

FrontISTR Tutorial Guide

FrontISTR Commons

October 13, 2020

Contents

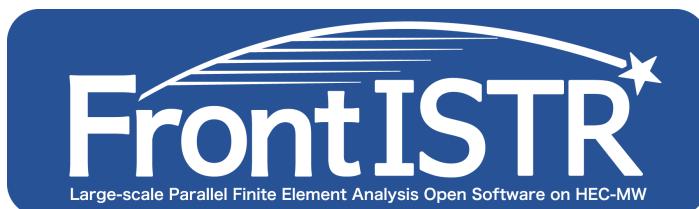
1	FrontISTR Tutorial Guide	3
1.1	Manuals	3
1.2	List of description on this manual	3
1.3	Linear Static Analysis (Elasticity)	4
1.3.1	Analysis target	4
1.3.2	Analysis target	5
1.3.3	Mesh data	6
1.3.4	Analysis procedure	7
1.3.5	Analysis results	8
1.4	Linear Static Analysis (Elasticity, Parallel)	9
1.4.1	Analysis target	9
1.4.2	Analysis	10
1.4.3	Analysis Results	13
1.5	Non-Linear Static Analysis (Hyperelasticity, Part 1)	13
1.5.1	Analysis target	13
1.5.2	Analysis content	14
1.5.3	Analysis results	15
1.6	Non-linear Static Analysis (Hyperelasticity, Part 2)	16
1.6.1	Analysis target	16
1.6.2	Analysis content	17
1.6.3	Analysis procedure	17
1.6.4	Analysis results	17
1.7	Non-linear Static Analysis (Elastoplastic, Part 1)	18
1.7.1	Analysis target	18
1.7.2	Analysis content	19
1.7.3	Analysis procedure	20
1.7.4	Analysis results	20
1.8	Non-linear Static Analysis (Elastoplastic, Part 2)	21
1.8.1	Analysis target	21
1.8.2	Analysis content	22
1.8.3	Analysis procedure	22
1.8.4	Analysis results	23
1.9	Non-linear Static Analysis (Elastoplastic, Part 2)	23
1.9.1	Analysis target	23
1.9.2	Analysis content	24
1.9.3	Analysis procedure	25
1.9.4	Analysis results	25
1.10	Non-Linear Static Analysis (Viscoelasticity)	26
1.10.1	Analysis target	26
1.10.2	Analysis content	26
1.10.3	Analysis procedure	27
1.10.4	Analysis results	27
1.11	Non-Linear Static Analysis (Creep)	28
1.11.1	Analysis target	28
1.11.2	Analysis content	29

1.11.3 Analysis procedure	29
1.11.4 Analysis results	29
1.12 Contact Analysis (Part 1)	30
1.12.1 Analysis target	30
1.12.2 Analysis content	31
1.12.3 Analysis procedure	32
1.12.4 Analysis results	32
1.13 Contact Analysis (Part 2)	33
1.13.1 Analysis target	33
1.13.2 Analysis content	34
1.13.3 Analysis procedure	35
1.13.4 Analysis results	35
1.14 Contact Analysis (Part 3)	36
1.14.1 Analysis target	36
1.14.2 Analysis contents	37
1.14.3 Analysis procedure	38
1.14.4 Analysis results	38
1.15 Linear Dynamic Analysis	38
1.15.1 Analysis target	39
1.15.2 Analysis contents	40
1.15.3 Analysis procedure	40
1.15.4 Analysis results	40
1.16 Non-Linear Dynamic Analysis	41
1.16.1 Analysis target	41
1.16.2 Analysis content	41
1.16.3 Analysis procedure	42
1.16.4 Analysis results	42
1.17 Non-Linear Contact Dynamic Analysis	43
1.17.1 Analysis target	43
1.17.2 Analysis content	44
1.17.3 Analysis Results	45
1.18 Eigenvalue Analysis	46
1.18.1 Analysis target	46
1.18.2 Analysis content	46
1.18.3 Analysis procedure	47
1.18.4 Analysis results	47
1.19 Heat Conduction Analysis	48
1.19.1 Analysis target	48
1.19.2 Analysis content	49
1.19.3 Analysis procedure	49
1.19.4 Analysis results	50
1.20 Frequency Response Analysis	50
1.20.1 Analysis target	50
1.20.2 Analysis content	51
1.20.3 Analysis procedure	52
1.20.4 Analysis results	52
1.21 Verification by Simple-Shaped Model	53
1.21.1 Elastic static analysis	53
1.21.2 Non-linear static analysis	59
1.21.3 Contact analysis (1)	61
1.21.4 Contact analysis (2): Hertz contact problem	62
1.21.5 (3) Eigenvalue analysis	64
1.21.6 (4) Heat conduction analysis	66
1.21.7 (5) Linear dynamic analysis	70
1.21.8 (6) Frequency response analysis	74
1.22 Actual Model Examples for Elastic Static Analysis	76
1.22.1 Analysis Model	76
1.22.2 Analysis results	78
1.23 Actual Model Examples for Eigenvalue Analysis	83

1.23.1 Analysis model	83
1.23.2 Analysis Results	84
1.24 Actual Model Examples for Heat Conduction Analysis	87
1.24.1 Analysis model	87
1.24.2 Analysis Results	89
1.25 Actual Model Examples for Linear Dynamic Analysis	90
1.25.1 Analysis model	90
1.25.2 Analysis results	91

1 FrontISTR Tutorial Guide

This software is the outcome of “Research and Development of Innovative Simulation Software” project supported by Research and Development for Next-generation Information Technology of Ministry of Education, Culture, Sports, Science and Technology. We assume that you agree with our license agreement of “MIT License” by using this software either for the purpose of profit-making business or for free of charge. This software is protected by the copyright law and the other related laws, regarding unspecified issues in our license agreement and contact, or condition without either license agreement or contact.



Item	Content
Name of Software	FrontISTR
Version	5.1
License	MIT License
Corresponding Clerks	FrontISTR Commons2-11-16 Yayoi, Bunkyo-ku, Tokyo/o Institute of Engineering Innovation, School of EngineeringE-mail support@frontistr.com

1.1 Manuals

- Introduction
- How to install
- Theory
- User's manual
- Tutorial
- FAQ

This manual describes the analysis implementation guidelines using a large-scale structural analysis program using the finite element method FrontISTR based on examples.

1.2 List of description on this manual

- PDF
- Tutorials:
 - Static Analysis (Elasticity)
 - Static Analysis (Elasticity, Parallel)
 - Static Analysis (Hyperelasticity Part 1)
 - Static Analysis (Hyperelasticity Part 2)
 - Static Analysis (Elastoplasticity Part 1)
 - Static Analysis (Elastoplasticity Part 2)
 - Static Analysis (Viscoelasticity)
 - Static Analysis (Creep)

- Contact Analysis (Part 1)
 - Contact Analysis (Part 2)
 - Contact Analysis (Part 3)
 - Linear Dynamic Analysis
 - Nonlinear Dynamic Analysis
 - Nonlinear Contact Dynamic Analysis
 - Eigenvalue Analysis
 - Heat Conduction Analysis
 - Frequency Response Analysis
- Example Verification:
 - Verification by Simple Geometric Model
 - Example of Actual Model for Elastic Static Analysis
 - Example of Actual Model for Eigenvalue Analysis
 - Example of Actual Model for Heat Conduction Analysis
 - Example of Actual Model for Linear Dynamic Analysis

1.3 Linear Static Analysis (Elasticity)

For this analysis, the data in tutorial/01_elastic_hinge is used.

1.3.1 Analysis target

The analysis object is a hinge part, and the geometry is shown in Figure 4.1.1 and the mesh data in Figure 4.1.2.

Item	Description	Notes	Reference
Type of analysis	Linear static analysis	!SOLUTION,TYPE=STATIC	
Number of nodes	84,056		
Number of elements	49,871		
Element type	10-node tetrahedron quadratic element	!ELEMENT,TYPE=342	Element Library
Material Name	STEEL	!ELASTIC	Material Data
Material property	ELASTIC		
Boundary condition	Restrained, Concentrated force		
Matrix solution	CG/SSOR	!SOLVER,METHOD=CG,PRE-COND=1	

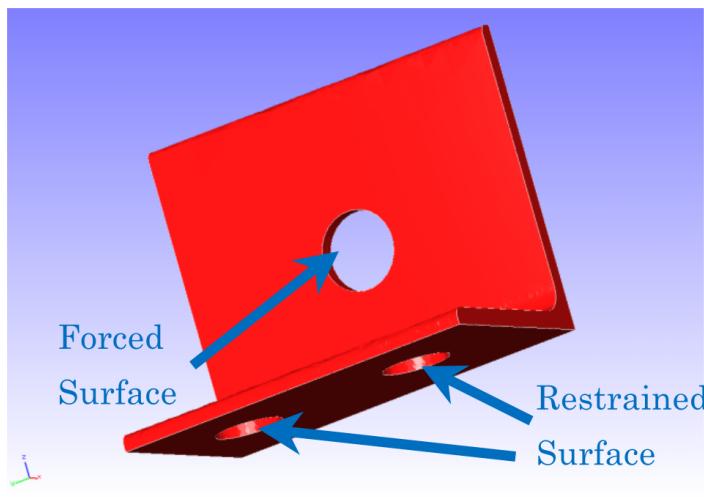


Fig. 4.1.1 : Shape of the hinge part

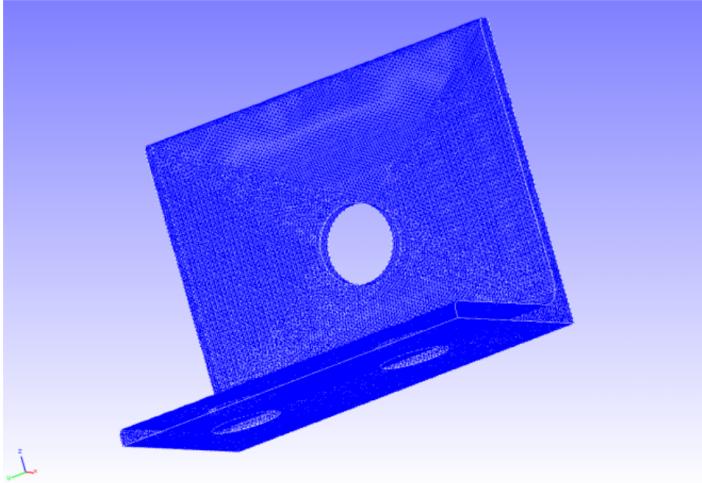


Fig. 4.1.2 : Mesh data of the hinge part

1.3.2 Analysis target

Extract the source code of FrontISTR and go to the directory of this example to see if you can find the files necessary for analysis.

File name	Type	Role
hecmw_ctrl.dat	Global control data	Specifies the input and output files for mesh data and analysis control data.
hinge.cnt	Analysis control data	Define the type of analysis, displacement boundary conditions, concentrated loads, etc., and also specify control of the solver and visualizer.
hinge.msh	Mesh data	Defines a finite element mesh and defines its material and section data

```
$ tar xvf FrontISTR.tar.gz
$ cd FrontISTR/tutorial/01_elastic_hinge
$ ls
hecmw_ctrl.dat  hinge.cnt  hinge.msh
```

A stress analysis is performed to constrain the displacement of the constrained surface shows in Figure 4.1.1 and to apply concentrated loads to the forcing surface.

The overall control data and analytical control data are shown below.

1.3.2.1 Global control data `hecmw_ctrl.dat`

```
#
# for solver
#
!MESH, NAME=fstrMSH, TYPE=HECMW-ENTIRE # Specify a single mesh data
hinge.msh
!CONTROL, NAME=fstrCNT                  # Specify analysis control data
hinge.cnt
!RESULT, NAME=fstrRES, IO=OUT            # Specify the result data
hinge.res
!RESULT, NAME=vis_out, IO=OUT            # Specify visualization data
hinge_vis
```

1.3.2.2 Analysis control data `hinge.cnt`

```

# Control File for FISTR
## Analysis Control
!VERSION 3 # Specify the version of the file format
!SOLUTION, TYPE=STATIC # Specify the type of analysis
!WRITE,RESULT # Specify the output of the result data
!WRITE,VISUAL # Specify the output of the visualization data
## Solver Control
### Boundary Condition
!BOUNDARY
  BND0, 1, 3, 0.000000 # Restrained surface 1
!BOUNDARY
  BND1, 1, 3, 0.000000 # Restrained surface 2
!CLOAD
  CL0, 1, 0.01000 # Specify a forced surface
### Material
!MATERIAL, NAME=STEEL # Specify material properties
!ELASTIC # Definition of elastic substances
  210000.0, 0.3
!DENSITY # Definition of mass density
  7.85e-6
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=YES,TIMELOG=YES # Solver control
  10000, 1
  1.0e-08, 1.0, 0.0
## Post Control
!VISUAL,metod=PSR # Specify the visualization method
!surface_num=1 # Number of surfaces in one surface rendering
!surface 1 # Specify the contents of the surface
!output_type=VTK # Specify the type of the visualization file
!END # Indicates the end of the analysis control data

```

1.3.3 Mesh data

(Some only)

```

!HEADER
  HECMW_Msh File generated by REVOCAP
!NODE
  1, -1.22042, 2.23355, 1.65220
  2, -1.27050, -3.10529, 1.59209
...
!ELEMENT, TYPE=342
  1, 1157, 3549, 3321, 3739, 12629, 12627, 12626, 12628,
  12631, 12630
  2, 8207, 3321, 3549, 3739, 12629, 12633, 12632, 12634,
  12630, 12631
...
!MATERIAL, NAME=STEEL, ITEM=2
!ITEM=1, SUBITEM=2
  210000.0, 0.3
!ITEM=2, SUBITEM=1
  7.85e-6
!SECTION, TYPE=SOLID, EGRP=Solid0, MATERIAL=STEEL
!EGROUP, EGRP=Solid0
  1
  2
...
!END

```

1.3.4 Analysis procedure

Execute the FrontISTR command fistr1 .

```
$ fistr1 -t 4  
(Runs in 4 threads.)
```

```
#####
#                         FrontISTR
#
#####
_____
version:      5.1.0
git_hash:     acab000c8c633b7b9d596424769e14363f720841
build:
  date:        2020-10-05T07:39:55Z
  MPI:         enabled
  OpenMP:      enabled
  option:      "-p --with-tools --with-refiner --with-metis --with-mumps --with-lapack --with-
               HECMW_METIS_VER: 5
execute:
  date:        2020-10-07T10:01:16+0900
  processes:   1
  threads:    4
  cores:       4
  host:
    0: flow-p06
_____
...
Step control not defined! Using default step=1
fstr_setup: OK
Start visualize PSF 1 at timestep 0

loading step= 1
sub_step= 1, current_time= 0.0000E+00, time_inc= 0.1000E+01
loading_factor= 0.0000000 1.0000000
### 3x3 BLOCK CG, SSOR, 1
  1 1.903375E+00
  2 1.974378E+00
  3 2.534627E+00
...
...
  2967 1.080216E-08
  2968 1.004317E-08
  2969 9.375729E-09
### Relative residual = 9.39429E-09

### summary of linear solver
  2969 iterations      9.394286E-09
  set-up time          : 1.953022E-01
  solver time          : 5.704201E+01
  solver/comm time     : 5.145826E-01
  solver/matvec        : 2.306329E+01
  solver/precond       : 2.632665E+01
  solver/1 iter         : 1.921253E-02
  work ratio (%)       : 9.909789E+01

Start visualize PSF 1 at timestep 1
### FSTR_SOLVE_NLGEOM FINISHED!
```

TOTAL TIME (sec) :	59.99
pre (sec) :	0.71
solve (sec) :	59.29

FrontISTR Completed !!

The analysis is completed when FrontISTR Completed !! is displayed, the analysis is done.

1.3.5 Analysis results

Once the analysis is complete, several new files will be created.

```
$ ls
0.log      hecmw_ctrl.dat  hinge.res.0.0          hinge_vis_psf.0001
FSTR.dbg.0 hecmw_vis.ini   hinge.res.0.1          hinge_vis_psf.0001.pvtu
FSTR.msg    hinge.cnt       hinge_vis_psf.0000
FSTR.sta    hinge.msh       hinge_vis_psf.0000.pvtu
```

The *.res.* is the result data of FrontISTR, which can be displayed by REVOCAP_PrePost and so on.

The *_vis_* is called visualization data, and can be displayed by general-purpose visualization software. In this example, the data is output in VTK format, so it can be displayed using ParaView and other visualization software.

A contour plot for the Mises stresses is created by REVOCAP_PrePost and shown in Figure 4.1.3. A portion of the analysis results log file is also shown below as numerical data for the analysis results.

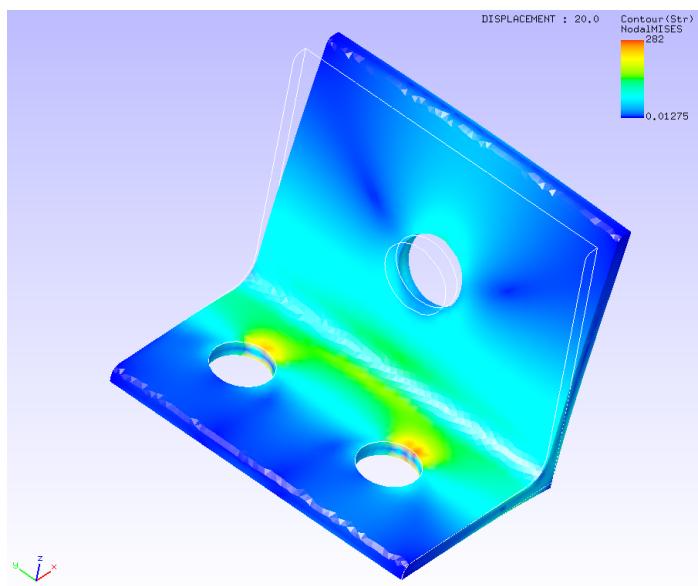


Fig. 4.1.3 : Analysis results of Mises stress

1.3.5.1 Analysis result log 0.log

```
fstr_setup: OK
##### Result step= 0
##### Local Summary @Node :Max/IdMax/Min/IdMin#####
//U1 0.0000E+00 1 0.0000E+00 1
//U2 0.0000E+00 1 0.0000E+00 1
//U3 0.0000E+00 1 0.0000E+00 1
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
//E33 0.0000E+00 1 0.0000E+00 1
//E12 0.0000E+00 1 0.0000E+00 1
```

```

//E23 0.0000E+00      1  0.0000E+00      1
//E31 0.0000E+00      1  0.0000E+00      1
//S11 0.0000E+00      1  0.0000E+00      1
//S22 0.0000E+00      1  0.0000E+00      1
//S33 0.0000E+00      1  0.0000E+00      1
//S12 0.0000E+00      1  0.0000E+00      1
//S23 0.0000E+00      1  0.0000E+00      1
//S31 0.0000E+00      1  0.0000E+00      1
//SMS 0.0000E+00      1  0.0000E+00      1
##### Local Summary @Element :Max/IdMax/Min/IdMin#####
//E11 0.0000E+00      1  0.0000E+00      1

```

1.4 Linear Static Analysis (Elasticity, Parallel)

To run 4 parallel static analysis(elasticity), and you can use the data in tutorial /02_elastic_hinge_parallel.

