# Steady State Dynamic Analysis using Ansys LS-DYNA Software

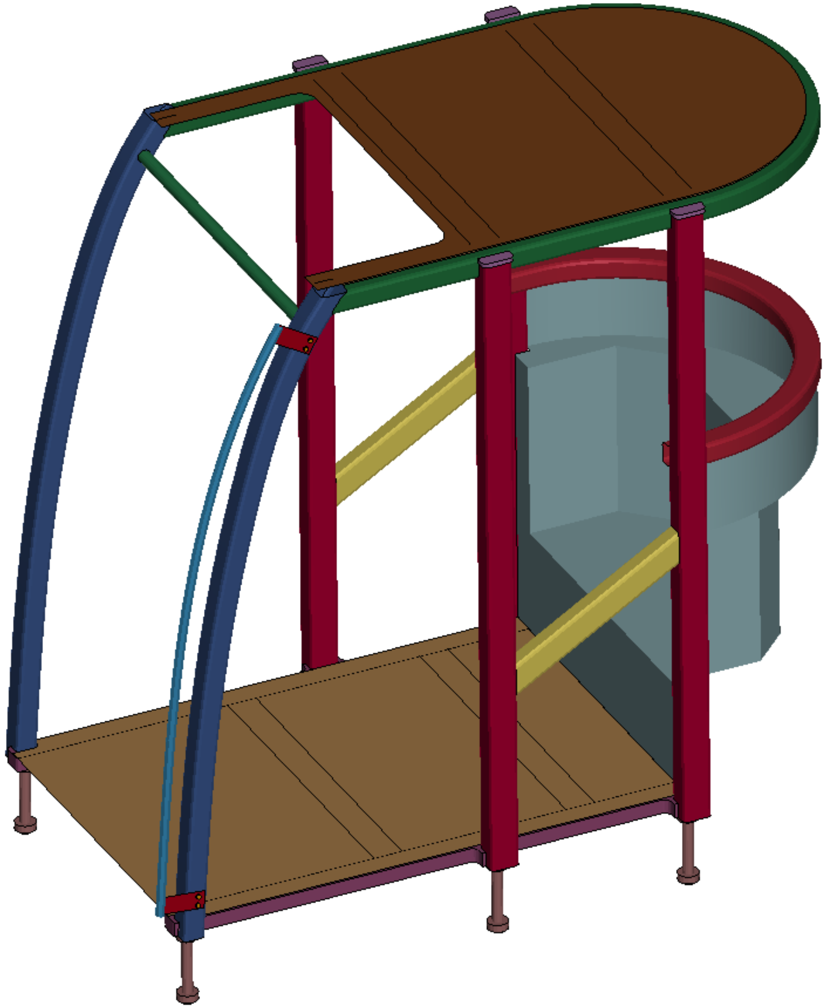
Example

Anders EB Jonsson

2025-12-02

# Overview

- Frequency domain analyses
  - Available resources
    - [LSDYNA NVH / Fatigue](lsdyna.ansys.com) (lsdyna.ansys.com)
    - LSDYNA Implicit
  - Some background
- Example: Vibrational analysis of a Roll-Over Protection Structure (ROPS)
  1. Apply bolt pre-tension
  2. Eigenvalue analysis
  3. Steady State Dynamics analysis
  4. Post-processing of results



# Frequency Domain Analyses

- Performed on a modal basis
  - We will need an Eigenvalue analysis
    - Direct solution option available for Steady State Dynamics

# Eigenvalue Analysis

- ***CONTROL_IMPLICIT_EIGENVALUE**
- For selecting number of eigenmodes, or frequency range, or specifying intermittent eigenvalue extraction (**NEIG**)
- Selecting eigensolver (**EIGMTH**)
- Requesting modal stresses (**MSTRES**)

Select eigensolver method

Number of eigenfrequencies

Specify frequency range

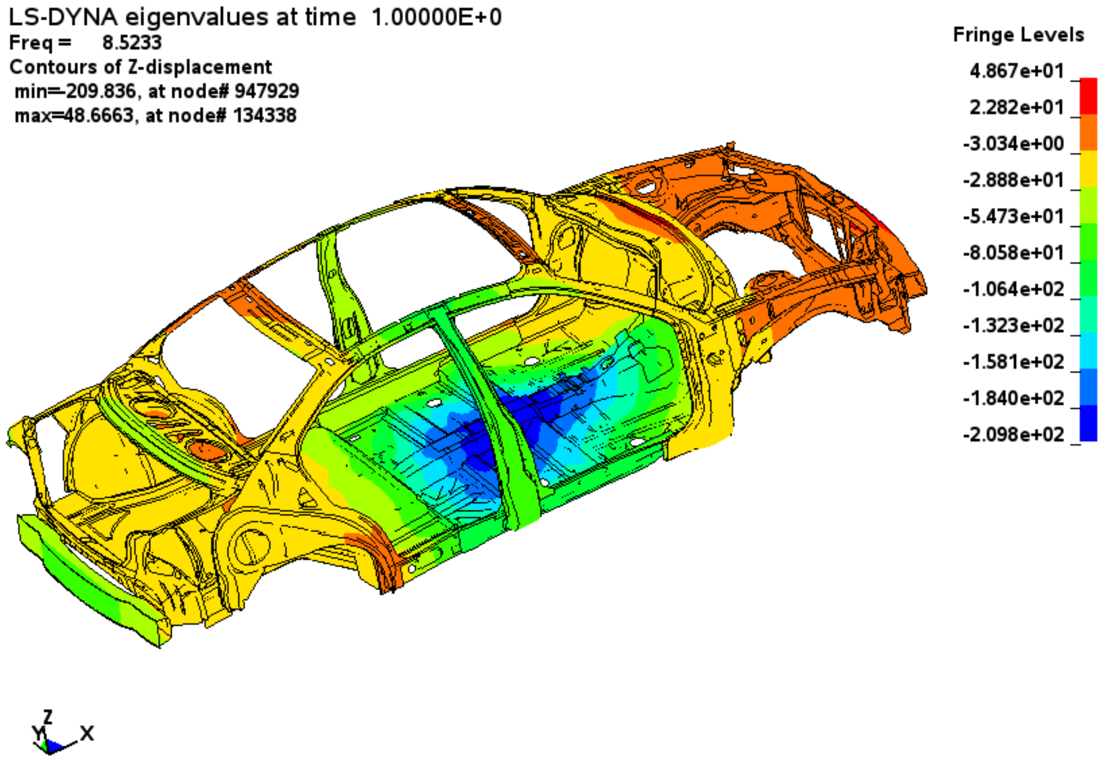
```
*CONTROL_IMPLICIT_EIGENVALUE
$#    neig    center     lflag    lftend     rflag    rhtend    eigmth    shfscl
       100       0.0         1        10         1      1000         2
$#  isolid     ibeam    ishell   itshell    mstres    evdump   mstrscl
         0         0         0         0         1
```

