially acceleration fringe plots in the postprocessor (similar for measures)
  - ➢ Since translational excitation can just be defined in the world coordinate system, also support user-defined coordinate systems (Workaround: Assemble the model into a higher-level assembly with suitable orientation!)

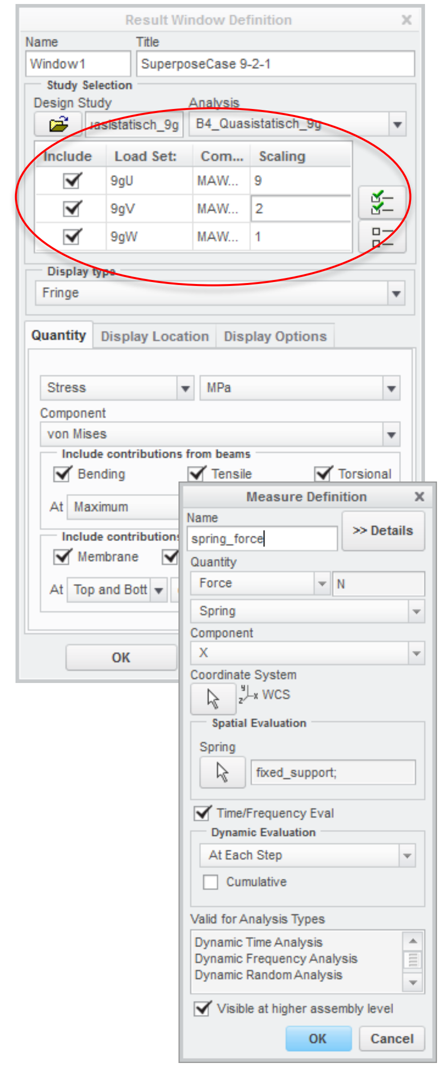




4. Feedback to the software developer PTC

4.1 Missing functionality & enhancement requests to PTC for dynamic analysis

- Better support of results superposition in the post processor:
  - Currently, results superposition is only supported within a single linear static analysis containing at least 2 load sets
  - It would be pretty useful if the user could superpose and scale results of (any) different analyses (even, although risky, any load step of a nonlinear analysis), as long as the models have similar geometry, mesh and plotting grid
  - This could e.g. allow to combine the static (constant) prestress state of a prestress modal analysis e.g. with the varying stress states of a dynamic time or frequency analysis based on a prestress modal analysis (enables mean– and deflection stress evaluation, which is currently not possible)
- Missing measures: Unfortunately, several measures are not supported in various dynamic analysis types, like e.g.:
  - No phase output for spring forces in dynamic frequency analysis (workaround is using displacements and phase of displacements of the spring end points, but this is by far not accurate enough for very stiff springs!)
  - No constraint or resultant force/moment measure output in dynamic analysis at all
  - No velocity and acceleration output in dynamic shock analysis (just displacement and stress output, even for fringe plots!)
  - No support of any fastener measures in random response analysis
  - …



4. Feedback to the software developer PTC

4.1 Missing functionality & enhancement requests to PTC for dynamic analysis

- Improved support for analyzing very large system models:
  - Experience won in many industrial projects shows that the code works very fine for analyzing large CAD-assemblies (see chapter 3.6): The excellent CAD-FEM integration and associativity of the analysis model with the CAD model allows a very quick modification of the design which can immediately be re-meshed and re-analyzed in the p-FEM environment
  - Also all idealizations (beams, shells, springs, discrete masses, rigid and weighted links) are supported and are well suitable to decrease model size
  - However, the level of detail which can be taken into account in the model is usually driven by the modal analysis with the requested number of modes, the chosen plotting grid and the optional modal stress request. For big models, a lot of RAM is required so that the engine does not crash!
  - To decrease RAM resource consumptions, it would be useful that subsequent dynamic analyses (which typically need much less RAM compared to the modal analysis) can reference a series of modal analyses with split frequency band requests, respectively, e.g. from 0-1000 Hz, 1000-2000 Hz and 2000-3000 Hz, and not only one single modal analysis requesting all modes of the complete frequency domain (0-3000 Hz)
  - Multi-threading should be supported in dynamic analysis, too (in modal analysis, the code already addresses many CPU automatically). Until now, the user can only obtain a significant speed increase in dynamic analysis by using an SSD hard drive!
- Implementation of superelements for model size reduction:
  - For very large system models, a submodeling technique would be very useful, that means certain subassemblies can just be represented as a “superelement” to decrease hardware resources and increase analysis speed