1.4.1 Analysis target

Item	Description	Notes	Reference
Type of analysis	Linear static analysis	!SOLUTION,TYPE=STATIC	
Number of nodes	84,056		
Number of elements	49,871		
Element type	10-node tetrahedron quadratic element	!ELEMENT,TYPE=342	Element Library
Material name	STEEL	!MATERIAL,NAME=STEEL	Material Data
Material property	ELASTIC	!ELASTIC	
Boundary conditions	Restraint, Concentrated force		
Matrix solution	CG/SSOR	!SOLVER,METHOD=CG,PRECOND=1	

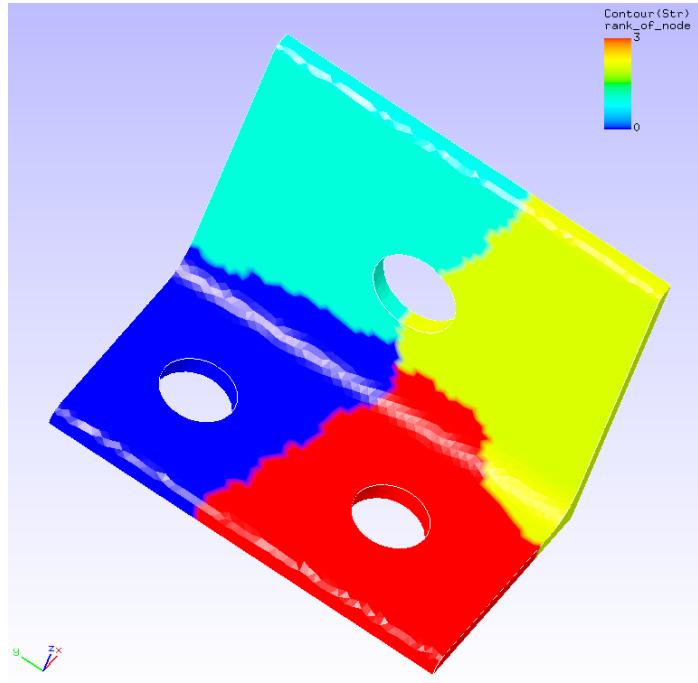


Figure 1: Analysis area of each node

Fig. 4.2.1 : Analysis area of each node

1.4.2 Analysis

Extract the code FrontISTR code and go to the directory in this example to check if you have the files necessary for analysis.

File name	Type	Role
hecmw_ctrl.dat	Global control data	
hinge.cnt	Analysis control data	
hinge.msh	Mesh data	
hecmw_part_ctrl.dat	Domain segmentation control data	Control data to divide the mesh data into regions by hecmw_part1

```
$ tar xvf FrontISTR.tar.gz
$ cd FrontISTR/tutorial/02_elastic_hinge_parallel
$ ls
hecmw_ctrl.dat  hecmw_part_ctrl.dat  hinge.cnt  hinge.msh
```

A stress analysis is performed to constrain the displacement of the constrained surface and add concentrated loads to the forced surface.

The overall control data, analysis control data and domain division control data are shown below.

1.4.2.1 Global control data `hecmw_ctrl.dat`

```
#  
# for partitioner  
#  
!MESH, NAME=part_in,TYPE=HECMW-ENTIRE # The original mesh data to be split in hecmw_part1  
hinge.msh  
!MESH, NAME=part_out,TYPE=HECMW-DIST # File name after splitting by hecmw_part1  
hinge_4  
#  
# for solver  
#  
!MESH, NAME=fstrMSH, TYPE=HECMW-DIST # Specify the mesh data to be split  
hinge_4  
!CONTROL, NAME=fstrCNT # Specify analysis control data  
hinge.cnt  
!RESULT, NAME=fstrRES, IO=OUT # Specify the result data  
hinge.res  
!RESULT, NAME=vis_out, IO=OUT # Specify the visualization data  
hinge_vis
```

1.4.2.2 Analysis control data `hinge.cnt`

```
# Control File for FISTR
## Analysis Control
!VERSION # Specify the version of the file format
3
!SOLUTION, TYPE=STATIC # Specify the type of analysis
!WRITE,RESULT # Specification of the result data output
```

```

!WRITE,VISUAL          # Specify the output of visualization data
## Solver Control
### Boundary Condition
!BOUNDARY
  BND0, 1, 3, 0.000000      # Restrained surface 1
!BOUNDARY
  BND1, 1, 3, 0.000000      # Restrained surface 2
!CLOAD
  CL0, 1, 0.01000          # Specify a forced surface
### Material
!MATERIAL, NAME=STEEL      # Specify material properties
!ELASTIC                  # Definition of elastic substances
  210000.0, 0.3
!DENSITY                  # Definition of mass density
  7.85e-6
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=YES,TIMELOG=YES  # Solver control
  10000, 2
  1.0e-08, 1.0, 0.0
## Post Control
!VISUAL,metod=PSR          # Specify the visualization methods
!surface_num=1              # Number of surfaces in a surface rendering
!surface 1                  # Specify the contents of the surface
!output_type=VTK            # Specify the type of the visualization file
!END                        # Indicates the end of the analysis control data

```

1.4.2.3 Domain division control data hecmw_part_ctrl.dat

```
!PARTITION,TYPE=NODE-BASED,METHOD=PMETIS,DOMAIN=4,UCD=part.inp
```

1.4.2.4 Analysis procedure

In order to run FrontISTR at MPI, the mesh data hinge.msh is first divided into four regions.

```
$ hecmw_part1
Oct 07 11:04:52 Info: Reading mesh file ...
Oct 07 11:04:52 Info: Starting domain decomposition...
Oct 07 11:04:52 TH(0/8) Info: Creating local mesh for domain #0 ...
Oct 07 11:04:52 TH(2/8) Info: Creating local mesh for domain #1 ...
Oct 07 11:04:52 TH(6/8) Info: Creating local mesh for domain #2 ...
Oct 07 11:04:52 TH(7/8) Info: Creating local mesh for domain #3 ...
Oct 07 11:04:52 Info: Domain decomposition done
```

New files called hinge_4.x and part.inp will be generated.

```
$ ls
hecmw_ctrl.dat  hecmw_part_ctrl.dat  hinge.msh  hinge_4.1  hinge_4.3
hecmw_part.log   hinge.cnt           hinge_4.0  hinge_4.2  part.inp
```

Next, you will execute the FrontISTR command fistr1 with MPI.

```
$ mpirun -np 4 fistr1 -t 1
(MPI 4 parallel, 1 OpenMP thread)
```

```
#####
#                                     FrontISTR                                #
#####
```

```
version:      5.1.0
git_hash:     acab000c8c633b7b9d596424769e14363f720841
```

```

build:
  date: 2020-10-05T07:39:55Z
  MPI: enabled
  OpenMP: enabled
  option: "-p --with-tools --with-refiner --with-metis --with-mumps --with-lapack --with-
HECMW_METIS_VER: 5
execute:
  date: 2020-10-07T11:07:21+0900
  processes: 4
  threads: 1
  cores: 4
  host:
    0: flow-p06
    1: flow-p06
    2: flow-p06
    3: flow-p06
  ...
  ...
Step control not defined! Using default step=1
fstr_setup: OK
Start visualize PSF 1 at timestep 0

loading step= 1
sub_step= 1, current_time= 0.0000E+00, time_inc= 0.1000E+01
loading_factor= 0.0000000 1.0000000
### 3x3 BLOCK CG, SSOR, 2
  1 2.183567E+00
  2 2.423900E+00
  3 2.939117E+00
  ...
  2084 1.158654E-08
  2085 1.032414E-08
  2086 9.436273E-09
### Relative residual = 9.43589E-09

### summary of linear solver
  2086 iterations 9.435886E-09
  set-up time : 4.695220E-02
  solver time : 7.103976E+01
  solver/comm time : 1.929294E+01
  solver/matvec : 1.544405E+01
  solver/precond : 3.243278E+01
  solver/1 iter : 3.405549E-02
  work ratio (%) : 7.284205E+01

Start visualize PSF 1 at timestep 1
### FSTR_SOLVE_NLGEOM FINISHED!

```

TOTAL TIME (sec) :	72.42
pre (sec) :	0.29
solve (sec) :	72.13

FrontISTR Completed !!

The analysis is completed when FrontISTR Completed!! is displayed, the analysis is done.

1.4.3 Analysis Results

Once the analysis is complete, several new files will be created.

```
$ ls
0.log      FSTR.dbg.3      hinge.cnt      hinge.res.2.1  hinge_vis_psf.0000
1.log      FSTR.msg       hinge.msh      hinge.res.3.0  hinge_vis_psf.0000.pvtu
2.log      FSTR.sta       hinge.res.0.0  hinge.res.3.1  hinge_vis_psf.0001
3.log      hecmw_ctrl.dat hinge.res.0.1  hinge_4.0    hinge_vis_psf.0001.pvtu
FSTR.dbg.0 hecmw_part.log hinge.res.1.0  hinge_4.1    part.inp
FSTR.dbg.1 hecmw_part_ctrl.dat hinge.res.1.1  hinge_4.2
FSTR.dbg.2 hecmw_vis.ini   hinge.res.2.0  hinge_4.3
```

The `*.res.*` is the result data, which contains results of analysis of each MPI node of FrontISTR.

The `*_vis_*` is called visualization data, and can be displayed by general-purpose visualization software. In this example, the data is output in VTK format and can be displayed using ParaView or other visualization software.

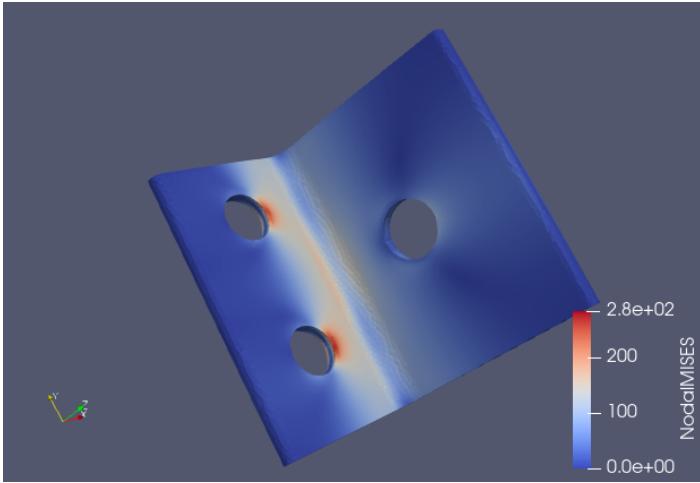


Fig. 4.2.2 Analytical results for Mises stress(displayed in VTK)

1.5 Non-Linear Static Analysis (Hyperelasticity, Part 1)

This analysis uses the data of tutorial /03_hyperelastic_cylinder.

1.5.1 Analysis target

The analysis target is a 1/8th model of a round bar. The geometry is shown in Figure 4.3.1 and the mesh data is shown in Figure 4.3.2.

Item	Description	Remarks	Reference
Type of analysis	Non-linear static analysis (hyperelasticity)	<code>!SOLUTION,TYPE=NL-STATIC</code>	
Number of nodes	629		
Number of elements	432		
Element type	Eight node hexahedral element	<code>!ELEMENT,TYPE=361</code>	
Material name	MAT1	<code>!MATERIAL,NODE=MAT1</code>	
Material property	ELASTIC	<code>!ELASTIC</code>	
Boundary condition	Restraint, Forced displacement		
Matrix solver	CG/SSOR	<code>!SOLVER,METHOD=CG,PRE-COND=1</code>	

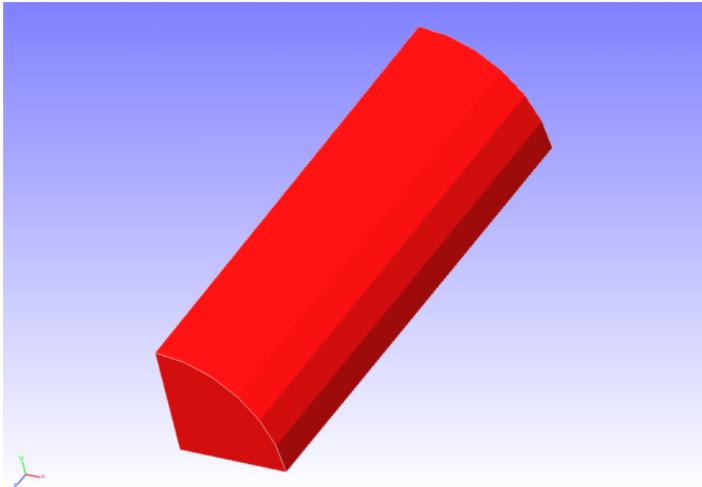


Fig. 4.3.1 : Shape of the round bar (1/8 model)

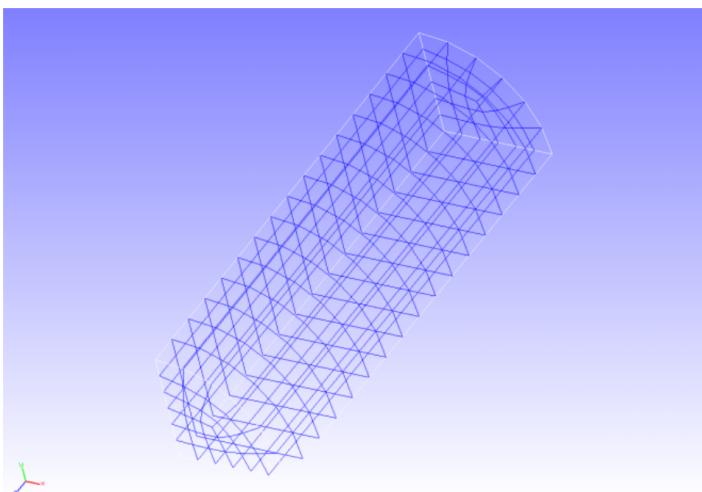


Fig. 4.3.2: Shape of the round bar (1/8 model)

1.5.2 Analysis content

In this stress analysis, an axial tensile displacement is given to a round bar. The Mooney–Rivlin model was used in the material constitutive equation of hyperelasticity. The analysis control data are presented below.

1.5.2.1 Analysis control data cylinder.cnt

```
# Control File for FISTR
## Analysis Control
!VERSION
3
!SOLUTION, TYPE=NLSTATIC
!WRITE,RESULT
!WRITE,VISUAL
## Solver Control
### Boundary Condition
!BOUNDARY, GRPID=1
LOADS, 3, 3, -7.0
FIX, 3, 3, 0.0
XSYMM, 1, 1, 0.0
YSYMM, 2, 2, 0.0
### STEP
!STEP, SUBSTEPS=5, CONVERG=1.0e-5
```

```

BOUNDARY, 1
### Material
!MATERIAL, NAME=MAT1
!HYPERELASTIC, TYPE=MOONEY-RIVLIN
  0.1486, 0.4849, 0.0789
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=YES,TIMELOG=YES
  10000, 1
  1.0e-8, 1.0, 0.0
## Post Control
!VISUAL, metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END

```

1.5.3 Analysis results

The results of the fifth substep are shown in Figure 4.3.3. A deformation diagram with Mises stress contours is created by REVOCAP_PrePost. A part of the analysis results log file is shown below as numerical data for the analysis results.

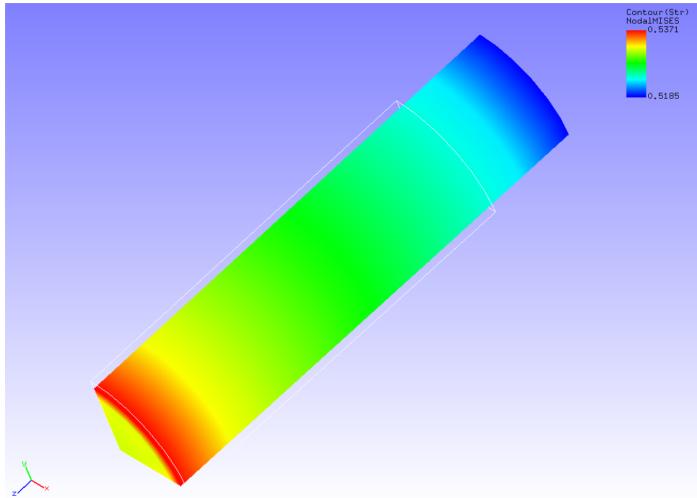


Fig. 4.3.3: Analysis results of deformation and Mises stress

```

fstr_setup: OK
##### Result step= 0
##### Local Summary @Node :Max/IdMax/Min/IdMin#####
//U1 0.0000E+00 1 0.0000E+00 1
//U2 0.0000E+00 1 0.0000E+00 1
//U3 0.0000E+00 1 0.0000E+00 1
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
//E33 0.0000E+00 1 0.0000E+00 1
//E12 0.0000E+00 1 0.0000E+00 1
//E23 0.0000E+00 1 0.0000E+00 1
//E31 0.0000E+00 1 0.0000E+00 1
//S11 0.0000E+00 1 0.0000E+00 1
//S22 0.0000E+00 1 0.0000E+00 1
//S33 0.0000E+00 1 0.0000E+00 1
//S12 0.0000E+00 1 0.0000E+00 1
//S23 0.0000E+00 1 0.0000E+00 1
//S31 0.0000E+00 1 0.0000E+00 1
//SMS 0.0000E+00 1 0.0000E+00 1

```

```
##### Local Summary @Element :Max/IdMax/Min/IdMin#####
//E11    0.0000E+00      1  0.0000E+00      1
//E22    0.0000E+00      1  0.0000E+00      1
```

1.6 Non-linear Static Analysis (Hyperelasticity, Part 2)

This analysis uses the data of tutorial /04_hyperelastic_spring.