Set mstres=1 to compute modal stresses

# Eigenvalues – Keywords and Output Databases

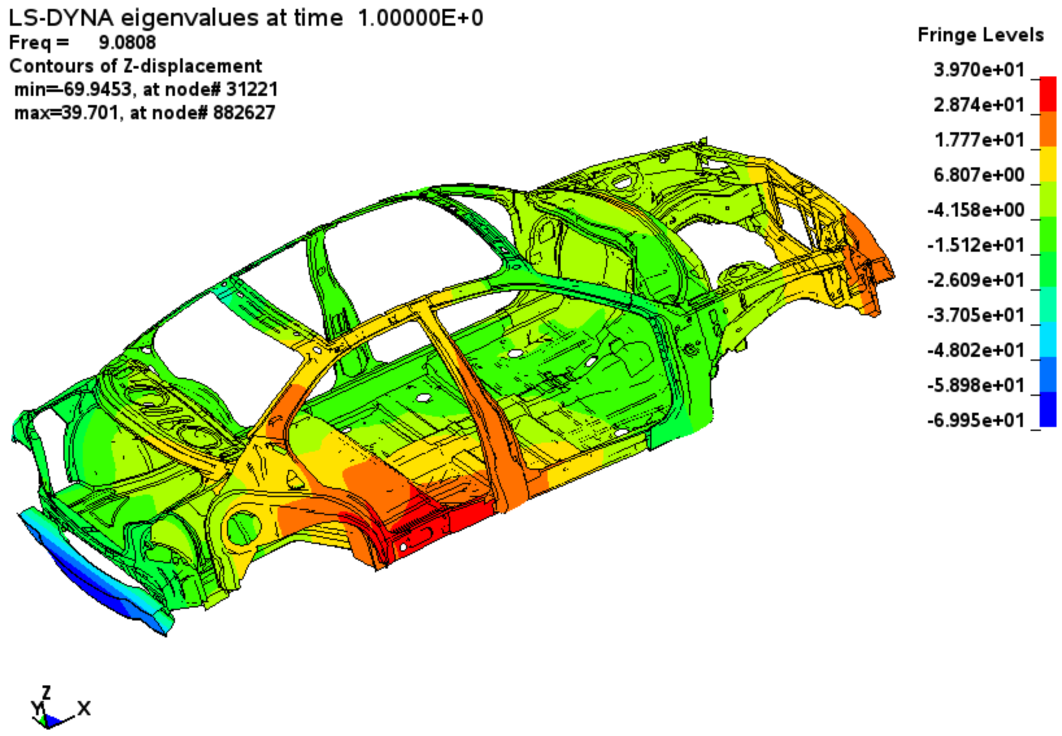
## *Keywords for modal analysis*

**`*CONTROL_IMPLICIT_GENERAL`**
**`*CONTROL_IMPLICIT_EIGENVALUE`**




## *Result databases*

- **`eigout`** **(text file)**
- **`d3eigv`** **(for 3D-Postprocessing)**



# Eigenvalues – the EIGOUT File

- Contains eigenfrequencies, modal participation factors, effective modal mass
- To represent a motion in in a specific direction, enough modes should be extracted so that the accumulated modal mass→ 1 in that direction

```
 IMPLICIT LS-DYNA iteration plot database
                   ls-dyna mpp d DEV DEV_124473_f258a3 date 09/21/2025

 r e s u l t s   o f   e i g e n v a l u e   a n a l y s i s:

 problem time =  1.00000E+00

 (all frequencies de-shifted)

                             |------ frequency -----|
        MODE    EIGENVALUE       RADIANS         CYCLES         PERIOD
           1   1.973715E+04   1.404890E+02   2.235952E+01   4.472369E-02
           2   9.046553E+04   3.007749E+02   4.786981E+01   2.088999E-02
           3   2.533575E+05   5.033463E+02   8.011005E+01   1.248283E-02
           4   2.849982E+05   5.338522E+02   8.496522E+01   1.176952E-02
...
MODAL PARTICIPATION FACTORS

         MODE       X-TRAN          Y-TRAN          Z-TRAN          X-ROT           Y-ROT           Z-ROT

           1   4.305965E-05   6.043872E-05   4.991996E-01  -6.764913E-04   9.761291E-04   1.451026E-07
           2  -9.113646E-05   2.187874E-03  -3.040782E-03  -1.080999E-01  -1.131364E-04  -5.623562E-05
           3   1.550588E-03   1.511422E-03  -2.054481E-04   5.846528E-05   2.321629E-02  -5.568228E-06
           4   2.061018E-03   5.908026E-02   3.520366E-05   5.825058E-04  -3.437073E-03   1.154004E-05
…
 MODAL EFFECTIVE MASS


         MODE            X-TRAN                     Y-TRAN                     Z-TRAN
                 Eff. Mass   Accum. %       Eff. Mass   Accum. %       Eff. Mass   Accum. %

           1    1.854134E-09     0.00%    3.652839E-09     0.00%    2.492003E-01    93.46%
           2    8.305854E-09     0.00%    4.786795E-06     0.00%    9.246356E-06    93.47%
           3    2.404323E-06     0.00%    2.284396E-06     0.00%    4.220891E-08    93.47%
           4    4.247796E-06     0.00%    3.490477E-03     1.31%    1.239298E-09    93.47%
…
```

# Frequency Domain Analyses

- Performed on a modal basis
  - We will need an Eigenvalue analysis
    - Direct solution option available for Steady State Dynamics
- ***FREQUENCY_DOMAIN_SSD_*{OPTION}**
  - Deterministic loading
  - The equivalent to
    - Harmonic Response in MAPDL
    - SOL111 (SOL108) in Nastran
    - *DYNAMIC, STEADY STATE in Abaqus
  - Option for `FATIGUE`
    - Perform fatigue analysis for sine-sweep excitation
  - Option for `ERP`
    - Calculate Equivalent Radiated Power
      - Estimate possible acoustic noise radiation

# Steady State Dynamics

*FREQUENCY_DOMAIN_SSD_*{OPTION}*

- The output requests will determine which frequencies get analyzed
- Keywords
  - *DATABASE_FREQUENCY_BINARY_D3SSD
    - Output in d3ssd, for 3D visualization of deformations and stresses
  - *DATABASE_FREQUENCY_ASCII_*{OPTION}*_SSD
    - Mandatory Option = ELOUT, NODFOR, NODOUT, SECFORC
    - Output in “ascii files” nodout_ssd, elout_ssd, etc. inside binout
    - For curve plotting
  - Specify frequency range
- Additional outputs
  - *DATABASE_FREQUENCY_BINARY_D3FTG
    - Output in d3ftg, for 3D visualization of fatigue results
  - *DATABASE_FREQUENCY_BINARY_D3ERP
    - Output in d3erp, for 3D visualization of equivalent radiated power

# Steady State Dynamics: Keyword

**`*FREQUENCY_DOMAIN_SSD`**

| Card 1 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| Variable | MDMIN | MDMAX | FNMIN | FNMAX | RESTMD | RESTDP | LCFLAG | RELATV |
| Type | I | I | F | F | I | I | I | I |
| Default | 1 | | 0.0 | | 0 | 0 | 0 | 0 |

Card 1: Select modes, restart option …

| Card 2 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| Variable | DAMP | LCDAM | LCTYP | DMPMAS | DMPSTF | DMPFLG | | |
| Type | F | I | I | F | F | I | | |
| Default | 0.0 | 0 | 0 | 0.0 | 0.0 | 0 | | |

Card 2: Damping options

| Card 3 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| Variable | | ISTRESS | MEMORY | NERP | STRTYP | NOUT | NOTYP | NOVA |
| Type | | I | I | I | I | I | I | I |
| Default | | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

Misc. settings

| Card 7 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| Variable | NID | NTYP | DOF | VAD | LC1 | LC2 | SF | VID |
| Type | I | I | I | I | I | I | I | I |
| Default | none | 0 | none | none | none | none | 0 | 0 |

Loadings

**NOTE**! LC1 and LC2 should have the same number of entries, and the same abscissa (frequency) values

Repeat Card 7 if multiple excitation loads are present.