altran

4. Feedback to the software developer PTC

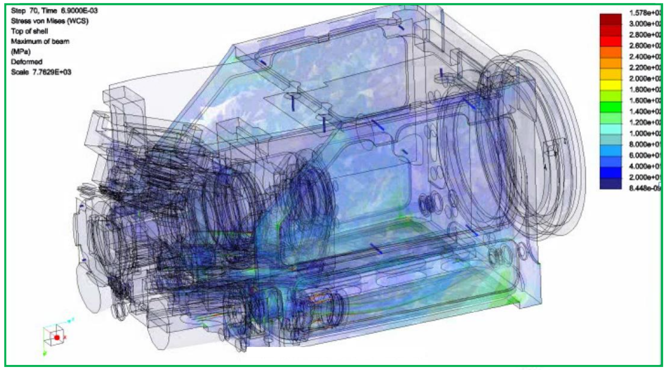
4.2 Known issues in dynamic analysis and possible workarounds

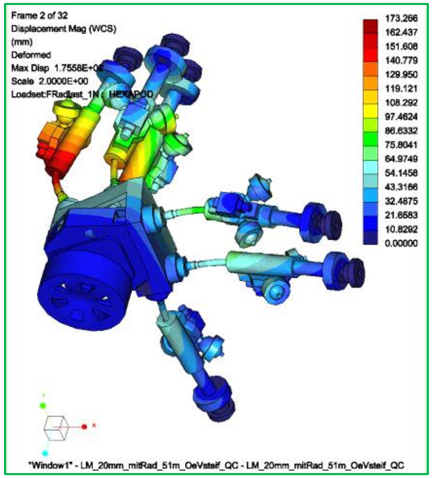
Project work with the product uncovered the following issues to be fixed by PTC R&D:

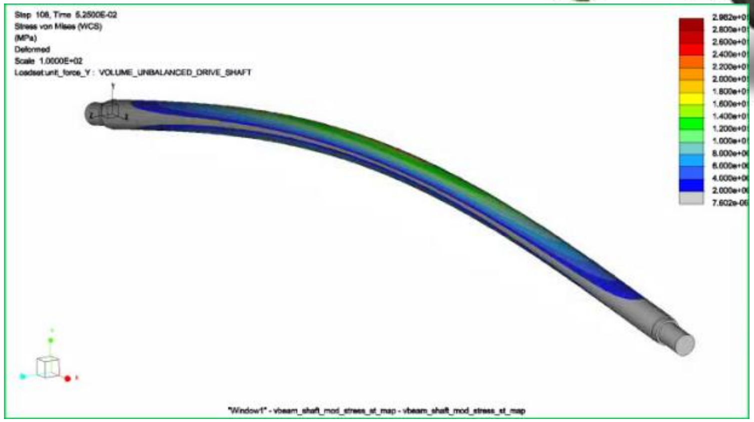
- SPR 4714483: Off-diagonal terms in the spring stiffness matrix of advanced springs lead to incorrect results in fundamental frequency computation (no workaround available, currently do not use off-diagonal terms at all!)
- SPR 2875703: In dynamic frequency analysis with force excitation and phases between the exciting forces, wrong animations/PP plots for certain result components may appear (try with the shaft example in chapter 3.3.2 and compare with dynamic time results; you may use dynamic time analysis as workaround!)
- SPR 2847768: In a random response analysis, Simulate may compute wrong von Mises stress hot spots (this may appear especially for very big system models)
- SPR 2867898: Mixed models with shell-solid links may deliver wrong results in a modal analysis (locking appears especially at high p-levels).
  Workaround: Try to prevent shell-solid links by suitable modeling/meshing!
- SPR 4461169: In a model containing thin solid elements (wedges and bricks); Simulate does not correctly detect rotational rigid body modes with standard settings in a modal analysis (wedges and bricks – unlike tetrahedrons – need very high p-levels to correctly detect rigid body modes at a frequency of zero!). Workaround: Use tetrahedrons or enforce high accuracy in the modal analysis to use high p-levels!
- SPR 4967524: In a model containing variable thickness shells; msengine hangs up during mass calculation. Workaround: Replace variable thickness shells with volumes or with stepwise varying constant thickness shells!