1.6.1 Analysis target

The spring is the object of the analysis, the geometry is shown in Figure 4.4.1 and the mesh data is shown in Figure 4.4.2.

Item	Description	Notes	Reference
Type of analysis	Non-linear static analysis	!SOLUTION,TYPE=NLSTATIC	
Number of nodes	78,771		
Number of elements	46,454		
Element type	Ten node tetrahedral quadratic element	!ELEMENT,TYPE=342	
Material name	MAT1	!MATERIAL,NAME=MAT1	
Material property	HYPERELASTIC	!HYPERELASTIC,TYPE=ARRUDA-BOYCE	
Boundary conditions	Restraint, Concentrated force		
Matrix solution	CG/SSOR	!SOLVER,METHOD=CG,PRECOND=1	

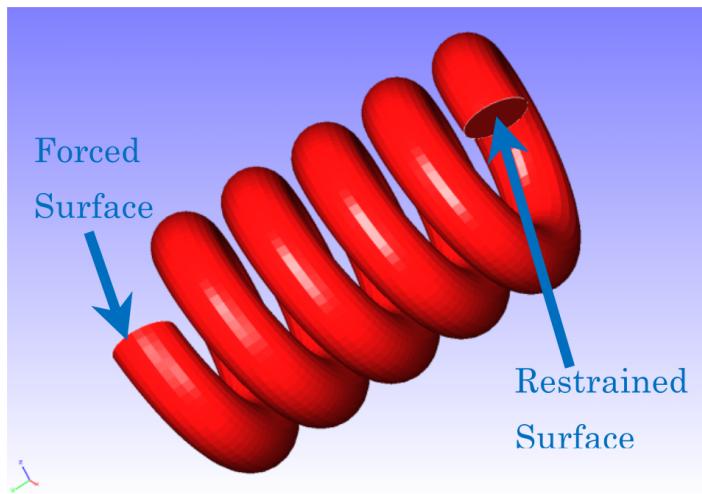


Fig. 4.4.1: Shape of the spring

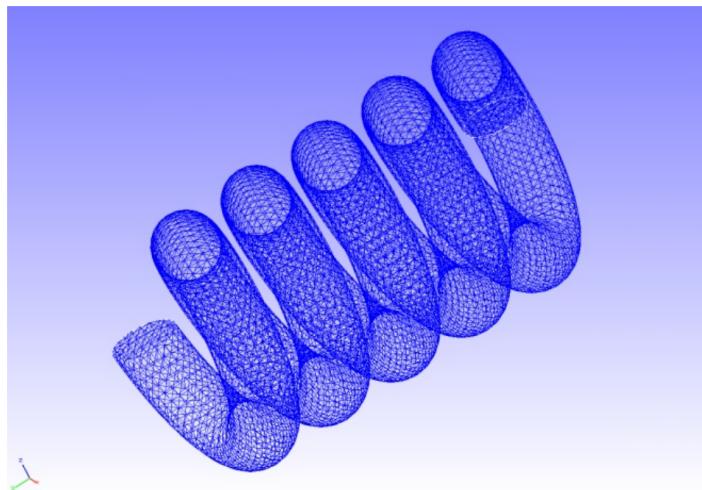


Fig. 4.4.2: Mesh data of the spring

1.6.2 Analysis content

In this stress analysis, the displacement of the constrained surface shown in Fig. 4.4.1 is restrained, and the displacement is given to the forced surface. The Arruda–Boyce model was used in the material constitutive equation of hyperelasticity. The analysis control data are presented below.

1.6.2.1 Analysis control data `spring.cnt`

```
# Control File for FISTR
## Analysis Control
!VERSION
3
!SOLUTION, TYPE=NLSTATIC
!WRITE,RESULT
!WRITE,VISUAL
## Solver Control
### Boundary Condition
!BOUNDARY, GRPID=1
LOADS, 2, 2, -5.0
FIX, 1, 3, 0.0
### STEP
!STEP, SUBSTEPS=1, CONVERG=1.0e-5
BOUNDARY, 1
### Material
!MATERIAL, NAME=MAT1
!HYPERELASTIC, TYPE=ARRUDA-BOYCE
0.71, 1.7029, 0.1408
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=YES,TIMELOG=YES
10000, 1
1.0e-8, 1.0, 0.0
## Post Control
!VISUAL, metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END
```

1.6.3 Analysis procedure

Execute the FrontISTR execution command `fistr1`.

```
$ cd FrontISTR/tutorial/04_hyperelastic_spring
$ fistr1 -t 4
(Runs in 4 threads.)
```

1.6.4 Analysis results

A deformation diagram with a contoured displacement is created by REVOCAP_PrePost and shown in Figure 4.4.3. A part of the log file is shown below as numerical data for the analysis results.

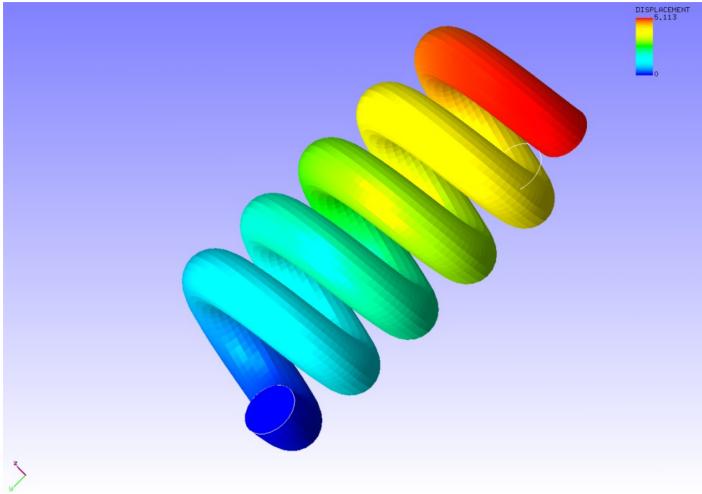


Fig. 4.4.3: Analysis results of deformation and displacement

1.6.4.1 Analysis results log 0.log

```
fstr_setup: OK
##### Result step= 0
##### Local Summary @Node :Max/IdMax/Min/IdMin#####
//U1 0.0000E+00 1 0.0000E+00 1
//U2 0.0000E+00 1 0.0000E+00 1
//U3 0.0000E+00 1 0.0000E+00 1
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
//E33 0.0000E+00 1 0.0000E+00 1
//E12 0.0000E+00 1 0.0000E+00 1
//E23 0.0000E+00 1 0.0000E+00 1
//E31 0.0000E+00 1 0.0000E+00 1
//S11 0.0000E+00 1 0.0000E+00 1
//S22 0.0000E+00 1 0.0000E+00 1
//S33 0.0000E+00 1 0.0000E+00 1
//S12 0.0000E+00 1 0.0000E+00 1
//S23 0.0000E+00 1 0.0000E+00 1
//S31 0.0000E+00 1 0.0000E+00 1
//SMS 0.0000E+00 1 0.0000E+00 1
##### Local Summary @Element :Max/IdMax/Min/IdMin#####
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
```

1.7 Non-linear Static Analysis (Elastoplastic, Part 1)

This analysis uses the data of tutorial /05_plastic_cylinder.

1.7.1 Analysis target

The object of analysis is the same round bar 1/8 model as in the Non-linear Static Analysis (Hyperelasticity, Part 1).

Item	Description	Remarks	Reference
Type of analysis	Non-linear static analysis (elastoplastic)	!SOLUTION,TYPE=NL-STATIC	
Number of nodes	629		
Number of elements	432		

Item	Description	Remarks	Reference
Element type	Eight node hexahedral element	<code>!ELEMENT,TYPE=361</code>	
Material name	MAT1	<code>!MATERIAL,NAMES=MAT1</code>	
Material property	ELASTIC,PLASTIC	<code>!ELASTIC !PLASTIC</code>	
Boundary condition	Restraint, Force displacement		
Matrix solver	CG/SSOR	<code>!SOLVER,METHOD=CG,PRECOND=1</code>	

1.7.2 Analysis content

The necking phenomenon of a round bar due to plastic deformation is analyzed. The Mises model is used for the yield function. The analytical control data is shown below.

1.7.2.1 Analysis control data necking.cnt

```
# Control File for FISTR
## Analysis Control
!VERSION
 3
!SOLUTION, TYPE=NSTATIC
!WRITE,RESULT,FREQUENCY=10
!WRITE,VISUAL,FREQUENCY=10
## Solver Control
### Boundary Condition
!BOUNDARY, GRPID=1
  LOADS, 3, 3, -7.0
  FIX,   3, 3, 0.0
  XSYMM, 1, 1, 0.0
  YSYMM, 2, 2, 0.0
### STEP
!STEP, SUBSTEPS=40, CONVERG=1.0e-3
  BOUNDARY, 1
### Material
!MATERIAL, NAME=MAT1
!ELASTIC
  206900.0, 0.29
!PLASTIC, YIELD=MISES, HARDEN=MULTILINEAR
  450.0, 0.0
  608.0, 0.05
  679.0, 0.1
  732.0, 0.2
  752.0, 0.3
  766.0, 0.4
  780.0, 0.5
### Output
!OUTPUT_VIS
  NSTRAIN, ON
!OUTPUT_RES
  ISTRESS, ON
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=NO,TIMELOG=YES
  2000, 1
  1.0e-8, 1.0, 0.0
## Post Control
!VISUAL,metod=PSR
!surface_num=1
```

```

!surface 1
!output_type=VTK
!END

```

1.7.3 Analysis procedure

Execute the FrontISTR execution command fistr1 .

```

$ cd FrontISTR/tutorial/05_plastic_cylinder
$ fistr1 -t 4
(Runs in 4 threads.)

```

1.7.4 Analysis results

The results of the 35th substep are shown in Figure 4.5.1. A deformation diagram with the Mises stress contours added is created by REVOCAPI_PrePost. A part of the analysis results log file is shown below as numerical data for the analysis results.

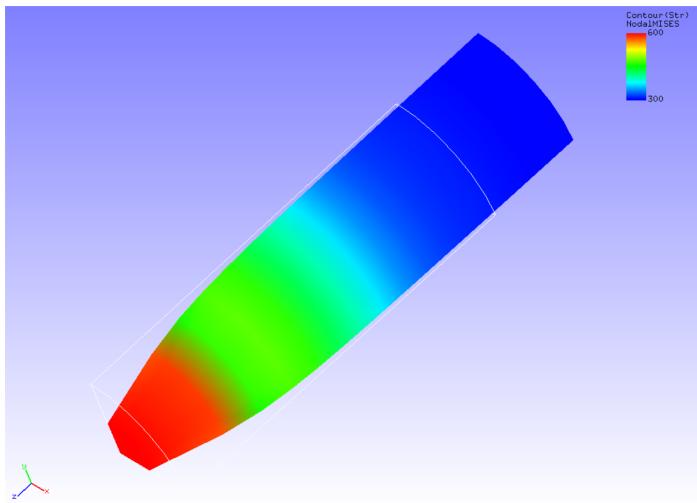


Fig. 4.5.1: Analysis results of deformation and Mises stress

1.7.4.1 Analysis results log 0.log

```

fstr_setup : OK
##### Result step= 0
##### Local Summary @Node :Max/IdMax/Min/IdMin#####
//U1 0.0000E+00 1 0.0000E+00 1
//U2 0.0000E+00 1 0.0000E+00 1
//U3 0.0000E+00 1 0.0000E+00 1
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
//E33 0.0000E+00 1 0.0000E+00 1
//E12 0.0000E+00 1 0.0000E+00 1
//E23 0.0000E+00 1 0.0000E+00 1
//E31 0.0000E+00 1 0.0000E+00 1
//S11 0.0000E+00 1 0.0000E+00 1
//S22 0.0000E+00 1 0.0000E+00 1
//S33 0.0000E+00 1 0.0000E+00 1
//S12 0.0000E+00 1 0.0000E+00 1
//S23 0.0000E+00 1 0.0000E+00 1
//S31 0.0000E+00 1 0.0000E+00 1
//SMS 0.0000E+00 1 0.0000E+00 1
##### Local Summary @Element :Max/IdMax/Min/IdMin#####
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1

```

1.8 Non-linear Static Analysis (Elastoplastic, Part 2)

This analysis uses the data of tutorial /06_plastic_can.

1.8.1 Analysis target

The target of this analysis is a 1/2 model of a container whose shape and mesh data are shown in Figs. 4.6.1 and 4.6.2, respectively. The mesh is a tetrahedral secondary element with 7236 elements and 14119 nodes.

Item	Description	Notes	Reference
Type of analysis	Non-linear static analysis(plastic)	!SOLUTION,TYPE=NLSTATIC	
Number of nodes	14,119		
Number of elements	7,236		
Element type	Ten node tetrahedral quadratic element	!ELEMENT,TYPE=342	
Material name	M1	!MATERIAL,NAME=M1	
Material property	ELASTIC, PLASTIC	!ELASTIC !PLASTIC,YIELD=DRACKER-PRAGER	
Boundary conditions	Restraint, Distribution force	!DLOAD	
Matrix solution	CG/SSOR	!SOLVER,METHOD=CG,PRE-COND=1	

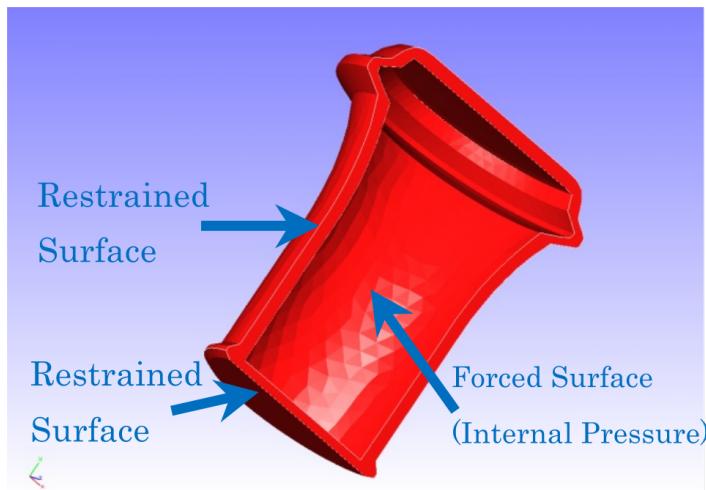


Fig. 4.6.1: Shape of the container

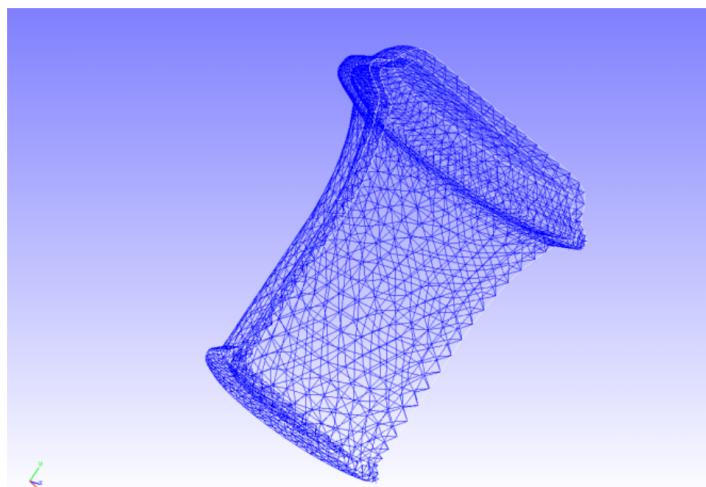


Fig. 4.6.2: Mesh data of the container

1.8.2 Analysis content

A stress analysis is performed by restraining the displacement of the restraining surface as shown in Fig. 4.6.1 and applying distributed loads to the inside of the vessel as a forced surface. The Drucker-Prager model is used for the yield function. The analytical control data are shown below.

1.8.2.1 Analysis control data can.cnt

```
# Control File for FISTR
## Analysis Control
!VERSION
 3
!SOLUTION, TYPE=NLSTATIC
!WRITE,RESULT
!WRITE,VISUAL
## Solver Control
### Boundary Condition
!BOUNDARY, GRPID=1
  BND0, 3, 3, 0.000000
!BOUNDARY, GRPID=1
  BND1, 1, 1, 0.000000
  BND1, 2, 2, 0.000000
  BND1, 3, 3, 0.000000
!DLOAD, GRPID=1
  DL0, S, 1.0
!DLOAD, GRPID=1
  DL1, S, 1.0
!DLOAD, GRPID=1
  DL2, S, 0.5
### STEP
!STEP, SUBSTEPS=10, CONVERG=1.0e-5
  BOUNDARY, 1
  LOAD, 1
### Material
!MATERIAL, NAME=M1
!ELASTIC
  24000.0, 0.2
!PLASTIC, YIELD=DRUCKER-PRAGER
  500.0, 20.0, 0.0
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=NO,TIMELOG=YES
  20000, 1
  1.0e-8, 1.0, 0.0
## Post Control
!VISUAL,metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END
```

1.8.3 Analysis procedure

Execute the FrontISTR execution command `fistr1`.

```
$ cd FrontISTR/tutorial/06_plastic_can
$ fistr1 -t 4
(Runs in 4 threads.)
```

1.8.4 Analysis results

For the results of the tenth substep analysis, a deformation diagram with the Mises stress contours added is created by REVOCAP_PrePost and shown in Figure 4.6.3. The deformation factor is set to 30. A part of the log file is shown below as numerical data of the analysis results.