# Keyword Template for Basic SSD Analysis

```
*INCLUDE
control_cards_linear.key
*INCLUDE
database_cards_static.key
*CONTROL_TERMINATION
$   ENDTIM    ENDCYC     DTMIN    ENDENG    ENDMAS
        1.
*CONTROL_IMPLICIT_EIGENVALUE
        50
                                                  1
*FREQUENCY_DOMAIN_SSD
$#   mdmin     mdmax     fnmin     fnmax    restmd    restdp    lcflag    relatv

$#   dampf     lcdam     lctyp    dmpmas    dmpstf    dmpflg
      0.03         0         0       0.0       0.0         0
$#  unused    unused    memory      nerp    strtyp      nout     notyp      nova

$#     nid      ntyp       dof       vad       lc1       lc2        sf       vid
      2041         0         2         0      1001      1002         0         0
$===============================================================================
*DATABASE_HISTORY_NODE
      1043      2041
*DATABASE_FREQUENCY_ASCII_NODOUT_SSD
       10.      500.       490
*DATABASE_FREQUENCY_BINARY_D3SSD
         2
       10.      500.        49
```

Set mstres=1 to compute modal stresses

Blank ➔ Use all computed eigenmodes

Specify damping. In this case constant modal damping, 3 %

For acoustic analyses

Specify loadings

Request history output

Request 3D output for visualization

# Frequency domain analyses

- Performed on a modal basis
  - We will need an Eigenvalue analysis
- `*FREQUENCY_DOMAIN_RANDOM_VIBRATION_`*`{OPTION}`*
  - Non-deterministic loading
  - The equivalent to
    - Random Vibration in MAPDL
    - *RANDOM RESPONSE in Abaqus
  - Option for `FATIGUE`
    - Perform fatigue analysis for random vibration

# Random Vibration: Keyword

- ***FREQUENCY_DOMAIN_RANDOM_VIBRATION**

| Card 1 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| Variable | MDMIN | MDMAX | FNMIN | FNMAX | RESTRT | | RESTRM | |
| Type | I | I | F | F | I | | I | |
| Default | 1 | | 0.0 | | 0 | | 0 | |

| Card 2 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| Variable | DAMP | LCDAM | LCTYP | DMPMAS | DMPSTF | DMPTYP | | |
| Type | F | I | I | F | F | I | | |
| Default | 0.0 | 0 | 0 | 0.0 | 0.0 | 0 | | |

| Card 3 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| Variable | VAFLAG | METHOD | UNIT | UMLT | VAPSD | VARMS | **NAPSD** | **NCPSD** |
| Type | I | I | I | F | I | I | I | I |
| Default | none | 0 | | | | | 1 | 1 |

| Card 4 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| Variable | LDTYP | IPANELU | IPANELV | TEMPER | | LDFLAG | | |
| Type | I | I | I | F | | I | | |
| Default | | | | 0.0 | | 0 | | |

Card 1: Select modes, restart, etc.

Card 2: Damping options. NOTE! It is <u>required to specify some damping</u>; it cannot be left zero for random vibration analysis.

Card 3, 4: Settings, loading type, load curves etc.

# Random Vibration Analysis

`*FREQUENCY_DOMAIN_RANDOM_VIBRATION_{OPTION}`

- The output requests, to some extent, will determine which frequencies get analyzed
- Keywords
  - `*DATABASE_FREQUENCY_BINARY_D3PSD`
    - Output in d3psd, for 3D visualization of PSD values for deformations and stresses per frequency
  - `*DATABASE_FREQUENCY_BINARY_D3RMS`
    - Output in d3rms, for 3D visualization of RMS values for deformations and stresses
  - `*DATABASE_FREQUENCY_ASCII_{OPTION}_PSD`
    - Mandatory Option = `ELOUT`, `NODOUT`
    - Output in “ascii files” nodout_psd, elout_psd, etc. inside binout
    - For curve plotting
  - Specify frequency range
- Additional outputs
  - `*DATABASE_FREQUENCY_BINARY_D3FTG`
    - Output in d3ftg, for 3D visualization of fatigue results

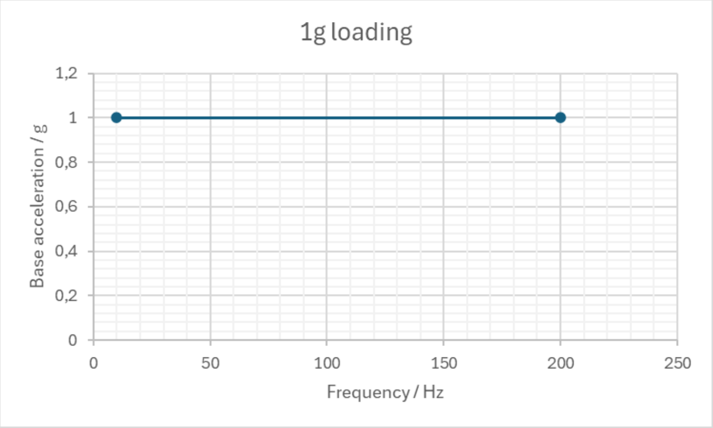

© 2025 Synopsys, Inc.

# Vibrational analysis of a ROPS

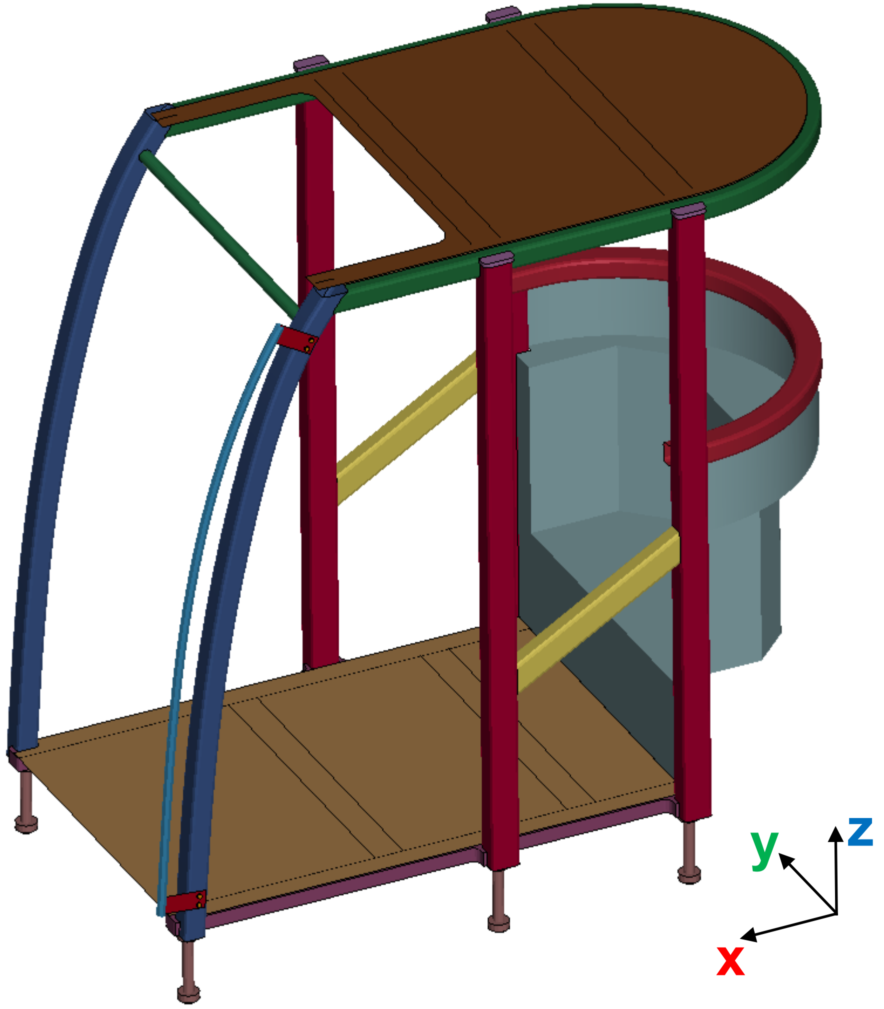
1. Apply bolt pre-tension

2. Eigenvalue analysis

3. Steady State Dynamics analysis
   - Apply 1g sine-sweep base acceleration in the lateral (y) direction, for frequencies between 10 and 200 Hz




4. Post-processing of results



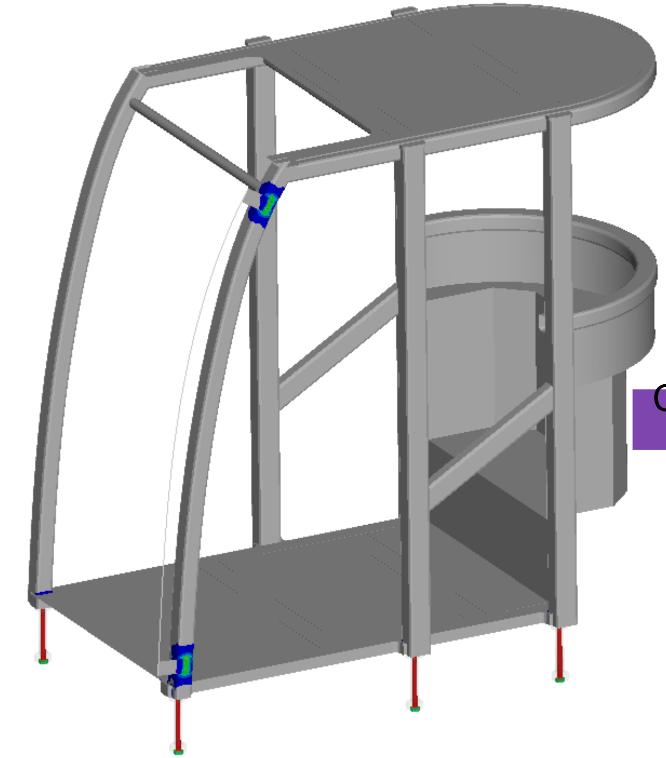


© 2025 Synopsys, Inc.