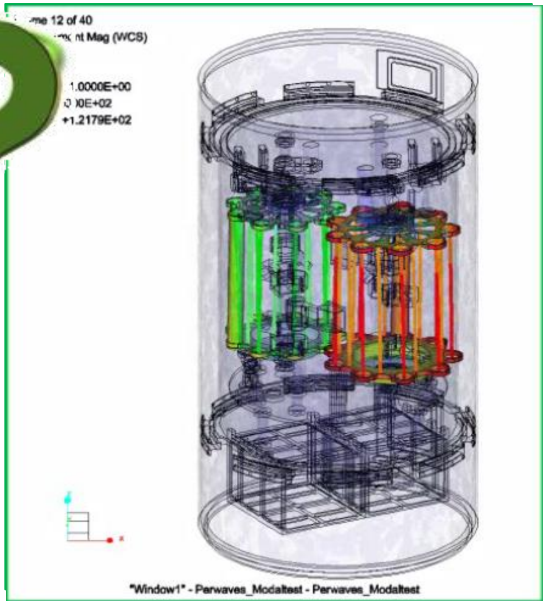
altran

Any Questions?









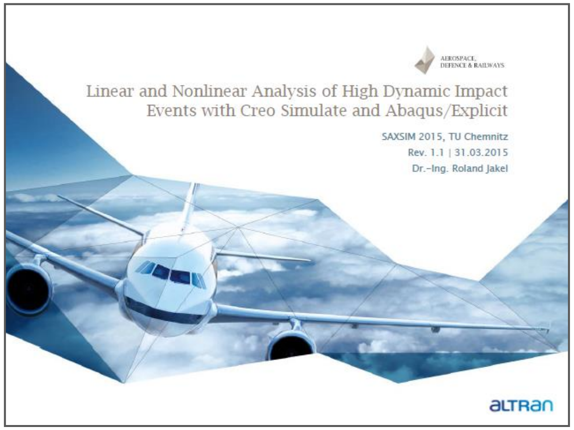
5. References

[1] Shock Response Spectrum – A Primer
J. Edward Alexander, BAE Systems, US Combat Systems
Minneapolis, Minnesota; J. Sound & Vibration, June 2009
http://www.sandv.com/downloads/0906alex.pdf

[2] An Introduction to the Shock Response Spectrum
Tom Irvine, Rev. R, July 29, 2010; www.vibrationdata.com

[3] Berechnung von Schockspektren und praktische Anwendung
der dynamischen Stoßanalyse in Creo Elements / Pro Mechanica
Roland Jakel, Presentation at the 3rd SAXSIM, TU Chemnitz, 19.04.2011, Rev. 1.0
www.saxsim.de, https://www.ptcusercommunity.com/blogs/RSS

[4] Linear and nonlinear Analysis of High Dynamic Impact Events with Creo Simulate and Abaqus/Explicit; Roland Jakel, Presentation at the 7th SAXSIM, TU Chemnitz, 31.03.2015, Rev. 1.1
www.saxsim.de, https://www.ptcusercommunity.com/blogs/RSS



5. References

[5] Zienkiewicz, O.C., „The Finite Element Method“; 3. Edition, McGraw-Hill, 1977, page 500-526

[6] https://femci.gsfc.nasa.gov/random/randomequations.html

[7] Berechnung von Schockantworten in Mathcad; Dr. Wigand Rathman; Universität Erlangen-Nürnberg, 18. Nov. 2010; Vortrag zum 10. Simulationsanwender-treffen im Rahmen der Plant PTC live (www.saxsim.de)

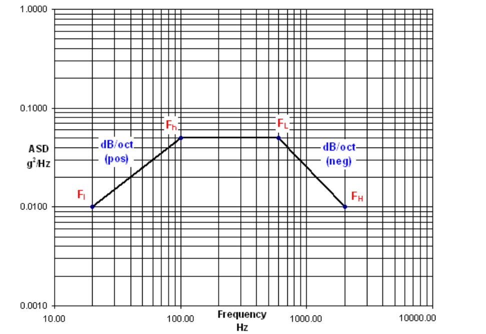
[8] Berechnung von Schockantworten für gemessene Anregungen; Dr. Wigand Rathman; Universität Erlangen-Nürnberg, 18. April 2011; Vortrag zum Mathcad-Workshop im Rahmen des 3. SAXIM (www.saxsim.de)

[9] Zienkiewitz, O and Zhu, J: A simple error estimator and adaptive procedure for practical engineering analysis; Int. J. Numer. Methods Engr. 24 (1987); 337-357

[6]

Random Vibration Specification Magnitude Equations

When performing a random vibration analysis, an input spec is generally given in a form such as the log-log plot in the figure or written in the table below. The problem is what to do with such information. We cannot input these values directly into NASTRAN because it will not accept a slope in dB/Oct. Individual points in G^2/Hz vs. Hz are required. This page details how to get from the input to graphical points needed.