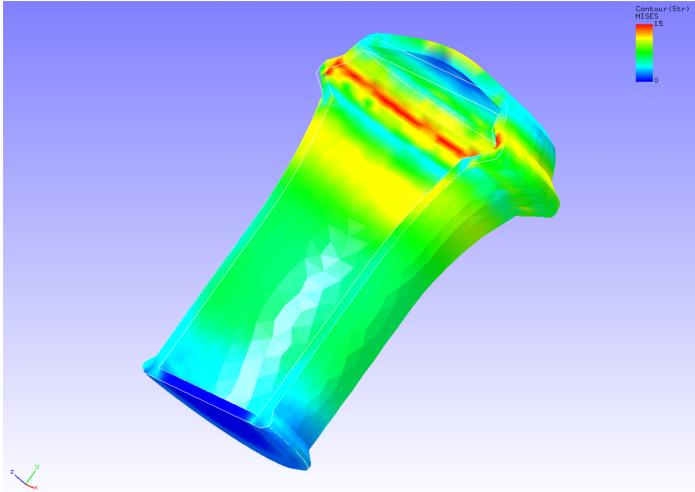


Fig. 4.6.3: Analysis results of deformation and Mises stress

1.8.4.1 Analysis results log 0.log.

```
fstr_setup: OK
##### Result step= 0
##### Local Summary @Node :Max/IdMax/Min/IdMin#####
//U1 0.0000E+00 1 0.0000E+00 1
//U2 0.0000E+00 1 0.0000E+00 1
//U3 0.0000E+00 1 0.0000E+00 1
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
//E33 0.0000E+00 1 0.0000E+00 1
//E12 0.0000E+00 1 0.0000E+00 1
//E23 0.0000E+00 1 0.0000E+00 1
//E31 0.0000E+00 1 0.0000E+00 1
//S11 0.0000E+00 1 0.0000E+00 1
//S22 0.0000E+00 1 0.0000E+00 1
//S33 0.0000E+00 1 0.0000E+00 1
//S12 0.0000E+00 1 0.0000E+00 1
//S23 0.0000E+00 1 0.0000E+00 1
//S31 0.0000E+00 1 0.0000E+00 1
//SMS 0.0000E+00 1 0.0000E+00 1
##### Local Summary @Element :Max/IdMax/Min/IdMin#####
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
```

1.9 Non-linear Static Analysis (Elastoplastic, Part 2)

This analysis uses the data of tutorial /06_plastic_can.

1.9.1 Analysis target

The target of this analysis is a 1/2 model of a container whose shape and mesh data are shown in Figs. 4.6.1 and 4.6.2, respectively. The mesh is a tetrahedral secondary element with 7236 elements and 14119 nodes.

Item	Description	Notes	Reference
Type of analysis	Non-linear static analysis(plastic)	!SOLUTION,TYPE=NSTATIC	
Number of nodes	14,119		
Number of elements	7,236		
Element type	Ten node tetrahedral quadratic element	!ELEMENT,TYPE=342	
Material name	M1	!MATERIAL,NAME=M1	
Material property	ELASTIC, PLASTIC	!ELASTIC !PLASTIC,YIELD=DRACKER-PRAGER !DLOAD	
Boundary conditions	Restraint, Distribution force		
Matrix solution	CG/SSOR	!SOLVER,METHOD=CG,PRE-COND=1	

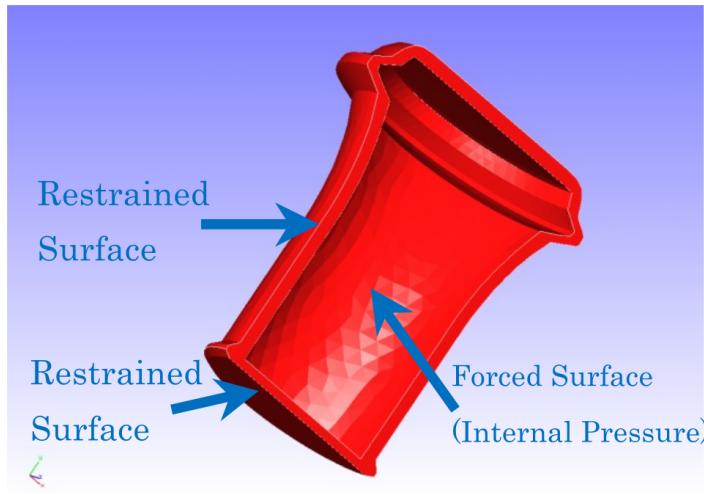


Fig. 4.6.1: Shape of the container

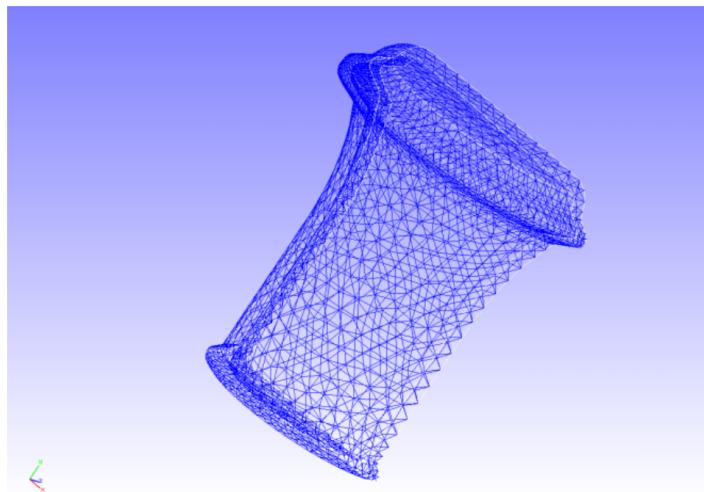


Fig. 4.6.2: Mesh data of the container

1.9.2 Analysis content

A stress analysis is performed by restraining the displacement of the restraining surface as shown in Fig. 4.6.1 and applying distributed loads to the inside of the vessel as a forced surface. The Drucker-Prager model is used for the yield function. The analytical control data are shown below.

1.9.2.1 Analysis control data can.cnt

```
# Control File for FISTR
## Analysis Control
!VERSION
 3
!SOLUTION, TYPE=NLSTATIC
!WRITE,RESULT
!WRITE,VISUAL
## Solver Control
### Boundary Condition
!BOUNDARY, GRPID=1
  BND0, 3, 3, 0.000000
!BOUNDARY, GRPID=1
  BND1, 1, 1, 0.000000
  BND1, 2, 2, 0.000000
  BND1, 3, 3, 0.000000
!DLOAD, GRPID=1
  DL0, S, 1.0
!DLOAD, GRPID=1
  DL1, S, 1.0
!DLOAD, GRPID=1
  DL2, S, 0.5
### STEP
!STEP, SUBSTEPS=10, CONVERG=1.0e-5
  BOUNDARY, 1
  LOAD, 1
### Material
!MATERIAL, NAME=M1
!ELASTIC
  24000.0, 0.2
!PLASTIC, YIELD=DRUCKER-PRAGER
  500.0, 20.0, 0.0
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=NO,TIMELOG=YES
  20000, 1
  1.0e-8, 1.0, 0.0
## Post Control
!VISUAL, metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END
```

1.9.3 Analysis procedure

Execute the FrontISTR execution command fistrl .

```
$ cd FrontISTR/tutorial/06_plastic_can
$ fistrl -t 4
(Runs in 4 threads.)
```

1.9.4 Analysis results

For the results of the tenth substep analysis, a deformation diagram with the Mises stress contours added is created by REVOCAP_PrePost and shown in Figure 4.6.3. The deformation factor is set to 30. A part of the log file is shown below as numerical data of the analysis results.

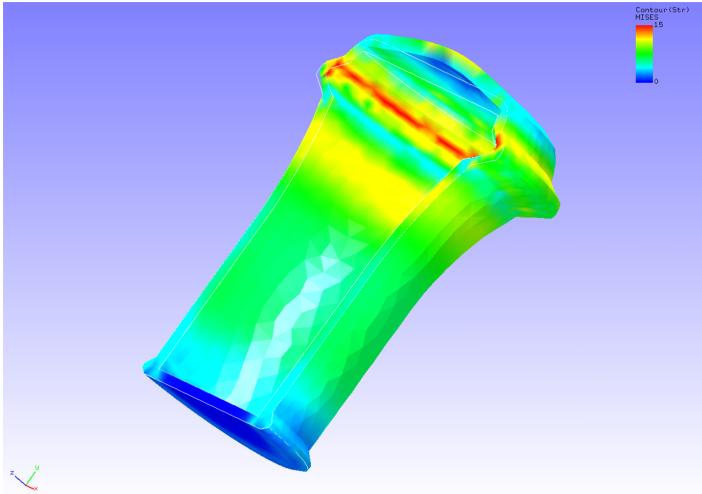


Fig. 4.6.3: Analysis results of deformation and Mises stress

1.9.4.1 Analysis results log 0.log.

```
fstr_setup : OK
##### Result step= 0
##### Local Summary @Node :Max/IdMax/Min/IdMin#####
//U1 0.0000E+00 1 0.0000E+00 1
//U2 0.0000E+00 1 0.0000E+00 1
//U3 0.0000E+00 1 0.0000E+00 1
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
//E33 0.0000E+00 1 0.0000E+00 1
//E12 0.0000E+00 1 0.0000E+00 1
//E23 0.0000E+00 1 0.0000E+00 1
//E31 0.0000E+00 1 0.0000E+00 1
//S11 0.0000E+00 1 0.0000E+00 1
//S22 0.0000E+00 1 0.0000E+00 1
//S33 0.0000E+00 1 0.0000E+00 1
//S12 0.0000E+00 1 0.0000E+00 1
//S23 0.0000E+00 1 0.0000E+00 1
//S31 0.0000E+00 1 0.0000E+00 1
//SMS 0.0000E+00 1 0.0000E+00 1
##### Local Summary @Element :Max/IdMax/Min/IdMin#####
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
```

1.10 Non-Linear Static Analysis (Viscoelasticity)

This analysis uses the data of tutorial /07_viscoelastic_cylinder.

1.10.1 Analysis target

The object of analysis is the same round-bar 1/8 model as the Non-Linear Static Analysis (Hyperelasticity, Part 1).

1.10.2 Analysis content

A stress relaxation analysis is performed by applying axial tensile displacement to a round bar. The analysis control data is shown below.

1.10.2.1 Analysis control data cylinder.cnt

```

# Control File for FISTR
## Analysis Control
!VERSION
 3
!SOLUTION, TYPE=NLSTATIC
!WRITE,VISUAL
!WRITE,RESULT
## Solver Control
### Boundary Condition
!BOUNDARY, GRPID=1
  LOADS, 3, 3, -7.0
  FIX,   3, 3, 0.0
  XSYMM, 1, 1, 0.0
  YSYMM, 2, 2, 0.0
### STEP
!STEP, TYPE=VISCO, CONVERG=1.0e-5
  0.2, 2.0
  BOUNDARY, 1
### Material
!MATERIAL, NAME=MAT1
!ELASTIC
  206900.0, 0.29
!VISCOELASTIC
  0.5, 1.0
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=YES,TIMELOG=YES
  10000, 1
  1.0e-8, 1.0, 0.0
## Post Control
!VISUAL,method=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END

```

1.10.3 Analysis procedure

Execute the FrontISTR execution command `fistr1`.

```

$ cd FrontISTR/tutorial/07_viscoelastic_cylinder
$ fistr1 -t 4
(Runs in 4 threads.)

```

1.10.4 Analysis results

The deformation diagram with the Mises stress contours added is created by REVOCAP_PrePost and shown in Figure 4.7.1. The analysis results after 2 seconds (10th step). A part of the log file is shown below as numerical data of the analysis results.

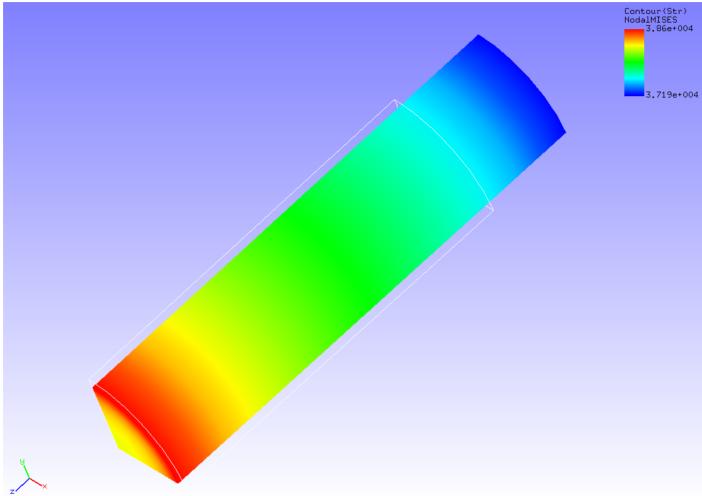


Fig. 4.7.1: Analysis results of deformation and Mises stress

1.10.4.1 Analysis results log 0.log.

```
fstr_setup : OK
##### Result step= 0
##### Local Summary @Node :Max/IdMax/Min/IdMin#####
//U1 0.0000E+00 1 0.0000E+00 1
//U2 0.0000E+00 1 0.0000E+00 1
//U3 0.0000E+00 1 0.0000E+00 1
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
//E33 0.0000E+00 1 0.0000E+00 1
//E12 0.0000E+00 1 0.0000E+00 1
//E23 0.0000E+00 1 0.0000E+00 1
//E31 0.0000E+00 1 0.0000E+00 1
//S11 0.0000E+00 1 0.0000E+00 1
//S22 0.0000E+00 1 0.0000E+00 1
//S33 0.0000E+00 1 0.0000E+00 1
//S12 0.0000E+00 1 0.0000E+00 1
//S23 0.0000E+00 1 0.0000E+00 1
//S31 0.0000E+00 1 0.0000E+00 1
//SMS 0.0000E+00 1 0.0000E+00 1
##### Local Summary @Element :Max/IdMax/Min/IdMin#####
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
```

1.11 Non-Linear Static Analysis (Creep)

This analysis uses the data of tutorial /08_creep_cylinder.

1.11.1 Analysis target

The analysis targets are the same round bar 1/8 model as in the Non-Linear Static Analysis (Hyperelasticity, part 1) in section 4.3.

Item	Description	Notes	Reference
Type of analysis	Non-linear static analysis	!SOLUTION,TYPE=NSTATIC	
Number of nodes	629		
Number of elements	432		
Element type	Eight node hexahedral element	!ELEMENT,TYPE=361	
Material name	MAT1	!MATERIAL,NAME=MAT1	

Item	Description	Notes	Reference
Material property	ELASTIC,CREEP	!ELASTIC !CREEP,TYPE=NORTON	
Boundary conditions	Restraint,Forced displacement		
Matrix solution	CG/SSOR	!SOLVER,METHOD=CG,PRECOND=1	

1.11.2 Analysis content

Creep behavior analysis is performed by applying tensile displacement in the axial direction to a round bar. The analysis control data is shown below.

1.11.2.1 Analysis control data cylinder.cnt.

```
# Control File for FISTR
## Analysis Control
!VERSION
 3
!SOLUTION, TYPE=NLSTATIC
!WRITE,RESULT
!WRITE,VISUAL
## Solver Control
### Boundary Conditon
!BOUNDARY, GRPID=1
  LOADS, 3, 3, -7.0
  FIX,   3, 3, 0.0
  XSYMM, 1, 1, 0.0
  YSYMM, 2, 2, 0.0
### STEP
!STEP, SUBSTEPS=5, CONVERG=1.0e-5
  BOUNDARY, 1
### Material
!MATERIAL, NAME=MAT1
!ELASTIC
  206900.0, 0.29
!CREEP, TYPE=NORTON
  1.e-10, 5.0, 0.0
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=YES,TIMELOG=YES
  10000, 1
  1.0e-8, 1.0, 0.0
## Post Control
!VISUAL,metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END
```

1.11.3 Analysis procedure

Execute the FrontISTR execution command fistr1.

```
$ cd FrontISTR/tutorial/08_creep_cylinder
$ fistr1 -t 4
(Runs in 4 threads.)
```

1.11.4 Analysis results

The results of the 5th substep are shown in Figure 4.8.1. A deformation diagram with Mises stress contours is created by REVOCAP_PrePost. A part of the analysis results log file is shown below as numerical data for the analysis results.

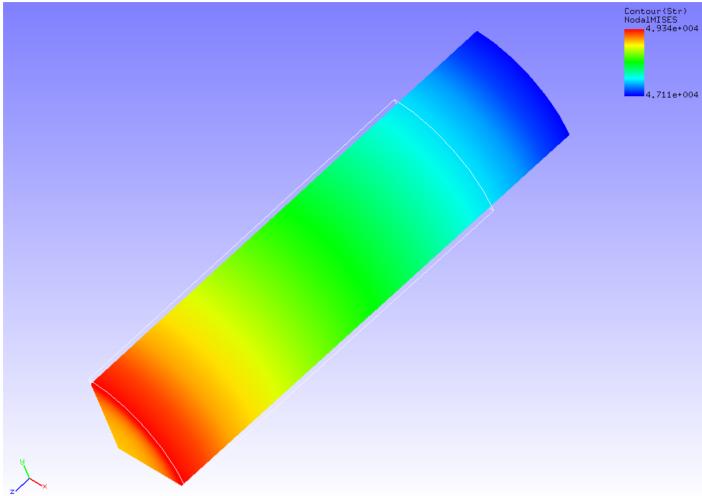


Fig. 4.8.1: Analysis results of deformation and Mises stress

1.11.4.1 Analysis results log 0.log.

```
fstr_setup : OK
##### Result step=      0
##### Local Summary @Node :Max/IdMax/Min/IdMin#####
//U1    0.0000E+00      1  0.0000E+00      1
//U2    0.0000E+00      1  0.0000E+00      1
//U3    0.0000E+00      1  0.0000E+00      1
//E11   0.0000E+00      1  0.0000E+00      1
//E22   0.0000E+00      1  0.0000E+00      1
//E33   0.0000E+00      1  0.0000E+00      1
//E12   0.0000E+00      1  0.0000E+00      1
//E23   0.0000E+00      1  0.0000E+00      1
//E31   0.0000E+00      1  0.0000E+00      1
//S11   0.0000E+00      1  0.0000E+00      1
//S22   0.0000E+00      1  0.0000E+00      1
//S33   0.0000E+00      1  0.0000E+00      1
//S12   0.0000E+00      1  0.0000E+00      1
//S23   0.0000E+00      1  0.0000E+00      1
//S31   0.0000E+00      1  0.0000E+00      1
//SMS   0.0000E+00      1  0.0000E+00      1
##### Local Summary @Element :Max/IdMax/Min/IdMin#####
//E11   0.0000E+00      1  0.0000E+00      1
//E22   0.0000E+00      1  0.0000E+00      1
```

1.12 Contact Analysis (Part 1)

This analysis uses the data of tutorial /09_contact_hertz.