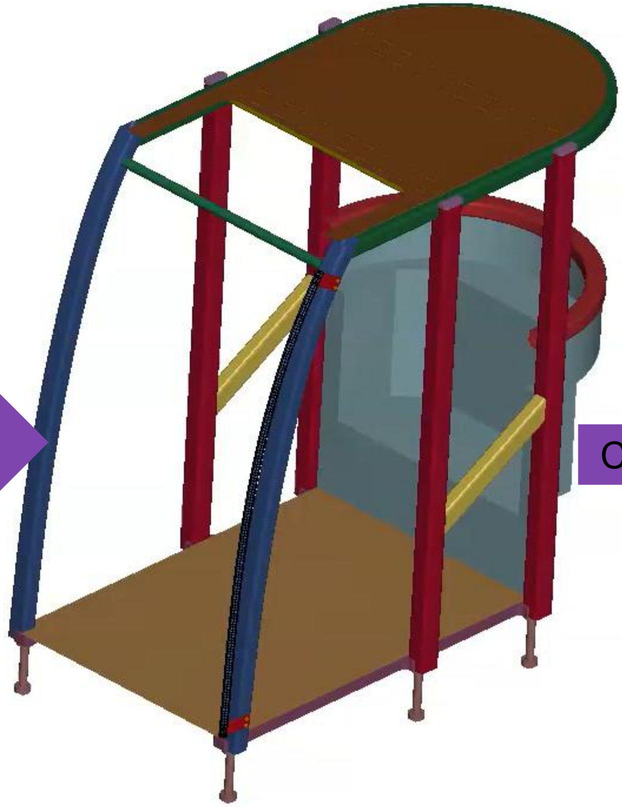
# Solution Procedures

- Two solution procedures are presented
- The Multistage approach (see for example Appendix X of the Ansys LS-DYNA Keyword Manual Vol. I) is presented on pages 18 – 27.
  - This approach divides the analysis into three separate stages, which are then performed in a sequence.
  - When submitting the final keywordfile for solution to the Ansys LS-DYNA software, the word “case” must be present on the execution line.
- The Time Sequence approach is presented on pages 28 – 36.
  - In this approach, the analysis steps are performed in a sequence

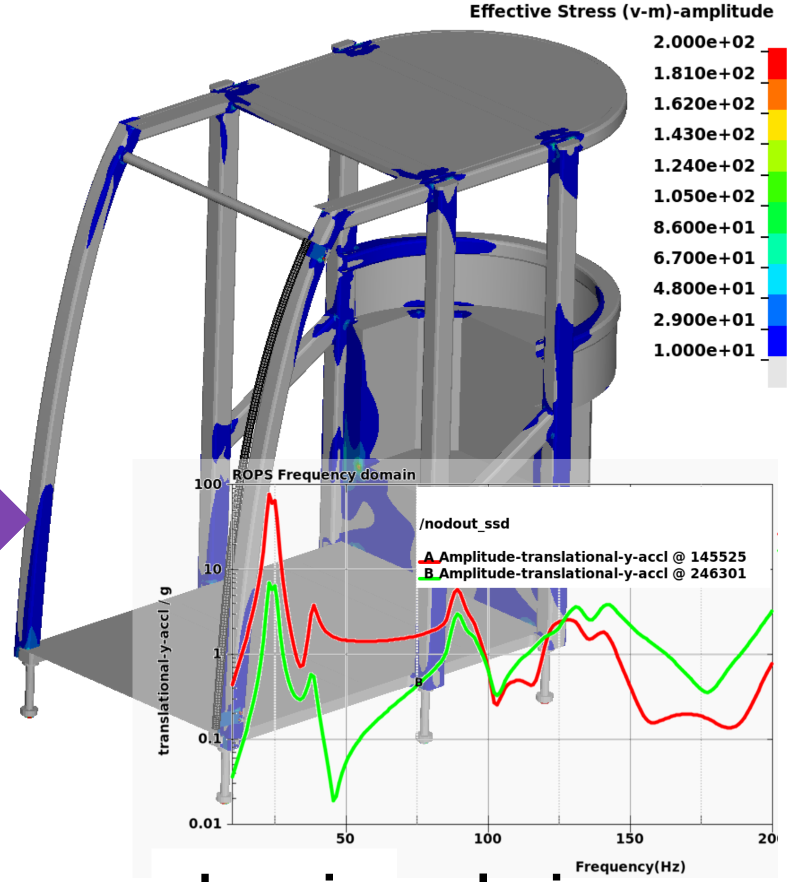
# Workflow: Multistage



Case1.Dynain.lsda



Case2.d3eigv




## Non-linear preload

```
*CONTROL_IMPLICIT_DYNAMICS
*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR_ID
*INITIAL_STRESS_SECTION
…
```

## Eigenvalue analysis

```
*CONTROL_IMPLICIT_EIGENVALUE
…
```

## Frequency domain analysis

```
*FREQUNCY_DOMAIN_PATH_NOJOBID
*FREQUNCY_DOMAIN_SSD
*DATABASE_FREQEUNCY_D3SSD
*DATABASE_FREQEUNCY_ASCII_NODOUT_SSD

…
```


© 2025 Synopsys, Inc.

# Overview of Files

- The ROPS Geometry, including material definitions
  - Rops_geo-001.key, the complete geometry
    - Rops_geo-001.key.nodel – only nodes and elements
    - Rops_geo-001.key.partsec – parts and sections, set definitions, materials, etc.
  - Required part sets and node sets for defining contacts, loadings, outputs etc..  are provided

- Control cards
  - control_cards_nonlin.key
  - database_cards_static.key

- Template
  - template_multistage.key

- Solution
  - run_multistage.key

# Multistage Analysis

## File Layout

### General Layout

Common
- Geometry
- Control cards
- Contacts
- Etc.

Cases
- One per stage
- Define loading
- Final state of previous stage as initial condition (dynain.lsda)

### In this case

Common
- Geometry
- Control cards
- Contacts
- Boundary conditions

Stage 1 – Boltpre
Original nodes/elements
*INITIAL_STRESS_SECTION

Stage 2 – Eigenvalues
case1.dynain.lsda
Define eigenvalue analysis

Stage 3 – Steady state dynamics
case1.dynain.lsda
case2.d3eigv
*FREQUENCY_DOMAIN_SSD

# Procedure for Multistage Analysis

- Start from the file template_multistage.key

- Use
  - a text editor,
  - LS-PrePost,
  - Or your personal favorite pre-processor

to add required keywords and complete the setup

- Since the bolt pre-tensioning is a nonlinear analysis, the relevant control cards are already defined

```
*INCLUDE
../control_cards_nonlin.key
*INCLUDE
../database_cards_static.key
```


© 2025 Synopsys, Inc.