(Note that regardless of popular opinion, G^2/Hz is actually an ***acceleration spectral density (ASD)***, not a power spectral density (PSD). PSD refers to the actual plot generated during testing, which simply reads the *power* output from the accelerometers.)

In tabular form, the input may be given in this form (beginning and ending frequencies are not always necessary if a continuous line is assumed):

Random Input Spec	
3.01 dB/Oct	
100.00 Hz	0.0500 G^2/Hz
600.00 Hz	0.0500 G^2/Hz
-4.02 dB/Oct	

Because no beginning or ending frequencies, F_l and F_H, are given in this table, they must first be decided upon. This is generally project specific. However, the frequency range is usually 20Hz to 2000Hz. From the graph, F_l = 20Hz and F_H = 2000Hz. F_h and F_L are 100Hz and 600Hz, respectively.

First, determine the number of octaves between the two frequencies. Keep in mind that an octave is the doubling of the frequency. So going from 1Hz to 2Hz is an octave and going from 1000Hz to 2000Hz is also an octave. Thus, the number of octaves could be estimated from the graph above. The equation to calculate the exact number is:

$$\#Octaves = \frac{\log(F_H/F_L)}{\log(2)}$$

where F_H is the higher frequency and F_L is the lower frequency.

Second, determine the number of dB by multiplying the number of octaves by the slope, making sure to use the correct sign (positive or negative) for the slope:

$$dB = \left(\frac{dB}{Oct}\right) \times (\#Oct) = 10\log\left(\frac{ASD_H}{ASD_L}\right)$$

The previous equation also shows the definition of dB, where ASD_H and ASD_L are the acceleration spectral densities for the higher and lower frequencies respectively (NOT for the higher and lower ASD values! That is, ASD_L can be greater than ASD_H whereas F_H is *always* greater than F_L).

Finally, solve for the ASD at the desired frequency:

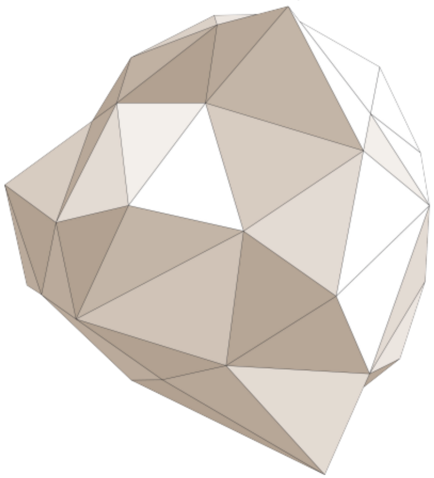
$$\frac{ASD_H}{ASD_L} = 10^{(dB/10)}$$

Or, for those of you who want a more expanded and complete version (where *m* is the slope in dB/Oct):

$$\frac{ASD_H}{ASD_L} = 10^{\left[\left(\frac{m}{10}\right)\left(\frac{\log(F_H/F_L)}{\log(2)}\right)\right]} = \left(\frac{F_H}{F_L}\right)^{\frac{m}{10\log(2)}}$$

And a final, simplified version:

$$\frac{ASD_H}{ASD_L} \approx \left(\frac{F_H}{F_L}\right)^{\frac{m}{3.01}}$$



INNOVATION MAKERS

Dr.-Ing. Roland Jakel
Senior Consultant, Structural Simulation

Phone: +49 (0) 421 / 557 18 4111
Mobile: +49 (0) 173 / 889 04 19
roland.jakel@altran.com

Henning Maue
Global aircraft engineering architect ASDR group
Head of Engineering ASDR Germany
Technical Unit Manager Airframe Structure

Phone: +49 (0) 40 35 96 31 96 122
Mobile: +49 (0) 172 83 63 335
henning.maue@altran.com