1.12.1 Analysis target

The analysis is a Hertz contact problem and the geometry to be analyzed is shown in Figure 4.9.1 and the mesh data is shown in Figure 4.9.2.

Item	Description	Notes	Reference
Type of analysis	Non-linear static analysis(elastic,contact)	!SOLUTION,TYPE=NL- STATIC !CONTACT	
Number of nodes	408		

Item	Description	Notes	Reference
Number of elements	168		
Element type	Eight node hexahedral element	!ELEMENT,TYPE=361	
Material name	MAT1	!MATERIAL,NAME=MAT1	
Material property	ELASTIC	!ELASTIC	
Boundary conditions	Restraint,Forced displacement		
Matrix solution	CG/SSOR	!SOLVER,METHOD=CG,PRE-COND=1	

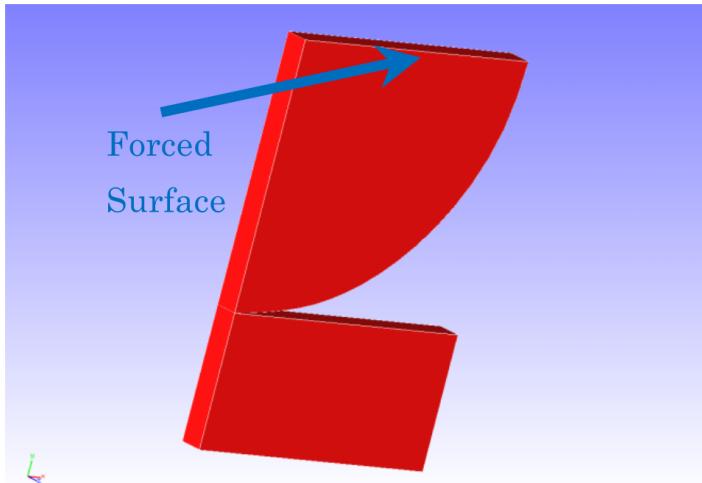


Fig. 4.9.1: Shape of the analysis target

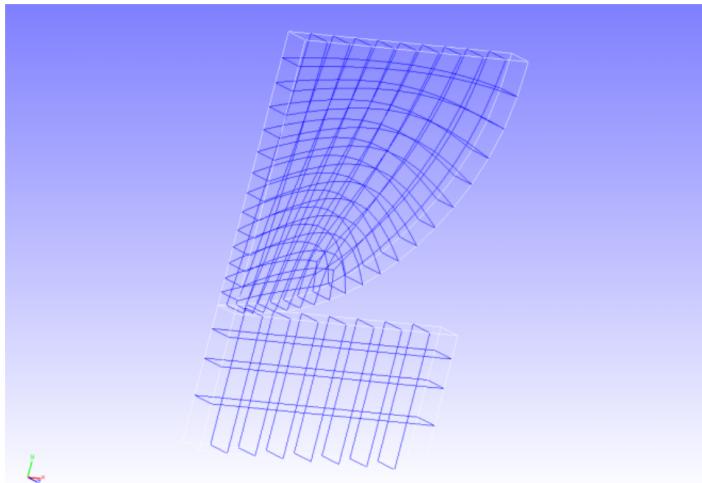


Fig. 4.9.2: Mesh data of the analysis target

1.12.2 Analysis content

The extended Lagrangian multiplier method is used to perform a contact analysis in which a forced displacement in the compressive direction is applied to the upper surface of a 1/4 disc model. The analytical control data are shown below.

1.12.2.1 Analysis control data `cgs3.cnt`.

```
# Control File for FISTR
## Analysis Control
!VERSION
```

```

3
!SOLUTION, TYPE=NSTATIC
!WRITE,RESULT
!WRITE,VISUAL
## Solver Control
### Boundary Condition
!BOUNDARY, GRPID=1
ALL, 3, 3, 0.0
BOTTOM, 2, 2, 0.0
CENTER, 1, 1, 0.0
UPPER, 2, 2, -0.306
!CONTACT_ALGO, TYPE=LAGRANGE
!CONTACT, GRPID=1
CP1, 0.0
### STEP
!STEP, SUBSTEPS=5, CONVERG=1.0e-5
BOUNDARY, 1
CONTACT, 1
### Material
!MATERIAL, NAME=MAT1
!ELASTIC
1100.0, 0.0
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=YES,TIMELOG=YES
1000, 1
1.0e-10, 1.0, 0.0
## Post Control
!VISUAL, metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END

```

1.12.3 Analysis procedure

Execute the FrontISTR execution command `fistr1`.

```

$ cd FrontISTR/tutorial/09_contact_hertz
$ fistr1 -t 4
(Runs in 4 threads.)

```

1.12.4 Analysis results

The results of the 5th substep are shown in Figure 4.9.3. A deformation diagram with contours of the y-direction displacement is created by REVOCAP_PrePost. A part of the analysis results log file is shown below as numerical data for the analysis results.

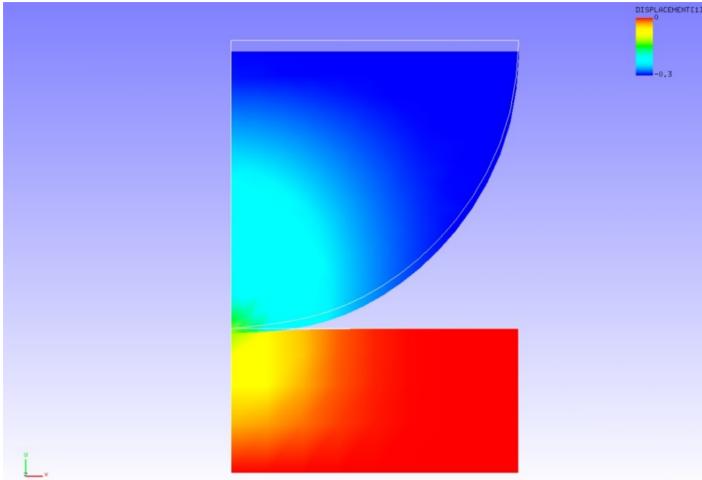


Fig. 4.9.3: Analysis results of deformation and y-direction displacement

1.12.4.1 Analysis results log 0.log.

```
fstr_setup: OK
##### Result step= 0
##### Local Summary @Node :Max/IdMax/Min/IdMin#####
//U1 0.0000E+00 1 0.0000E+00 1
//U2 0.0000E+00 1 0.0000E+00 1
//U3 0.0000E+00 1 0.0000E+00 1
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
//E33 0.0000E+00 1 0.0000E+00 1
//E12 0.0000E+00 1 0.0000E+00 1
//E23 0.0000E+00 1 0.0000E+00 1
//E31 0.0000E+00 1 0.0000E+00 1
//S11 0.0000E+00 1 0.0000E+00 1
//S22 0.0000E+00 1 0.0000E+00 1
//S33 0.0000E+00 1 0.0000E+00 1
//S12 0.0000E+00 1 0.0000E+00 1
//S23 0.0000E+00 1 0.0000E+00 1
//S31 0.0000E+00 1 0.0000E+00 1
//SMS 0.0000E+00 1 0.0000E+00 1
##### Local Summary @Element :Max/IdMax/Min/IdMin#####
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
```

1.13 Contact Analysis (Part 2)

This analysis uses the data of tutorial /10_contact_2tubes.

1.13.1 Analysis target

The analysis is a cylindrical indentation problem, and the geometry of the analysis target is shown in Figure 4.10.1 and the mesh data is shown in Figure 4.10.2.

Item	Description	Notes	Reference
Type of analysis	Non-linear static analysis(elastic,contact)	!SOLUTION,TYPE=NL- STATIC !CONTACT	
Number of nodes	4,000		
Number of elements	2,888		

Item	Description	Notes	Reference
Element type	Eight node hexahedral element	!ELEMENT,TYPE=361	
Material name	M1	!MATERIAL,NAME=M1	
Material property	ELASTIC	!ELASTIC	
Boundary conditions	Restraint,Forced displacement		
Matrix solution	Direct method	!!SOLVER,METHOD=MUMPS	

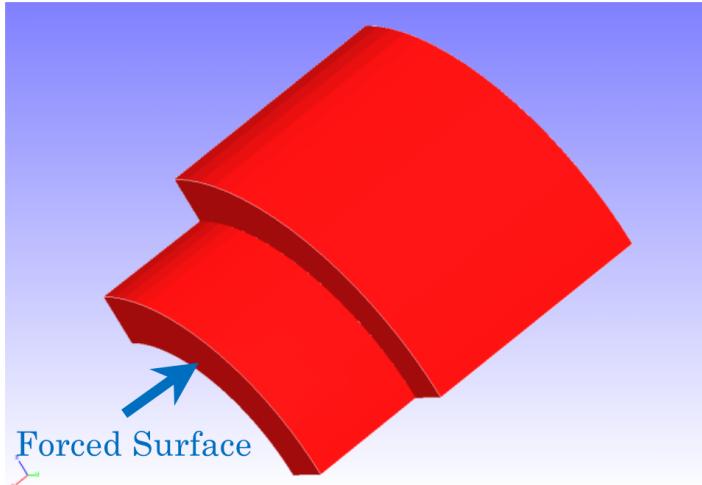


Fig. 4.10.1: Shape of the analysis target

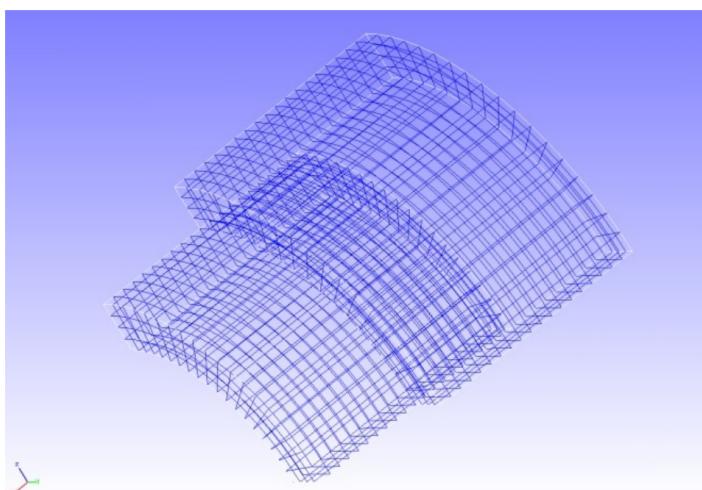


Fig. 4.10.2: Mesh data of the analysis target

1.13.2 Analysis content

The Lagrangian multiplier method is used to perform contact analysis to give the forced displacement in the push-in direction to the forced surface shown in Figure 4.10.1. The analytical control data is shown below.

1.13.2.1 Analysis control data 2tubes.cnt.

```
# Control File for FISTR
## Analysis Control
!VERSION
 3
!SOLUTION, TYPE=NSTATIC
!WRITE,RESULT
!WRITE,VISUAL
```

```

## Solver Control
### Boundary Condition
!BOUNDARY, GRPID=1
  X0, 1, 3, 0.0
  Y0, 2, 2, 0.0
  Z0, 3, 3, 0.0
!BOUNDARY, GRPID=2
  X1, 1, 1, 0.0
!BOUNDARY, GRPID=3
  X1, 1, 1, -1.0
!CONTACT_ALGO, TYPE=SLAGRANGE
!CONTACT, GRPID=1, INTERACTION=FSLID, NPENALTY=1.0e+2
  CP1, 0.0, 1.0e+5
### STEP
!STEP, SUBSTEPS=4, CONVERG=1.0e-5
  BOUNDARY, 1
  BOUNDARY, 3
  CONTACT, 1
### Material
!MATERIAL, NAME=M1
!ELASTIC
  2.1e+5, 0.3
### Solver Setting
!SOLVER, METHOD=MUMPS
## Post Control
!VISUAL, metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END

```

1.13.3 Analysis procedure

Execute the FrontISTR execution command fistr1 .

```

$ cd FrontISTR/tutorial/10_contact_2tubes
$ fistr1 -t 4
(Runs in 4 threads.)

```

1.13.4 Analysis results

The results of the fourth substep are shown in Figure 4.10.3. A deformation diagram with Mises stress contours is created by REVOCAP_PrePost. A part of the analysis results log file is shown below as numerical data for the analysis results.

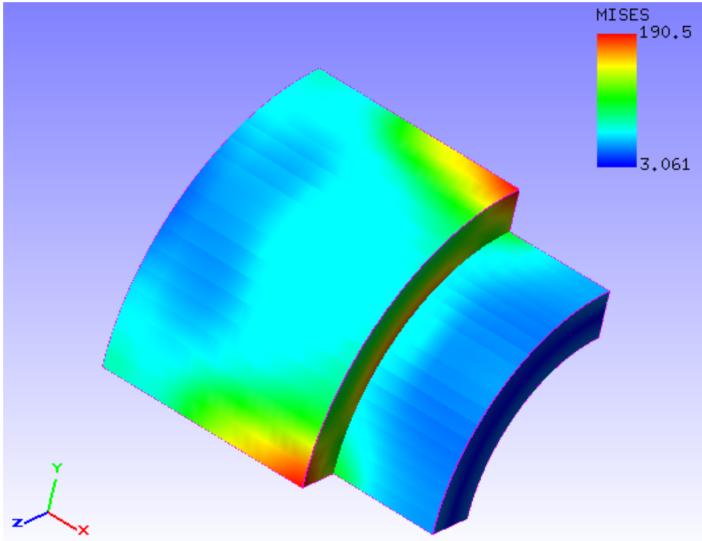


Fig. 4.10.3: Analysis results of deformation and Mises stress

1.13.4.1 Analysis results log 0.log.

```
fstr_setup : OK
##### Result step= 0
##### Local Summary @Node : Max/IdMax/Min/IdMin#####
//U1 0.0000E+00 1 0.0000E+00 1
//U2 0.0000E+00 1 0.0000E+00 1
//U3 0.0000E+00 1 0.0000E+00 1
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
//E33 0.0000E+00 1 0.0000E+00 1
//E12 0.0000E+00 1 0.0000E+00 1
//E23 0.0000E+00 1 0.0000E+00 1
//E31 0.0000E+00 1 0.0000E+00 1
//S11 0.0000E+00 1 0.0000E+00 1
//S22 0.0000E+00 1 0.0000E+00 1
//S33 0.0000E+00 1 0.0000E+00 1
//S12 0.0000E+00 1 0.0000E+00 1
//S23 0.0000E+00 1 0.0000E+00 1
//S31 0.0000E+00 1 0.0000E+00 1
//SMS 0.0000E+00 1 0.0000E+00 1
##### Local Summary @Element : Max/IdMax/Min/IdMin#####
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
```

1.14 Contact Analysis (Part 3)

This analysis uses the data of tutorial /11_contact_2beam.

1.14.1 Analysis target

The analysis is a contact problem between two beams, and an overview of the analytical model is shown in Figure 4.11.1.

Item	Description	Notes	Reference
Type of analysis	Non-linear static analysis(elastoplastic,contact)	!SOLUTION,TYPE=NL- STATIC !CONTACT	
Number of nodes	252		

Item	Description	Notes	Reference
Number of elements	80		
Element type	Eight node hexahedral element	!ELEMENT,TYPE=361	
Material name	M1	!MATERIAL,NAME=M1	
Material property	ELASTIC,PLASTIC	!ELASTIC !PLASTIC	
Boundary conditions	Restraint,Force displacement		
Matrix solution	Direct method	!SOLVER,METHOD=MUMPS	

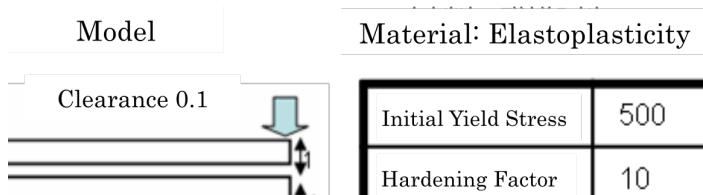


Fig. 4.11.1 Outline of Analysis Model

1.14.2 Analysis contents

The Lagrangian multiplier method is used to perform a contact analysis to give a forced displacement to the top surface of the upper beam. The analysis control data is shown below.

1.14.2.1 Analysis control data 2beams.cnt.

```
!!
!! Control File for FISTR
!!
!VERSION
3
!SOLUTION, TYPE=NLSSTATIC
!WRITE,RESULT
!WRITE,VISUAL
!BOUNDARY, GRPID=1
    ng1, 1, 3, 0.0
    ng2, 1, 3, 0.0
    ng3, 3, 3, -3.0
!CONTACT_ALGO, TYPE=SLAGRANGE
!CONTACT, GRPID=1, INTERACTION=FSLID
    CP1, 0.0, 1.0e+5
!STEP, SUBSTEPS=100, CONVERG=1.0e-4, MAXITER=1000
    BOUNDARY, 1
    CONTACT, 1
!MATERIAL, NAME=M1
!ELASTIC
    2.1e+5, 0.3
!PLASTIC, YIELD=MISES
    500.0, 10.0
!SOLVER,METHOD=MUMPS
## Post Control
!VISUAL, metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END
```

1.14.3 Analysis procedure

Execute the FrontISTR execution command `fistr1`.

```
$ cd FrontISTR/tutorial/11_contact_2beam
$ fistr1 -t 4
(Runs in 4 threads.)
```

1.14.4 Analysis results

The results of the 100th substep are shown in Figure 4.11.2. A deformation diagram with the Mises stress contours added is created by REVOCAP_PrePost. A part of the analysis results log file is shown below as numerical data for the analysis results.