# Stage 1: Bolt pre-tensioning

## Common Contacts and Boundary Conditions

- Modify/add the Common block, before (outside) `*CASE_BEGIN_1`

- Geometry
  - Include common geometry

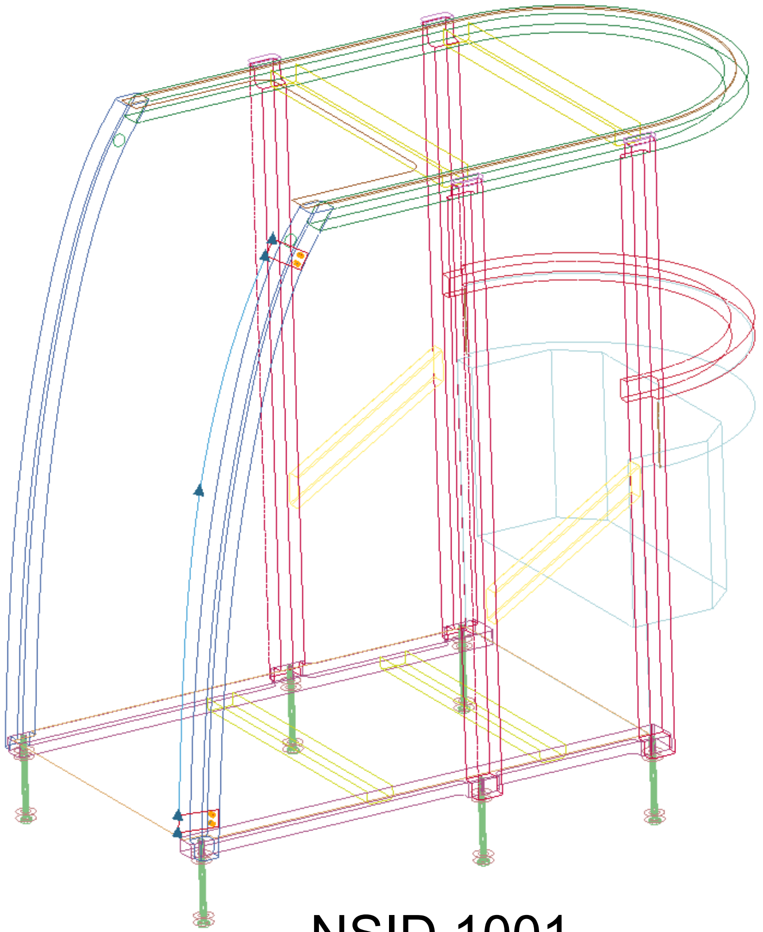
- Contacts
  - One single surface Mortar, for the structure
    - Use Part set ID 10111
    - Specify a coefficient of friction of 0.1
  - One global tied contact
    - Use `*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET_ID` in this case
      - Node Set ID 10101 to Part Set ID 10102
  - One extra tied contact
    - Node Set ID 10201 to Part ID 6

- Boundary conditions
  - Constrain the fundament (PID 19) using zero displacement curve ID 1000 and `*BOUNDARY_PRESCRIBED_MOTION_RIGID` (×6)

# Stage 1: Bolt pre-tensioning

## Control, Loading and Output

- Modify/add the Stege 1 Block, Between `*CASE_BEGIN_1` and `*CASE_END_1`
- Control
  - Activate implicit dynamics, and phase it out between t = 0.2 and t=0.3
  - Keyword: `*CONTROL_IMPLICIT_DYNAMICS`
- Bolt pre-tension
  - There are 10 bolts (screws) in the model
    - 6 for the Fundament
    - 4 for the grab handle
  - Create Load curve ID 1002, as a linear ramp from (0,0) to (1,600)
  - Use keyword `*INITIAL_STRESS_SECTION`
    - 4 for the grab handle: Cross Section IDs 1 - 4, Part set ID 111
    - 6 for the Fundament: Cross Section IDs 5 – 10, Part set ID 110
    - Create one Initial Stress ID per Cross Section ID
    - Use Load Curve ID 1002 to apply a cross-sectional normal stress of 600 MPa
- Request history output for Node Set ID 1001
  - Some nodes on the grab handle



NSID 1001


© 2025 Synopsys, Inc.

# Stage 2: Eigenvalue analysis

## Eigenvalues

- Modify/add the Stege 2 Block, Between `*CASE_BEGIN_2` and `*CASE_END_2`

- Eigenvalues
  - Use keyword `*CONTROL_IMPLICIT_EIGENVALUE`
    - Request to find all eigenfrequencies between 1 and 400 Hz
    - (Hint: Set NEIG to something high, for example 1000)

- Stress stiffening effect
  - Use keyword `*CONTROL_IMPLICIT_EIGENVALUE`
    - Set *IGS* = 1
  - To fully account for the stress stiffening effect due to the pre-tensioning

# Stage 3: Steady State Dynamics

## Frequency Domain Analysis

- Modify/add the Stege 3 Block, Between `*CASE_BEGIN_3` and `*CASE_END_3`

- Steady state dynamics
  - Use keyword `*FREQUENCY_DOMAIN_SSD`
    - Select modes and set RESTMD = 1 (on Card 1)
    - Define 3 % modal damping (on Card 2)
    - Define base acceleration in the global y-direction, using Load curve IDs 3001, 3002, and a scale factor corresponding to 1g.
  - To specify d3eigv – file, use the keyword `*FREQUENCY_DOMAIN_PATH_NOJOBID`

- Output
  - Specify frequencies and number of outputs for 3D visualization (d3ssd) and curve plotting (binout)
  - Keywords: `*DATABASE_FREQUENCY_BINARY_D3SSD`, `*DATABASE_FREQUENCY_ASCII_NODOUT_D3SSD`

- Job submission
  - The word “case” must be present on the execution line, when submitting the keyword file to the Ansys LS-DYNA software

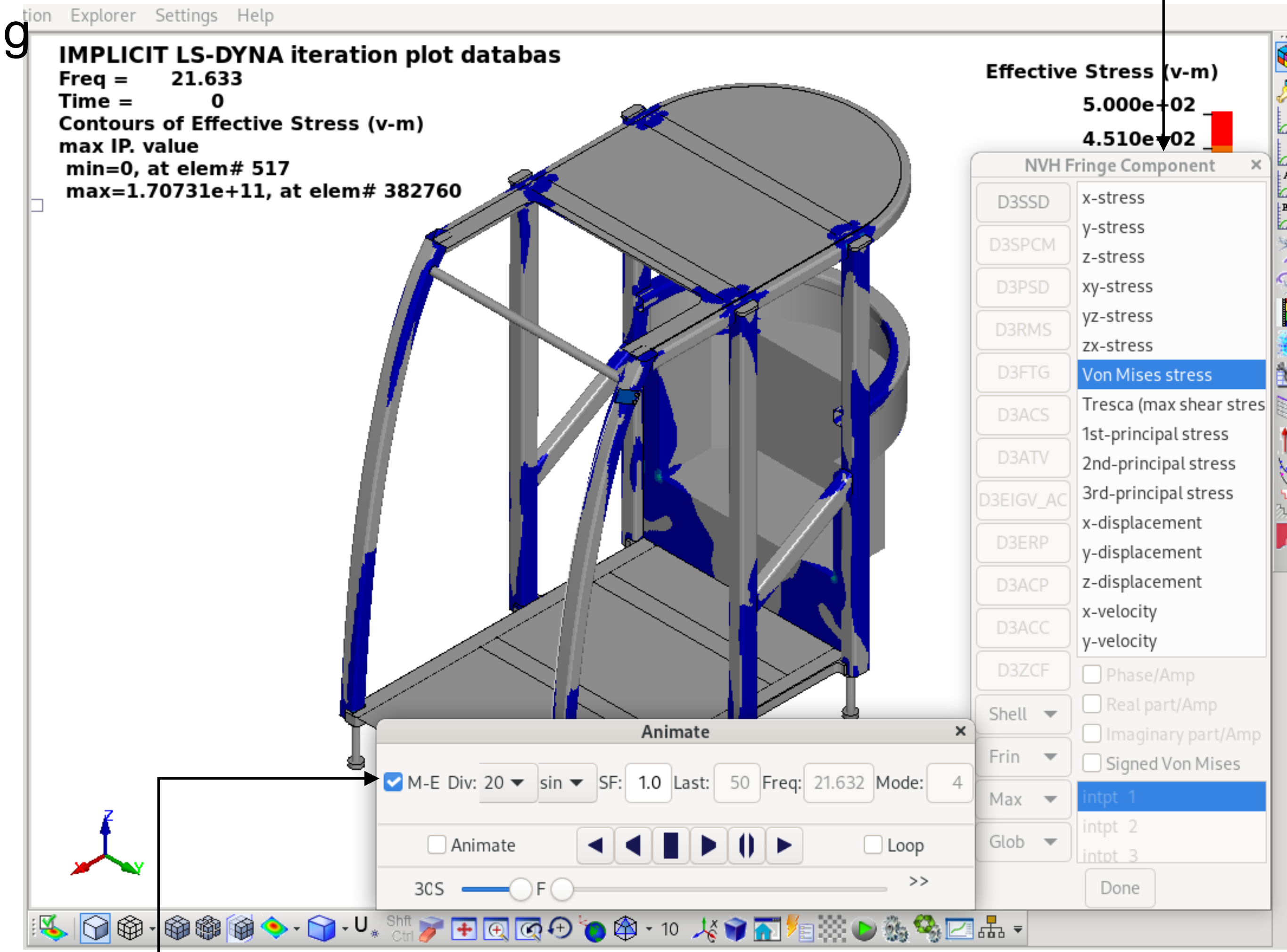
 © 2025 Synopsys, Inc. 

# Postprocessing of results

## Multistage Analysis

NVH Fringe component

- The results of the nonlinear bolt pre-tensioning analysis are found in case1.d3plot for 3D visualization, and case1.binout0000 for curve plotting
- The mode shapes (for 3D visualization) are found in case2.d3eigv, and the eigenfrequencies are also printed in case2.eigout
- The results of the Steady State Dynamics analysis are found in the case3.d3ssd for 3D visualization, and case3.binout000 for curve plotting
- Open in LS-PrePost, or your favorite postprocessor

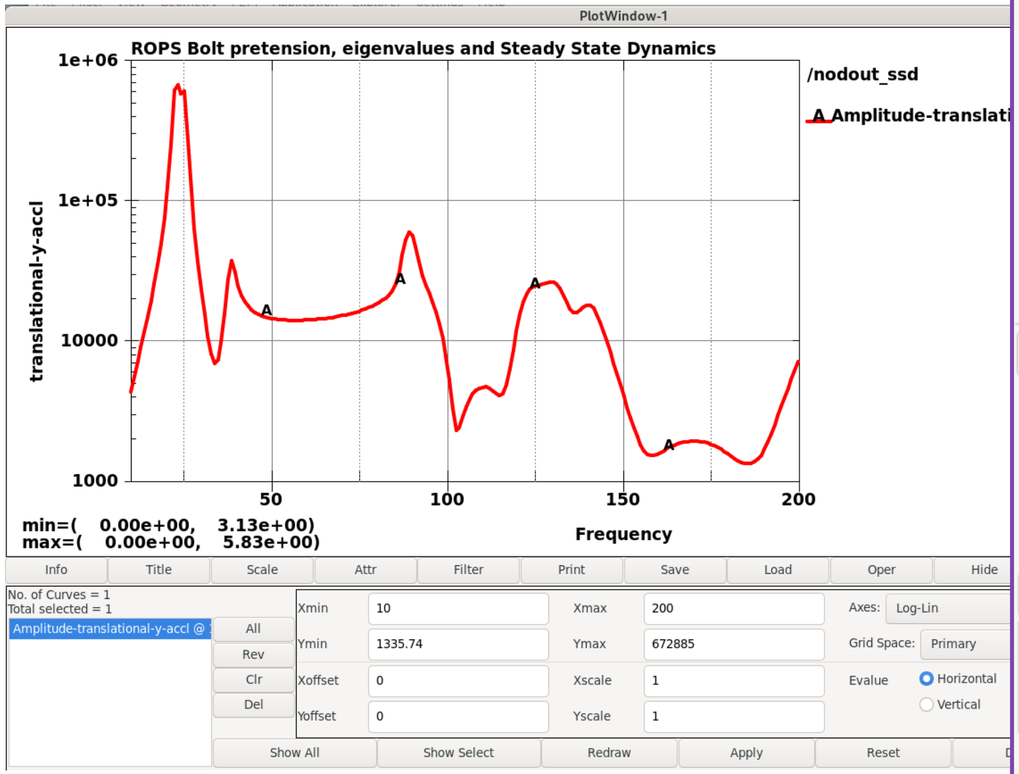


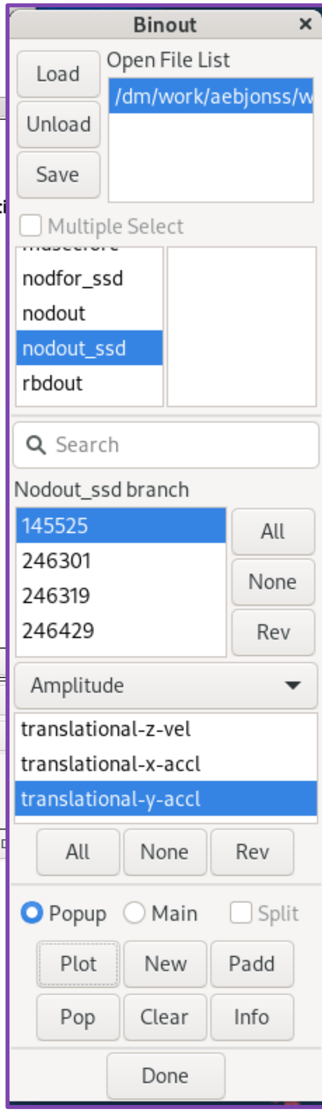

Check "M-E" for modal expansion

# Postprocessing of results

## Multistage Analysis

- The results of the nonlinear bolt pre-tensioning analysis are found in case1.d3plot for 3D visualization, and case1.binout0000 for curve plotting
- The mode shapes (for 3D visualization) are found in case2.d3eigv, and the eigenfrequencies are also printed in case2.eigout
- The results of the Steady State Dynamics analysis are found in the case3.d3ssd for 3D analysis, and case3.binout000 for curve plotting
- Open in LS-PrePost, or your favorite postprocessor




Nodal history plotting from nodout_ssd inside case3.binout


© 2025 Synopsys, Inc.

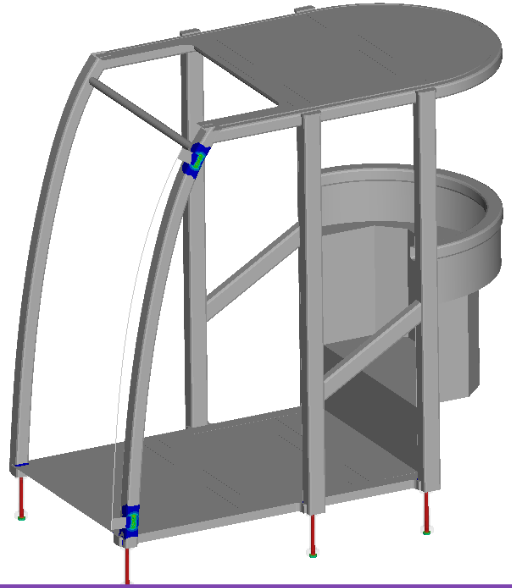
# Workflow: Sequential

Time sequence

## Non-linear preload

```
*CONTROL_IMPLICIT_DYNAMICS
*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR_ID
*INITIAL_STRESS_SECTION
…
```