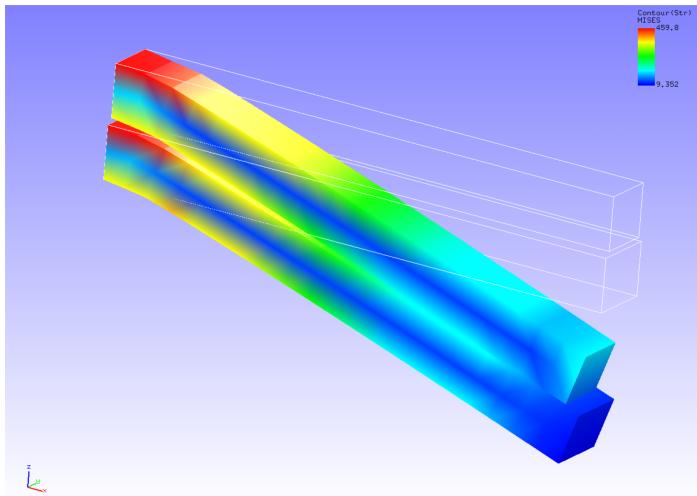


Fig. 4.11.2: Analysis results of deformation and Mises stress

1.14.4.1 Analysis results log 0.log.

```
fstr_setup: OK
##### Result step= 0
##### Local Summary @Node :Max/IdMax/Min/IdMin#####
//U1 0.0000E+00 1 0.0000E+00 1
//U2 0.0000E+00 1 0.0000E+00 1
//U3 0.0000E+00 1 0.0000E+00 1
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
//E33 0.0000E+00 1 0.0000E+00 1
//E12 0.0000E+00 1 0.0000E+00 1
//E23 0.0000E+00 1 0.0000E+00 1
//E31 0.0000E+00 1 0.0000E+00 1
//S11 0.0000E+00 1 0.0000E+00 1
//S22 0.0000E+00 1 0.0000E+00 1
//S33 0.0000E+00 1 0.0000E+00 1
//S12 0.0000E+00 1 0.0000E+00 1
//S23 0.0000E+00 1 0.0000E+00 1
//S31 0.0000E+00 1 0.0000E+00 1
//SMS 0.0000E+00 1 0.0000E+00 1
##### Local Summary @Element :Max/IdMax/Min/IdMin#####
//E11 0.0000E+00 1 0.0000E+00 1
//E22 0.0000E+00 1 0.0000E+00 1
```

1.15 Linear Dynamic Analysis

This analysis uses the data of tutorial /12_dynamic_beam.

1.15.1 Analysis target

The analysis target is a cantilevered beam, and the geometry is shown in Figure 4.12.1 and the mesh data is shown in Figure 4.12.2.

Item	Description	Notes	Reference
Type of analysis	Linear dynamic analysis	<code>!SOLUTION,TYPE=DYNAMIC</code> <code>!DYNAMIC,TYPE=LINEAR</code>	
Number of nodes	525		
Number of elements	240		
Element type	Ten node tetrahedral quadratic element	<code>!ELEMENT,TYPE=342</code>	
Material name	M1		
Boundary conditions	Restraint,Concentrated force	<code>!CLOAD</code>	
Matrix solution	CG/SSOR	<code>!SOLVER,METHOD=CG,PRE-COND=1</code>	

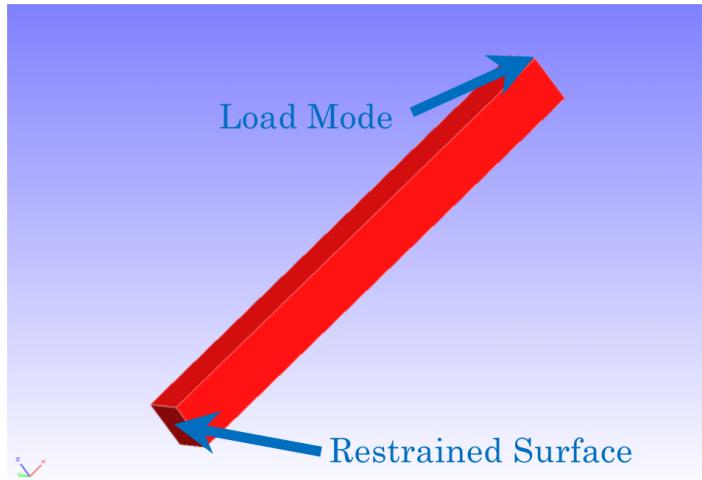


Fig. 4.12.1: Shape of the cantilever

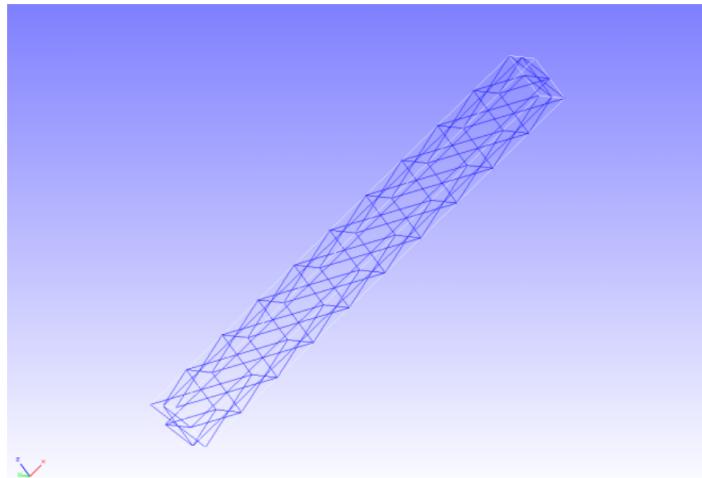


Fig. 4.12.2: Mesh data of the cantilever

1.15.2 Analysis contents

A linear dynamic analysis is performed after the displacement of the restraining surface shown in Figure 4.12.1 is restrained and a concentrated load is applied to the load nodes. The analytical control data are shown below.

1.15.2.1 Analysis control data beam.cnt.

```
# Control File for FISTR
## Analysis Control
!VERSION
3
!WRITE,LOG,FREQUENCY=5000
!WRITE,RESULT,FREQUENCY=5000
!SOLUTION, TYPE=DYNAMIC
!DYNAMIC, TYPE=LINEAR
11 , 1
0.0 , 1.0 , 500000 , 1.0000e-8
0.5 , 0.25
1 , 1 , 0.0 , 0.0
100000 , 3121 , 500
1 , 1 , 1 , 1 , 1 , 1
## Solver Control
### Boundary Condition
!BOUNDARY, AMP=AMP1
FIX , 1 , 3 , 0.0
!CLOAD, AMP=AMP1
CL1 , 3 , -1.0
### Material
# define in mesh file
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=NO,TIMELOG=NO
10000 , 1
1.0e-06 , 1.0 , 0.0
!END
```

1.15.3 Analysis procedure

Execute the FrontISTR execution command fistrl .

```
$ cd FrontISTR/tutorial/12_dynamic_beam
$ fistrl -t 4
(Runs in 4 threads.)
```

1.15.4 Analysis results

Figure 4.12.3 shows a time series of the displacement of a monitoring node (load node, 3121) specified in the analysis control data, created by Microsoft Excel. A part of the monitoring node displacement output file (dyna_disp_3121.txt) is shown below as numerical data for the analysis results.

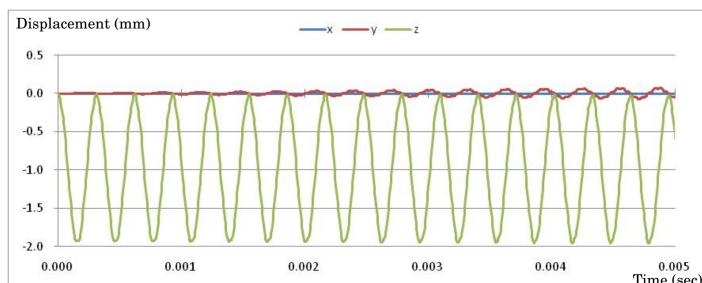


Fig. 4.12.3: Time-series displacement of monitoring nodes

0	0.0000E+000	3121	0.0000E+000	0.0000E+000	0.0000E+000
500	5.0000E-006	3121	5.3301E-005	-2.6682E-005	-1.5646E-002
1000	1.0000E-005	3121	4.0790E-005	-1.0696E-006	-4.4118E-002
1500	1.5000E-005	3121	9.1017E-005	5.7542E-005	-8.1017E-002
2000	2.0000E-005	3121	1.8944E-005	5.6499E-005	-1.2358E-001
2500	2.5000E-005	3121	3.4535E-005	6.1147E-005	-1.7787E-001
3000	3.0000E-005	3121	3.0248E-005	1.6211E-004	-2.2844E-001
3500	3.5000E-005	3121	4.2434E-005	1.1706E-004	-2.7330E-001
4000	4.0000E-005	3121	-2.0130E-005	1.2298E-004	-3.2436E-001
4500	4.5000E-005	3121	4.1976E-005	-4.2753E-005	-3.8902E-001
5000	5.0000E-005	3121	5.6526E-005	1.2043E-004	-4.6494E-001
5500	5.5000E-005	3121	1.9195E-005	8.8901E-006	-5.4673E-001
6000	6.0000E-005	3121	3.9722E-005	-8.0492E-005	-6.4665E-001
6500	6.5000E-005	3121	9.0688E-005	-1.9603E-004	-7.5697E-001
7000	7.0000E-005	3121	3.8175E-005	1.3406E-004	-8.6961E-001
7500	7.5000E-005	3121	-2.1776E-005	2.9617E-004	-9.6952E-001
8000	8.0000E-005	3121	-1.6732E-005	2.0223E-004	-1.0672E+000
8500	8.5000E-005	3121	1.0129E-004	4.9717E-004	-1.1583E+000
9000	9.0000E-005	3121	4.4797E-005	6.6073E-004	-1.2421E+000
9500	9.5000E-005	3121	-5.5023E-007	7.2865E-004	-1.3154E+000
10000	1.0000E-004	3121	4.6793E-005	3.6134E-004	-1.3947E+000

1.16 Non-Linear Dynamic Analysis

This analysis uses the data of tutorial /13_dynamic_beam_nonlinear.

1.16.1 Analysis target

The target of the analysis is the same cantilevered beam as the Linear Dynamic Analysis in the previous section.

Item	Description	Notes	Reference
Type of analysis	Nonlinear dynamic analysis	!SOLUTION,TYPE=DYNAMIC !DYNAMIC,TYPE=NONLINEAR	
Number of nodes	525		
Number of elements	240		
Element type	Ten node tetrahedral quadratic element	!ELEMENT,TYPE=342	
Material name	M1	!MATERIAL,NAME=M1	
Boundary conditions	Restraint,Concentrated force	!CLOAD	
Matrix solution	CG/SSOR	!SOLVER,METHOD=CG,PRE-COND=1	

1.16.2 Analysis content

A nonlinear dynamic analysis is performed after the displacement of the constrained surface shown in Figure 4.12.1 is constrained and a concentrated load is applied to the load nodes. The analytical control data are shown below.

1.16.2.1 Analysis control data beam.cnt.

```
# Control File for FISTR
## Analysis Control
!VERSION
3
!WRITE,RESULT,FREQUENCY=100
!SOLUTION, TYPE=DYNAMIC
!DYNAMIC, TYPE=NONLINEAR
```

```

1 , 1
0.0 , 0.1 , 100000 , 1.0000e-8
0.5 , 0.25
1 , 1 , 0.0 , 0.0
1000 , 3121 , 100
1 , 1 , 1 , 1 , 1 , 1
## Solver Control
### Boundary Condition
!BOUNDARY, GRPID=1, AMP=AMP1
FIX, 1 , 3 , 0.0
!CLOAD, GRPID=1, AMP=AMP1
CL1, 3, -1.0
### STEP
!STEP, CONVERG=1.0e-3
BOUNDARY, 1
LOAD, 1
### Material
!DENSITY
1.0e-8
!HYPERELASTIC, TYPE=NEOHOOKE
1000.0 , 0.00005
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=NO,TIMELOG=NO
10000 , 1
1.0e-06, 1.0 , 0.0
!END

```

1.16.3 Analysis procedure

Execute the FrontISTR execution command fistrl .

```

$ cd FrontISTR/tutorial/13_dynamic_beam_nonlinear
$ fistrl -t 4
(Runs in 4 threads.)

```

1.16.4 Analysis results

A time series of the displacement of a monitoring node (load node, node number 3121) specified in the analysis control data is shown in Figure 4.13.1, created by Microsoft Excel. A part of the monitoring node displacement output file (dyna_disp_3121.txt) is shown below as numerical data for the analysis results.

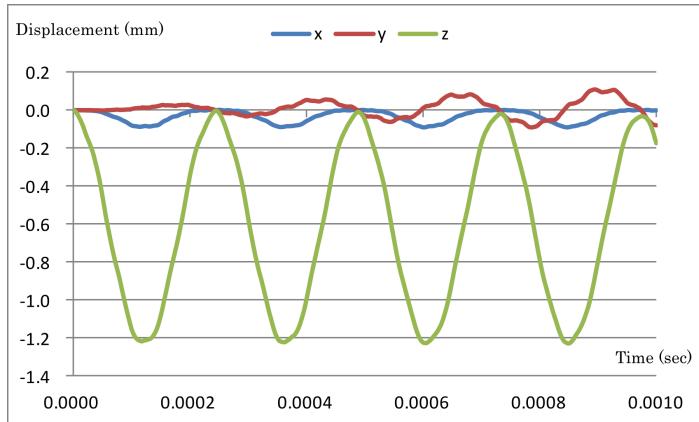


Fig. 4.13.1: Time-series displacement of monitoring nodes

1.16.4.1 Displacement of monitoring nodes dyna_disp_3121.txt.

100	1.0000E-006	3121	7.6885E-005	-7.3733E-005	-6.0988E-004
200	2.0000E-006	3121	3.3089E-005	-7.5879E-006	-8.2481E-004
300	3.0000E-006	3121	8.9272E-005	-5.6180E-005	-1.2550E-003
400	4.0000E-006	3121	5.8434E-005	-2.9113E-005	-1.9326E-003
500	5.0000E-006	3121	3.3598E-005	-3.7069E-005	-2.6955E-003
600	6.0000E-006	3121	9.2438E-005	-2.9415E-005	-3.4297E-003
700	7.0000E-006	3121	4.4742E-005	-1.9064E-005	-4.2128E-003
800	8.0000E-006	3121	4.2702E-005	-3.7315E-005	-5.2563E-003
900	9.0000E-006	3121	7.9468E-005	4.8283E-006	-6.1239E-003
1000	1.0000E-005	3121	2.5902E-005	-3.1393E-005	-7.1463E-003
1100	1.1000E-005	3121	6.9365E-005	-1.1486E-005	-8.3515E-003

1.17 Non-Linear Contact Dynamic Analysis

This analysis uses the data of tutorial /14_dynamic_plate_contact.

1.17.1 Analysis target

The object of the analysis was the fall impact analysis of a square material on a floor surface, and the geometry is shown in Fig. 4.14.1 and the mesh data is shown in Fig. 4.14.2.

Item	Description	Notes	Reference
Type of analysis	Nonlinear contact dynamic analysis	!SOLUTION,TYPE=DYNAMIC !DYNAMIC,TYPE=NONLINEAR !CONTACT	
Number of nodes	10,712		
Number of elements	8,232		
Element type	Eight node hexahedral element	!ELEMENT,TYPE=361	
Material name	M1, M2	!MATERIAL,NAME=M1 !MATERIAL,NAME=M2	
Material property	ELASTIC, PLASTIC	!ELASTIC !PLASTIC	
Boundary conditions	Restraint, Initial velocity	!VELOCITY,TYPE=INITIAL	
Matrix solution	Direct method	!SOLVER,METHOD=MUMPS	

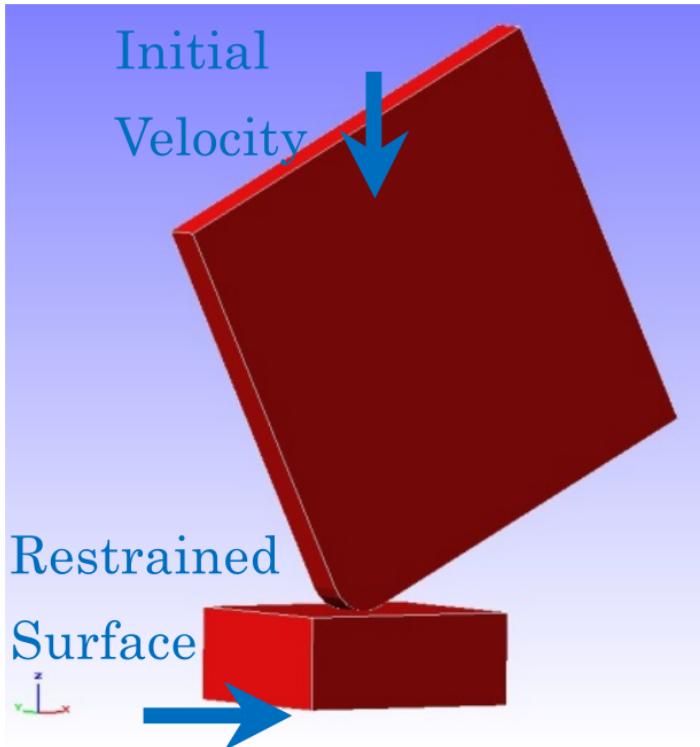


Fig. 4.14.1: Shape of the floor surface and square material

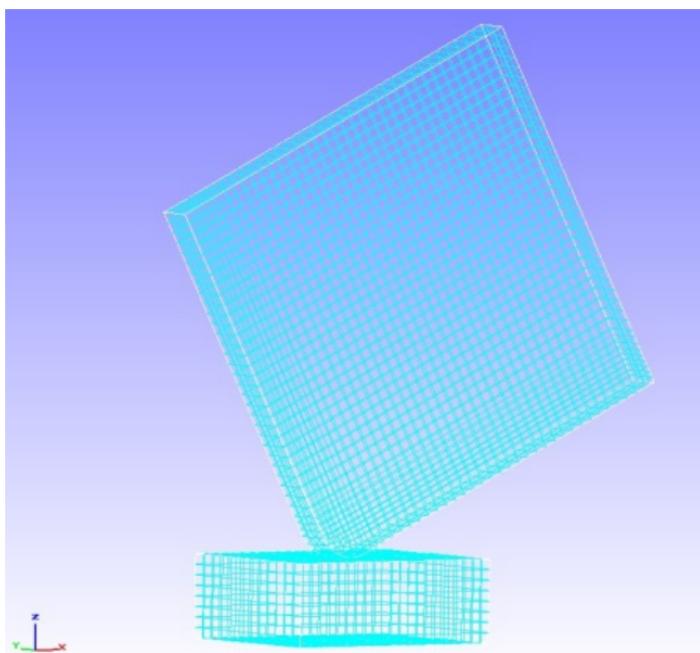


Fig. 4.14.2: Mesh data of the floor surface and square material

1.17.2 Analysis content

The initial speed of 4427 mm/s is set for the square material to be analyzed, and the contact motion analysis is performed. The analysis control data is shown below.