d3plot



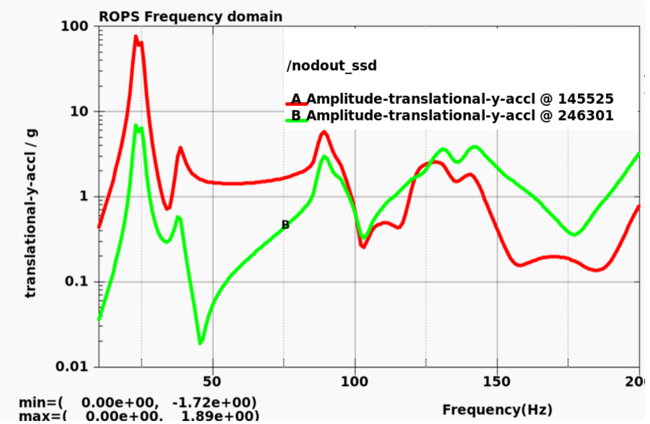
## Intermittent Eigenvalue Analysis

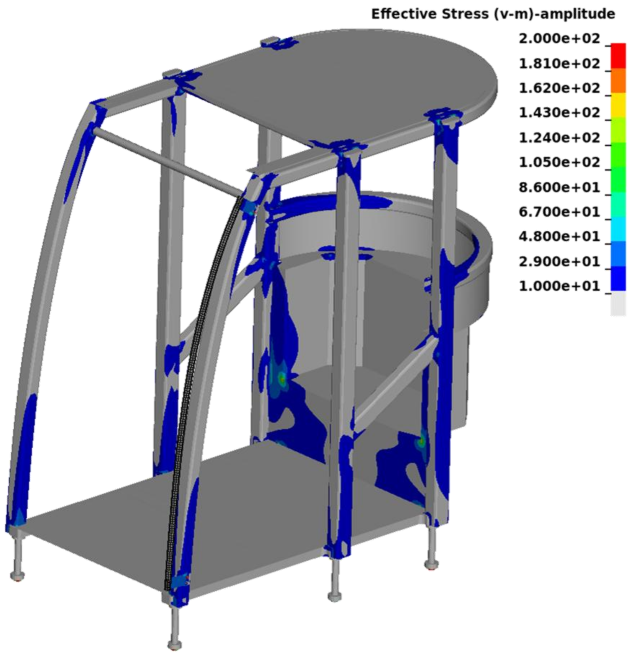
```
*CONTROL_IMPLICIT_EIGENVALUE
NEIG=-1*LCID
…
```

d3eigv1

## Frequency domain analysis

```
*FREQUNECY_DOMAIN_SSD
*DATABASE_FREQEUNCY_D3SSD
*DATABASE_FREQEUNCY_ASCII_NODOOUT_SSD
…
```





© 2025 Synopsys, Inc.

# Procedure for Sequential Analysis

- Start from the file template_sequence.key
- Use
  - a text editor,
  - LS-PrePost,
  - Or your personal favorite pre-processor

to add required keywords and complete the setup

- Since the bolt pre-tensioning is a nonlinear analysis, the relevant control cards are already defined

```
*INCLUDE
../control_cards_nonlin.key
*INCLUDE
../database_cards_static.key
```

# Bolt pre-tensioning

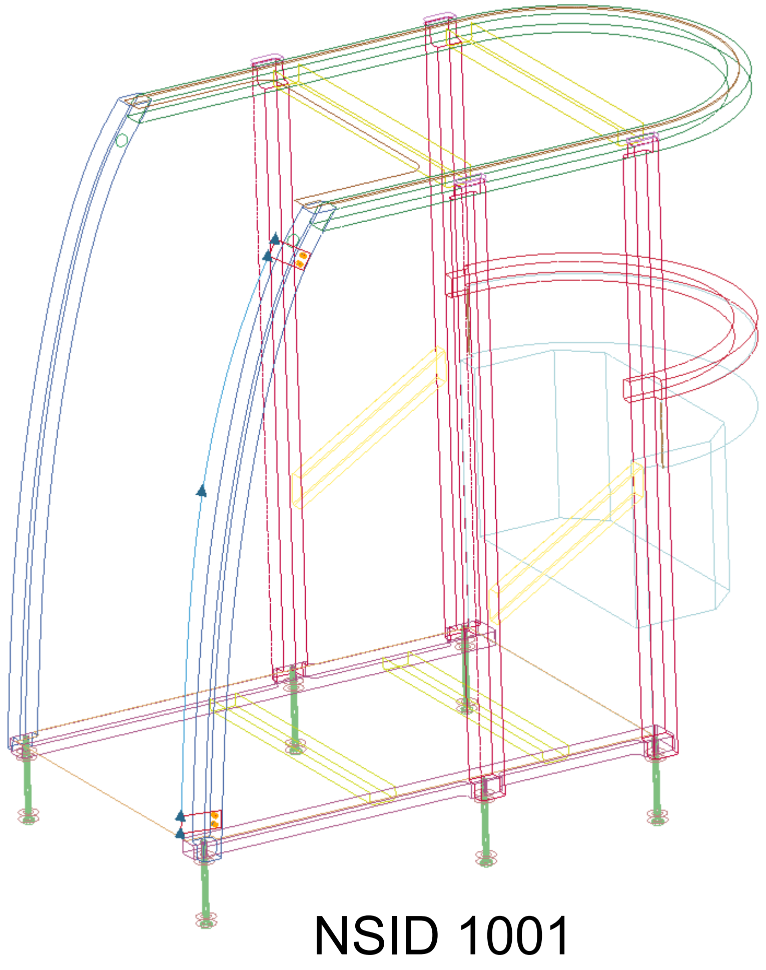
## Contacts and Boundary Conditions

- Geometry
  - Include the complete geometry
- Contacts
  - One single surface Mortar, for the structure
    - Use Part set ID 10111
    - Specify a coefficient of friction of 0.1
  - One global tied contact
    - Use `*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET_ID` in this case
      - Node Set ID 10101 to Part Set ID 10102
  - One extra tied contact
    - Node Set ID 10201 to Part ID 6
- Boundary conditions
  - Constrain the fundament (PID 19) using zero displacement curve ID 1000 and `*BOUNDARY_PRESCRIBED_MOTION_RIGID` (×6)

# Bolt pre-tensioning

## Control, Loading and Output

- Control
  - Activate implicit dynamics, and phase it out between t = 0.2 and t=0.3
  - Keyword: `*CONTROL_IMPLICIT_DYNAMICS`
- Bolt pre-tension
  - There are 10 bolts (screws) in the model
    - 6 for the Fundament
    - 4 for the grab handle
  - Create Load curve ID 1002, as a linear ramp from (0,0) to (1,600)
  - Use keyword `*INITIAL_STRESS_SECTION`
    - 4 for the grab handle: Cross Section IDs 1 - 4, Part set ID 111
    - 6 for the Fundament: Cross Section IDs 5 – 10, Part set ID 110
    - Create one Initial Stress ID per Cross Section ID
    - Use Load Curve ID 1002 to apply a cross-sectional normal stress of 600 MPa
- Request history output for Node Set ID 1001
  - Some nodes on the grab handle





© 2025 Synopsys, Inc.

# Eigenvalue analysis

## Eigenvalues

- Eigenvalues
  - Use keyword `*CONTROL_IMPLICIT_EIGENVALUE`
    - Request to find all eigenfrequencies between 1 and 400 Hz
    - Set NEIG = -701 to use Curve ID 701 for requesting eigenvalues at time t = 1
- Stress stiffening effect
  - Use keyword `*CONTROL_IMPLICIT_EIGENVALUE`
    - Set *IGS* = 1
  - To fully account for the stress stiffening effect due to the pre-tensioning

# Steady State Dynamics

## Frequency Domain Analysis

- Steady state dynamics
  - Use keyword `*FREQUENCY_DOMAIN_SSD`
    - Define 3 % modal damping (on Card 2)
    - Define base acceleration in the global y-direction, using Load curve IDs 3001, 3002, and a scale factor corresponding to 1g.
- Output
  - Specify frequencies and number of outputs for 3D visualization (d3ssd) and curve plotting (binout)
  - Keywords: `*DATABASE_FREQUENCY_BINARY_D3SSD`, `*DATABASE_FREQUENCY_ASCII_NODOUT_D3SSD`