1.17.2.1 Analysis control data plateToGround.cnt.

```
!!  
!! Control File for FISTR  
!!
```

```

!VERSION
3
!WRITE,LOG,FREQUENCY=10
!WRITE,RESULT,FREQUENCY=10
!SOLUTION, TYPE=DYNAMIC
!DYNAMIC, TYPE=NONLINEAR
1 , 1
0.0 , 1.0 , 200, 1.0000e-8
0.65 , 0.330625
1, 1, 0.0, 0.0
20, 2621, 1
1, 1, 1, 1, 1, 1
!BOUNDARY, GRPID = 1
bottom , 1 , 3 , 0.0
!VELOCITY, TYPE = INITIAL
plate , 3 , 3 , -4427.0
!CONTACT_ALGO, TYPE=SLAGRANGE
!CONTACT, GRPID=1, INTERACTION=FSLID
CP1, 0.0 , 1.0e+5
!STEP, CONVERG=1.0e-8, ITMAX=100
BOUNDARY, 1
CONTACT, 1
!MATERIAL, NAME = M1
!ELASTIC
2.00000e+5, 0.3
!PLASTIC
1.0e+8, 0.0
!MATERIAL, NAME = M2
!ELASTIC
1.16992e+5, 0.3
!PLASTIC
70.0, 0.0
!SOLVER,METHOD=MUMPS
!END

```

1.17.3 Analysis Results

A contour diagram of the Mises stresses during the fall impact is shown in Figure 4.14.3. In addition, a portion of the energy output file (dyna_energy.txt) for the monitoring nodes is shown below as numerical data for the analysis results.

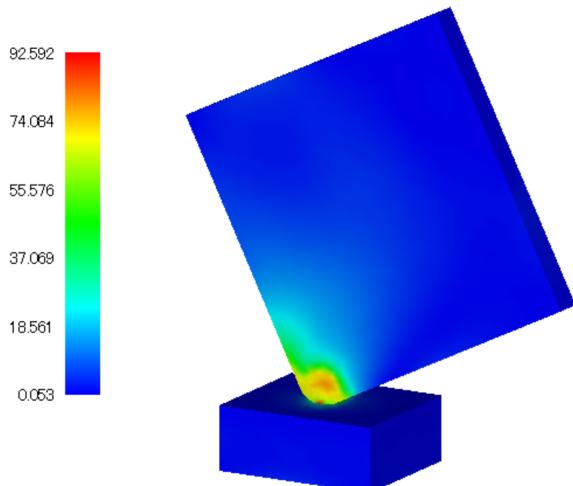


Fig. 4.14.3: Mises stress of the falling impact

1.17.3.1 Displacement of monitoring nodes dyna_energy.txt.

time	step	time	kinetic energy	strain energy	total energy
0	0.0000E+000		9.7816E-003	0.0000E+000	9.7816E-003
1	1.0000E-008		9.7756E-003	4.9470E-006	9.7806E-003
2	2.0000E-008		9.7654E-003	1.4636E-005	9.7800E-003
3	3.0000E-008		9.7566E-003	2.2609E-005	9.7792E-003
4	4.0000E-008		9.7505E-003	3.7965E-005	9.7884E-003
5	5.0000E-008		9.7425E-003	6.4932E-005	9.8074E-003
6	6.0000E-008		9.7214E-003	8.4571E-005	9.8060E-003
7	7.0000E-008		9.7139E-003	9.0613E-005	9.8045E-003
8	8.0000E-008		9.7184E-003	1.0958E-004	9.8280E-003
9	9.0000E-008		9.7175E-003	1.5717E-004	9.8747E-003
10	1.0000E-007		9.6909E-003	1.7998E-004	9.8709E-003
11	1.1000E-007		9.6917E-003	1.9733E-004	9.8890E-003
12	1.2000E-007		9.7137E-003	2.2403E-004	9.9377E-003
13	1.3000E-007		9.6813E-003	2.4397E-004	9.9253E-003

1.18 Eigenvalue Analysis

This analysis uses the data of tutorial /15_eigen_spring.

1.18.1 Analysis target

The objects of analysis are the same springs as in the Non-linear Static Analysis (Hyperelasticity, Part 2) in the previous section.

Item	Description	Notes	Reference
Type of analysis	Eigen value analysis	!SOLUTION,TYPE=EIGEN	
Number of nodes	78,771		
Number of elements	46,454		
Element type	Ten node tetrahedral quadratic element	!ELEMENT,TYPE=342	
Material name	MAT1	!MATERIAL,NAME=MAT1	
Boundary conditions	Restraint		
Matrix solution	Direct method	!SOLVER,METHOD=DIRECT	

1.18.2 Analysis content

The displacements of the constrained surfaces shown in Figure 4.4.1 are constrained, and eigenvalue analysis is performed up to the fifth order. The analysis control data is shown below.

1.18.2.1 Analysis control data spring.cnt.

```
# Control File for FISTR
## Analysis Control
!VERSION
 3
!SOLUTION, TYPE=EIGEN
!EIGEN
 5, 1.0E-8, 60
!WRITE,RESULT
!WRITE,VISUAL
## Solver Control
### Boundary Conditon
!BOUNDARY
 XFIX, 1, 1, 0.0
 YFIX, 2, 2, 0.0
 ZFIX, 3, 3, 0.0
### Material
```

```

# define in mesh file
### Solver Setting
!SOLVER,METHOD=DIRECT
## Post Control
!VISUAL,metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END

```

1.18.3 Analysis procedure

Execute the FrontISTR execution command fistr1 .

```

$ cd FrontISTR/tutorial/15_eigen_spring
$ fistr1 -t 4
(Runs in 4 threads.)

```

1.18.4 Analysis results

Using the resulting data file spring.res.0.3, the third order vibration mode (compression and extension of the spring in the y direction) is created by REVOCAP_PrePost and shown in Figure 4.15.1. The deformation factor is set to 1000. In addition, a list of natural frequencies output to the analysis results log file is shown below as numerical data for the analysis results.

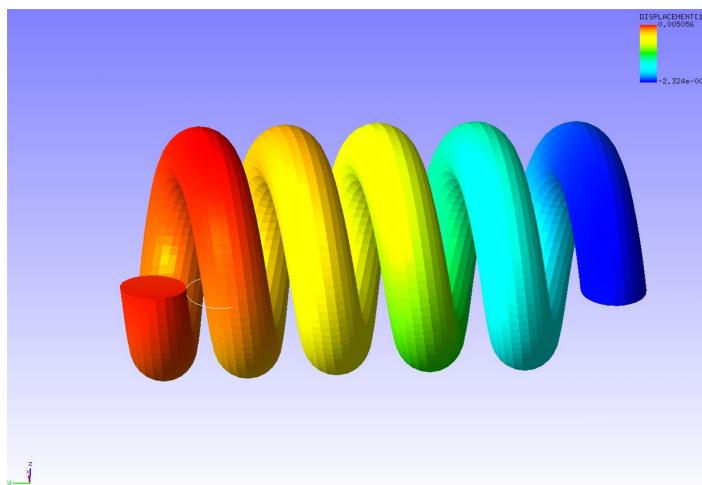


Fig. 4.15.1: Third vibration mode of a spring

1.18.4.1 Analysis results log 0.log.

fstr_setup : OK

```
*****
*RESULT OF EIGEN VALUE ANALYSIS*
*****
```

NUMBER OF ITERATIONS = 47
TOTAL MASS = 3.4184E-06

EFFECTIVE MASS		ANGLE	FREQUENCY	PARTICIPATION FACTOR		
NO.	EIGENVALUE	FREQUENCY	(HZ)	X	Y	Z
X	Y	Z				

```

 1  7.8307E+06  2.7983E+03  4.4537E+02  1.0289E+00 -8.8939E-02 -7.0520E-01
1.3006E-06  9.7176E-09  6.1094E-07
 2  7.8716E+06  2.8056E+03  4.4653E+02  6.9687E-01  1.0755E-01  1.0106E+00
6.1290E-07  1.4598E-08  1.2890E-06
 3  3.2600E+07  5.7097E+03  9.0872E+02  4.8622E-03  1.2364E+00 -7.9172E-02
4.0069E-11  2.5908E-06  1.0624E-08
 4  3.8366E+07  6.1940E+03  9.8581E+02 -2.9654E-02  3.3849E-01 -6.7819E-03
9.8232E-10  1.2799E-07  5.1379E-11
 5  1.2931E+08  1.1372E+04  1.8098E+03  5.1856E-01  4.7604E-02  6.7703E-01
2.8377E-07  2.3915E-09  4.8371E-07

```

Iter.#	Eigenvalue	Abs.	Residual
1	7.8307E+06	9.2347E-09	
2	7.8716E+06	7.8593E-08	
3	3.2600E+07	1.0084E-07	
4	3.8366E+07	5.3113E-08	
5	1.2931E+08	6.8762E-08	

1.19 Heat Conduction Analysis

This analysis uses the data of tutorial /16_heat_block.

1.19.1 Analysis target

The analysis target is a perforated block, the geometry is shown in Figure 4.16.1 and the mesh data is shown in Figure 4.16.2.

Item	Description	Notes	Reference
Type of analysis	Heat conduction analysis	!SOLUTION,TYPE=HEAT	
Number of nodes	37,386		
Number of elements	32,160		
Element type	Eight node hexahedral element	!ELEMENT,TYPE=361	
Material name	AL	!MATERIAL,NAME=AL	
Boundary conditions	Prescribed temperature	!FIXTEMP	
Matrix solution	CG/SSOR	!SOLVER,METHOD=CG,PRECOND=1	

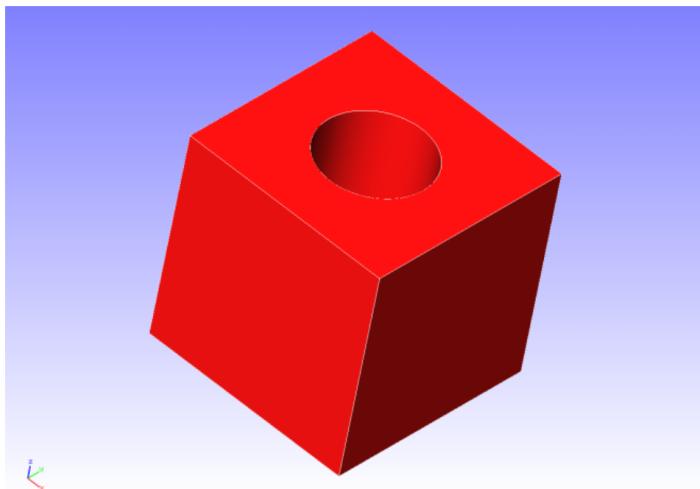


Fig. 4.16.1: Shape of the perforated block

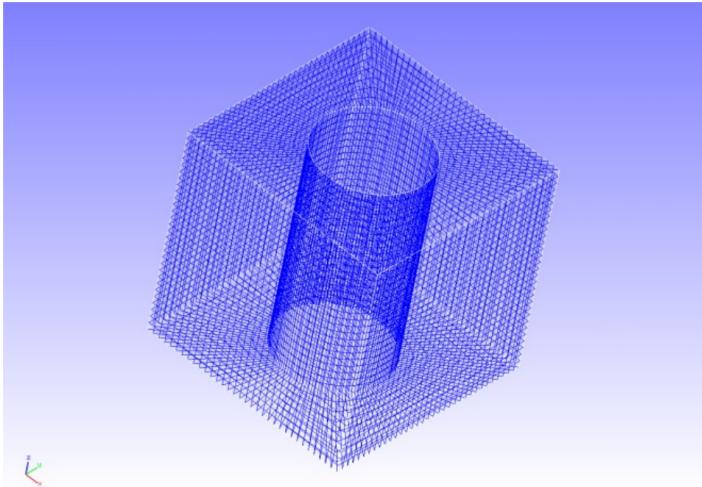


Fig. 4.16.2: Mesh data of the perforated block

1.19.2 Analysis content

Steady-state heat conduction analysis is performed to provide a heat source to the cylindrical inner surface of the object. The analysis control data are shown below.

1.19.2.1 Analysis control data block.cnt.

```
# Control File for FISTR
## Analysis Control
!VERSION
 3
!SOLUTION, TYPE=HEAT
!HEAT
 0.0
!WRITE,RESULT
!WRITE,VISUAL
## Solver Control
### Boundary Condition
!FIXTEMP
  FTMPC, 100.0
  FTMPS1, 20.0
  FTMPS2, 20.0
  FTMPS3, 20.0
  FTMPS4, 20.0
### Material
# define in mesh file
### Solver Setting
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=YES,TIMELOG=YES
  100, 1
  1.0e-8, 1.0, 0.0
## Post Control
!VISUAL,method=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END
```

1.19.3 Analysis procedure

Execute the FrontISTR execution command fistr1 .

```

$ cd FrontISTR/tutorial/16_heat_block
$ fistr1 -t 4
(Runs in 4 threads.)

```

1.19.4 Analysis results

A temperature contour plot was created by REVOCAP_PrePost and is shown in Figure 4.16.3. A part of the analysis results log file is shown below as numerical data for the analysis results.

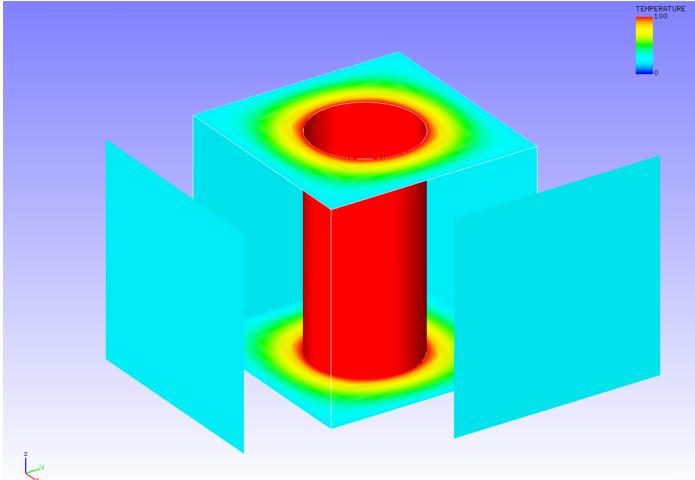


Fig. 4.16.3: Analysis results of temperature

1.19.4.1 Analysis results log 0.log.

fstr_setup : OK

```

ISTEP =      1
Time  =     0.000
Maximum Temperature :    100.000
Maximum Node No.   :        9
Minimum Temperature :    20.000
Minimum Node No.   :       85
Maximum Temperature(global) : 100.000
Minimum Temperature(global) : 20.000

```

1.20 Frequency Response Analysis

This analysis uses the data of tutorial /17_freq_beam.

1.20.1 Analysis target

The analysis target is a cantilevered beam, and the geometry is shown in Figure 4.17.1 and the mesh data is shown in Figure 4.17.2.

Item	Description	Notes	Reference
Type of analysis	Frequency response analysis	!SOLUTION,TYPE=EIGEN !SOLUTION,TYPE=DYNAMIC	
Number of nodes	55		
Number of elements	126		
Element type	Four node tetrahedral element	!ELEMENT,TYPE=341	
Material name	Material-1	!MATERIAL,NAME=Material-1	

Item	Description	Notes	Reference
Boundary conditions	Restraint, Concentrated force, eigen value	!EIGENREAD	
Matrix solution	CG/SSOR	!SOLVER,METHOD=CG,PRECOND=1	

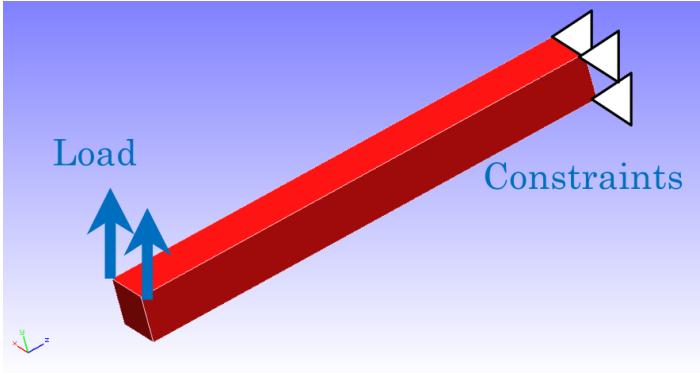


Fig. 4.17.1 : Shape of the cantilever

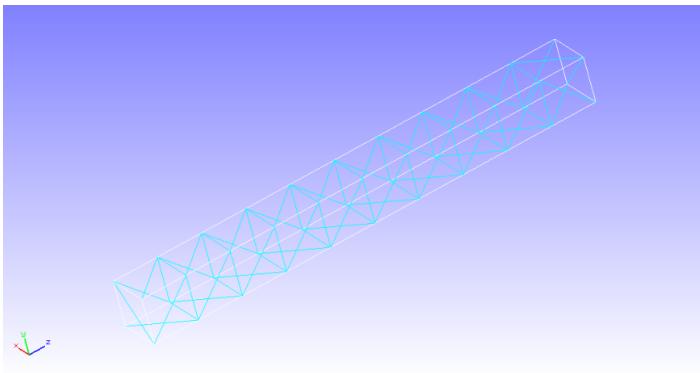


Fig. 4.17.2 : Mesh data of the cantilever

1.20.2 Analysis content

The end of a cantilevered beam to be analyzed is fully constrained, and a frequency response analysis is performed by applying concentrated loads to two nodes at the opposite end.