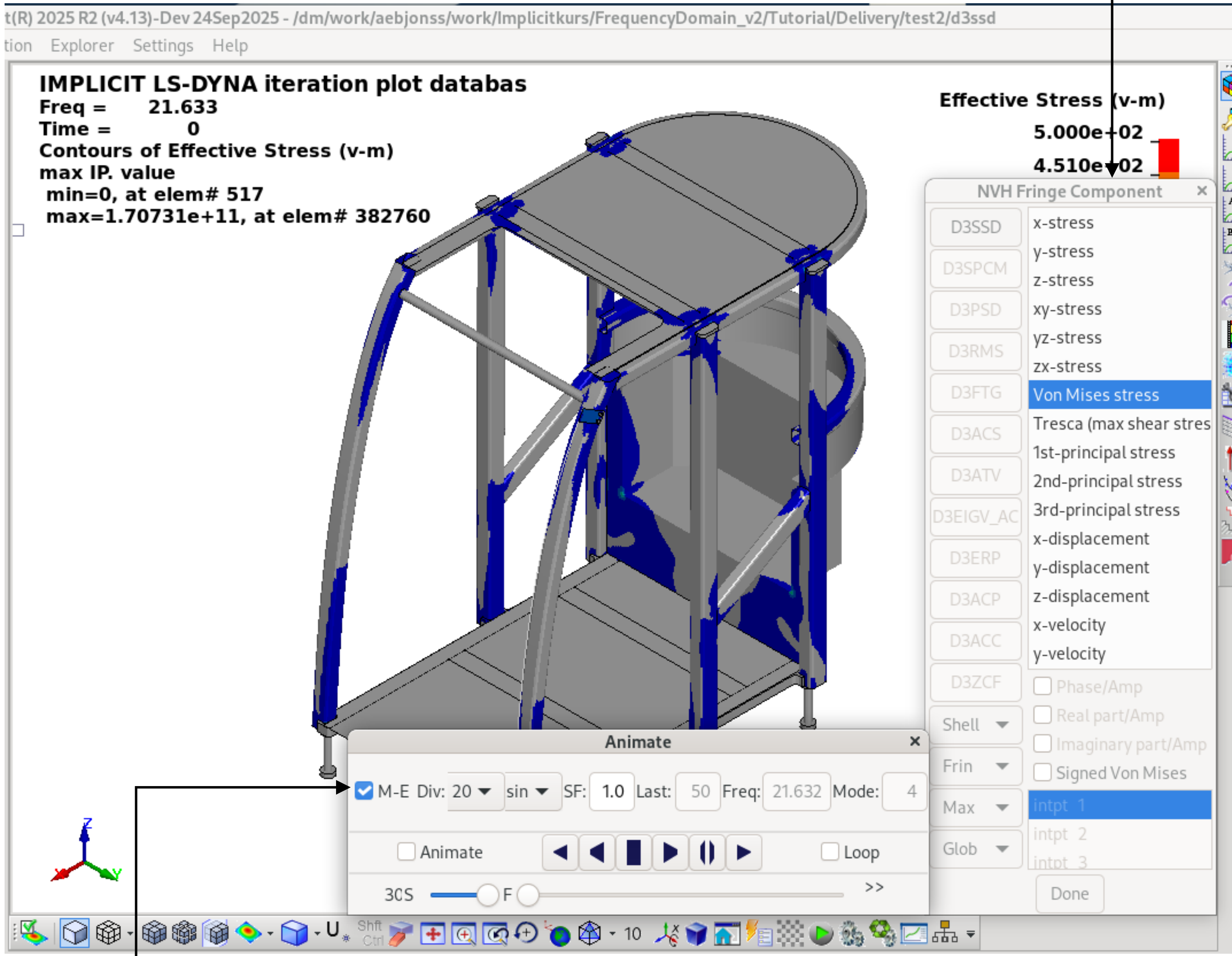

© 2025 Synopsys, Inc.

# Postprocessing of results

## Sequential Analysis

- The results of the nonlinear bolt pre-tensioning analysis are found in d3plot for 3D visualization, and binout0000 for curve plotting
- The mode shapes (for 3D visualization) are found in d3eigv1, and the eigenfrequencies are also printed in eigout1
- The results of the Steady State Dynamics analysis are found in the d3ssd for 3D visualization, and binout000 for curve plotting
- Open in LS-PrePost, or your favorite postprocessor

NVH Fringe component

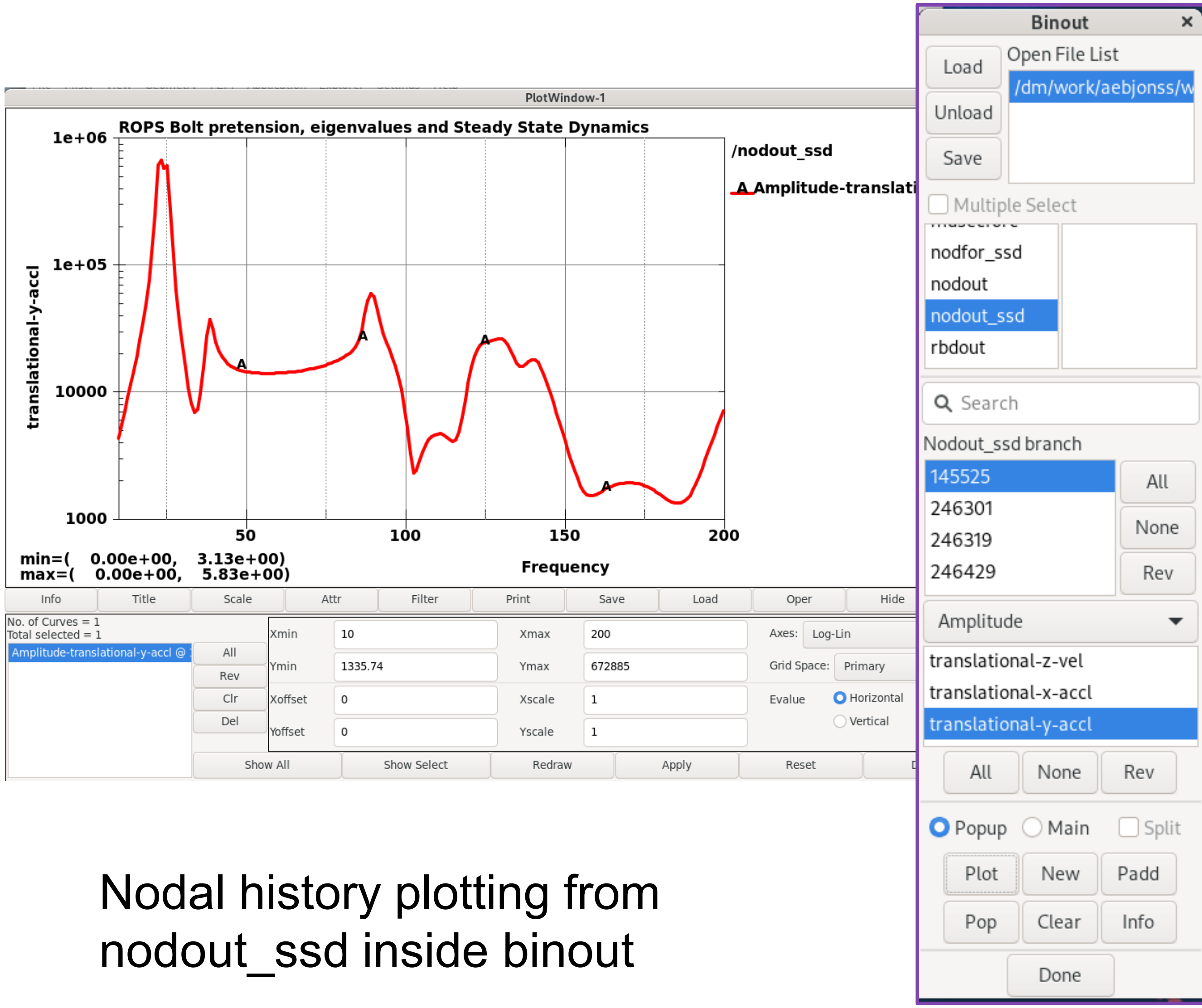



Check “M-E” for modal expansion

# Postprocessing of results

## Sequential Analysis

- The results of the nonlinear bolt pre-tensioning analysis are found in d3plot for 3D visualization, and binout0000 for curve plotting
- The mode shapes (for 3D visualization) are found in d3eigv1, and the eigenfrequencies are also printed in eigout1
- The results of the Steady State Dynamics analysis are found in the d3ssd for 3D visualization, and binout000 for curve plotting
- Open in LS-PrePost, or your favorite postprocessor




Nodal history plotting from nodout_ssd inside binout



© 2025 Synopsys, Inc.

# Thank you

Synopsys Confidential Information