After analyzing eigenvalues up to the 10th order under the same boundary conditions, the analysis is performed using eigenvalues and eigenvectors up to the 5th order. The analysis control data for frequency response analysis is shown below.

1.20.2.1 Analysis control data `beam_eigen.cnt`.

```
# Control File for FISTR
!VERSION
3
!WRITE,RESULT
!WRITE,VISUAL
!SOLUTION, TYPE=EIGEN
!EIGEN
10, 1.0E-8, 60
!BOUNDARY
_PickedSet4, 1, 3, 0.0
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=NO,TIMELOG=YES
10000, 1
```

```

1.0e-8, 1.0, 0.0
!VISUAL, metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END

```

1.20.2.2 Analysis control data beam_freq.cnt.

```

# Control File for FISTR
!VERSION
3
!WRITE,RESULT
!WRITE,VISUAL
!SOLUTION, TYPE=DYNAMIC
!DYNAMIC
11 , 2
14000, 16000, 20, 15000.0
0.0 , 6.6e-5
1, 1, 0.0 , 7.2E-7
10, 2, 1
1, 1, 1, 1, 1, 1
!EIGENREAD
eigen_0.log
1, 5
!BOUNDARY
_PickedSet4 , 1, 3, 0.0
!FLOAD, LOAD CASE=2
_PickedSet5 , 2, 1.
!FLOAD, LOAD CASE=2
_PickedSet6 , 2, 1.
!SOLVER,METHOD=CG,PRECOND=1,ITERLOG=NO,TIMELOG=YES
10000 , 1
1.0e-8, 1.0, 0.0
!VISUAL, metod=PSR
!surface_num=1
!surface 1
!output_type=VTK
!END

```

1.20.3 Analysis procedure

First, change hecmw_ctrl_eigen.dat to hecmw_ctrl.dat for eigenvalue analysis, and then run eigenvalue analysis.

Next, change hecmw_ctrl_freq.dat to hecmw_ctrl.dat and 0.log to eigen_0.log (which is specified in the control data for frequency response analysis), and then perform the frequency response analysis.

```

$ cp hecmw_ctrl_eigen.dat hecmw_ctrl.dat
$ fistr1 -t 4
$ mv 0.log eigen_0.log
$ cp hecmw_ctrl_freq.dat hecmw_ctrl.dat
$ fistr1 -t 4

```

1.20.4 Analysis results

The relationship between the frequency and displacement amplitude of a monitoring node (node number 1) specified in the analysis control data is shown in Figure 4.17.3, created using Microsoft Excel. A part of the analysis result log file is shown below as numerical data for the analysis results.

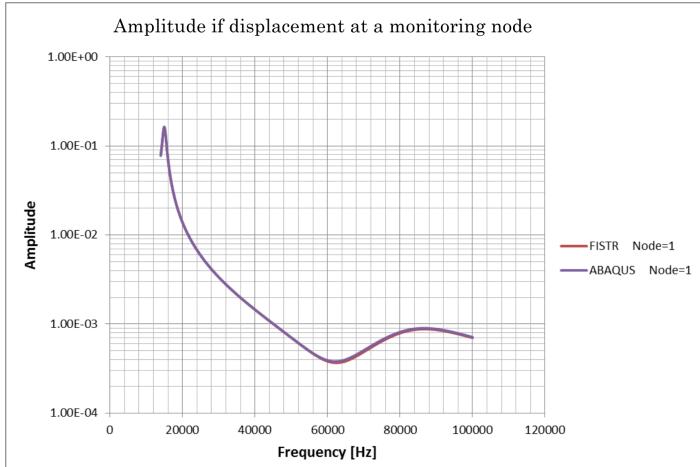


Fig.4.17.3 Relationship between frequency and displacement amplitude of the monitoring nodes

1.20.4.1 Analysis result log 0.log

```
fstr_setup: OK
Rayleigh alpha: 0.000000000000000
Rayleigh beta: 7.199999999999999E-007
read from=eigen_0.log
start mode= 1
end mode= 5
start frequency: 14000.00000000000
end frequency: 16000.00000000000
number of the sampling points 20
monitor nodeid= 1
 14100.000000000000 [Hz] : 8.3935554530220141E-002
 14100.000000000000 [Hz] : 1 .res
 14200.000000000000 [Hz] : 9.1211083510158913E-002
 14200.000000000000 [Hz] : 2 .res
 14300.000000000000 [Hz] : 9.9579777897537178E-002
 14300.000000000000 [Hz] : 3 .res
 14400.000000000000 [Hz] : 0.10914967595035865
 14400.000000000000 [Hz] : 4 .res
 14500.000000000000 [Hz] : 0.11992223203402431
 14500.000000000000 [Hz] : 5 .res
```

1.21 Verification by Simple-Shaped Model

1.21.1 Elastic static analysis

This verification was performed with a mesh-divided cantilever, as shown in Fig. 9.1.1. The analysis was performed with seven load conditions, exA-exG, as illustrated in Fig. 9.1.2. Please note that exG has the same load conditions as those of exA using the direct method solver.

The verification result of each load condition is presented in Tables 9.1.1–9.1.7.

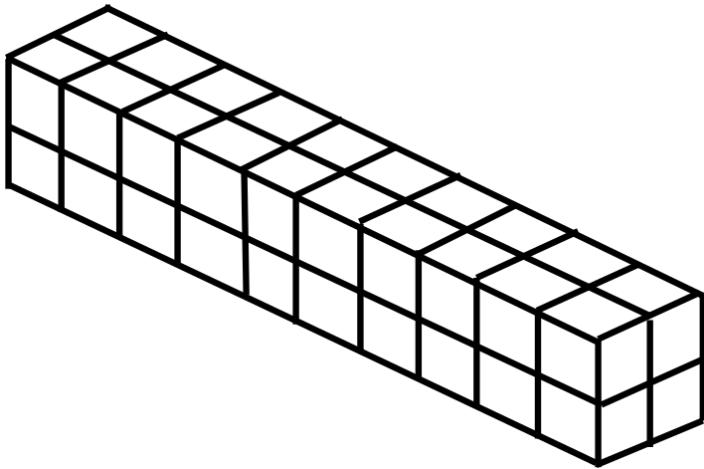
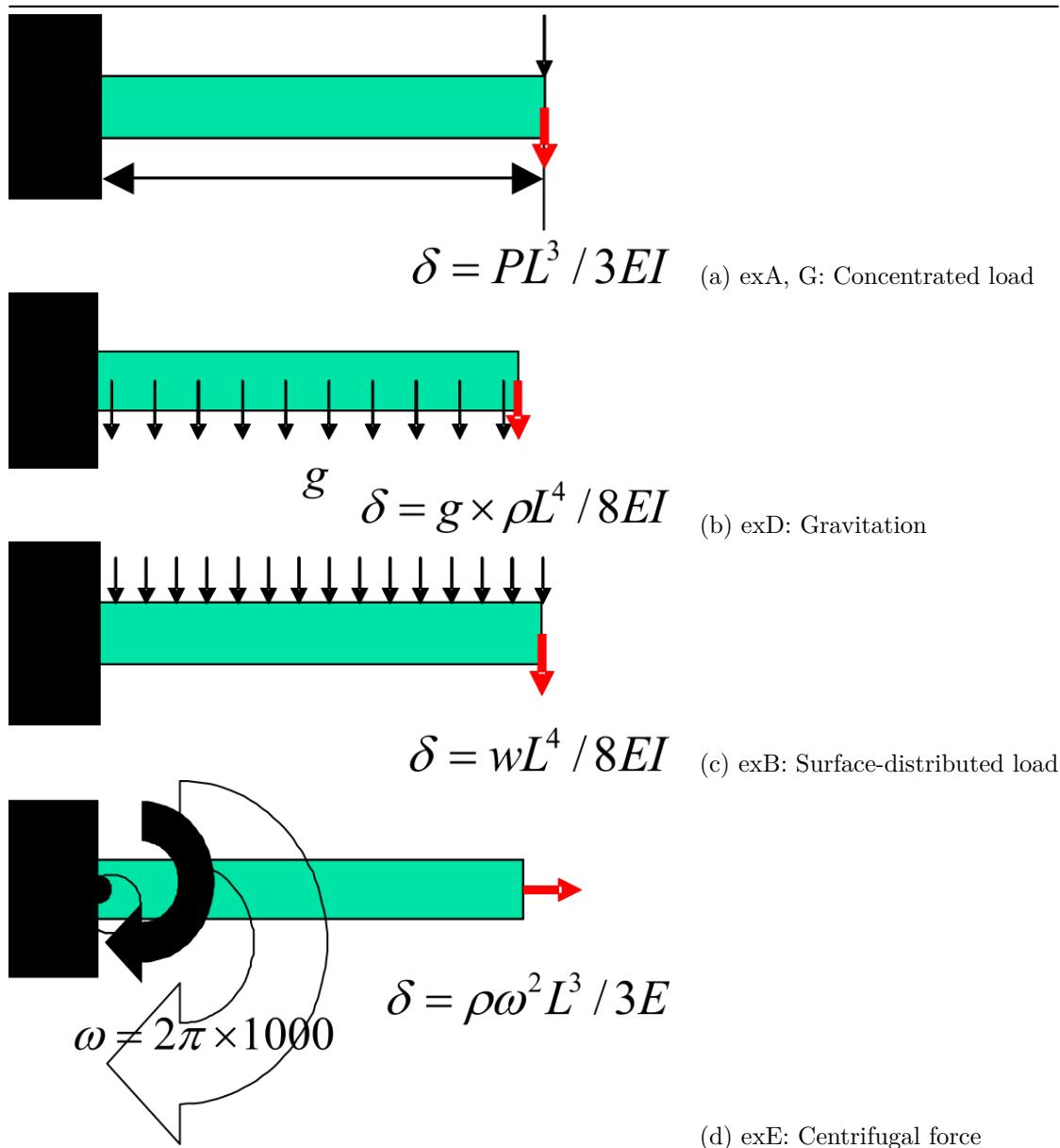
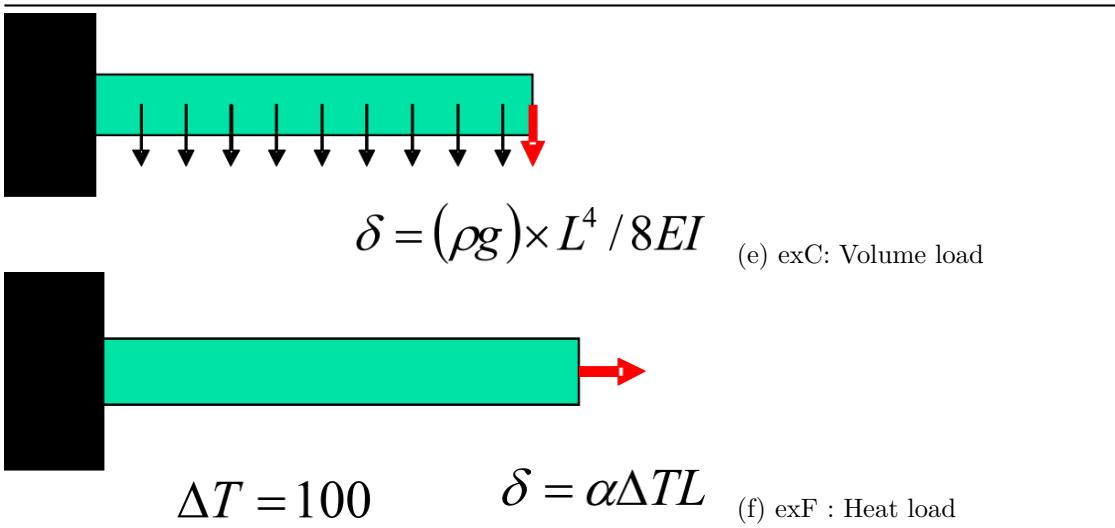


Fig. 9.1.1: Example of Mesh Partitioned Cantilever Beam (Hexahedral Element)





Item	Value
Young's Modulus	$E = 4000.0 \text{ kgf/mm}^2$
Length	$L = 10.0 \text{ mm}$
Poisson's Ratio	$\nu = 0.3$
Sectional area	$A = 1.0 \text{ mm}^2$
Mass density	$\rho = 8.0102 \times 10^{-10} \text{ kg s}^2/\text{mm}^4$
Second moment of area	$I = 1.0/12.0 \text{ mm}^4$
Gravitational acceleration	$g = 9800.0 \text{ mm/s}^2$
Linear coefficient of thermal expansion	$\alpha = 1.0 \times 10^{-5}$

Fig. 9.1.2: Verification conditions of the cantilever model

Table 9.1.1: exA: Verification results of the concentrated load problem

Case Name	Number of elements	NAS-TRAN	Predi-cated Value : $\delta_{max} = -1.000$	ABAQUS	Fron-tISTR	Remarks
A231	40	-0.338	-0.371	-0.371		33 nodes / plane stress status problem
A232	40	-0.942	-1.002	-1.002		105 nodes / plane stress status problem
A241	20	-0.720	-0.711	-0.711		33 nodes / plane stress status problem
A242	20	-0.910	-1.002	-1.002		85 nodes / plane stress status problem
A341	240	-0.384	-0.384	-0.386		99 nodes
A342	240	-0.990	-0.990	-0.999		525 nodes
A351	80	-0.353	-0.355	-0.351		99 nodes
A352	80	-0.993	-0.993	-0.992		381 nodes
A361	40	-0.954	-0.985	-0.984		99 nodes

Case Name	Number of elements	Predicated Value : $\delta_{max} =$ -1.000			Remarks
A362	40	-0.994	-0.993	-0.993	220 nodes
A731	40	-	-	-0.991	33 nodes / direct method
A741	20	-	-	-0.996	33 nodes / direct method

Table 9.1.2: exB: Verification results of the surface-distributed load problem

Case name	Number of elements	NASTRAN	ABAQUS	FrontISTR	Predicated value : $\delta_{max} =$ -3.750	Remarks
B231	40	-1.281	-1.403	-1.403		33 nodes / plane stress status problem
B232	40	-3.579	-3.763	-3.763		105 nodes / plane stress status problem
B241	20	-3.198	-2.680	-2.680		33 nodes / plane stress status problem
B242	20	-3.426	-3.765	-3.765		85 nodes / plane stress status problem
B341	240	-1.088	-1.449	-1.454		99 nodes
B342	240	-3.704	-3.704	-3.748		525 nodes
B351	80	-3.547	-1.338	-1.325		99 nodes
B352	80	-0.3717	-3.716	-3.713		381 nodes
B361	40	-3.557	-3.691	-3.688		99 nodes
B362	40	-3.726	-3.717	-3.717		220 nodes
B731	40	-	-	-3.722		33 nodes / direct method
B741	20	-	-	-3.743		33 nodes / direct method

Table 9.1.3: exC: Verification results of the volume load problem

Case Name	Number of elements	NASTRAN	ABAQUS	FrontISTR	Predicated Value : $\delta_{max} =$ -2.944^{-5}	Remarks
C231	40	-	-1.101e-5	-1.101e-5		33 nodes / plane stress problem
C232	40	-	-2.951e-5	-2.951e-5		105 nodes / plane stress problem
C241	20	-	-2.102e-5	-2.102e-5		33 nodes / plane stress problem

Case Name	Number of elements		Predicated Value : $\delta_{max} = -2.944^{-5}$		Remarks
C242	20	-	-2.953e-5	-2.953e-5	85 nodes / plane stress problem
C341	240	-	-1.136e-5	-1.140e-5	99 nodes
C342	240	-	-2.905e-5	-2.937e-5	525 nodes
C351	80	-	-1.050e-5	-1.039e-5	99 nodes
C352	80	-	-2.914e-5	-2.911e-5	381 nodes
C361	40	-	-2.895e-5	-2.893e-5	99 nodes
C362	40	-	-2.915e-5	-2.915e-5	220 nodes
C731	40	-	-	-2.922e-5	33 nodes / direct method
C741	20	-	-	-2.938e-5	33 nodes / direct method

Table 9.1.4: exD: Verification results of the gravitation problem

Case name	Number of elements	NASTRAN	ABAQUS	Predicated Value : $\delta_{max} = -2.944^{-5}$	Remarks
D231	40	-1.101e-5	-1.101e-5	-1.101e-5	33 nodes / plane stress status problem
D232	40	-2.805e-5	-2.951e-5	-2.951e-5	105 nodes / plane stress status problem
D241	20	-2.508e-5	-2.102e-5	-2.102e-5	33 nodes / plane stress status problem
D242	20	-2.684e-5	-2.953e-5	-2.953e-5	85 nodes / plane stress status problem
D341	240	-1.172e-5	-1.136e-5	-1.140e-5	99 nodes
D342	240	-2.906e-5	-2.905e-5	-2.937e-5	525 nodes
D351	80	-1.046e-5	-1.050e-5	-1.039e-5	99 nodes
D352	80	-2.917e-5	-2.914e-5	-2.911e-5	381 nodes
D361	40	-2.800e-5	-2.895e-5	-2.893e-5	99 nodes
D362	40	-2.919e-5	-2.915e-5	-2.915e-5	220 nodes
D731	40	-	-	-2.922e-5	33 nodes / direct method
D741	20	-	-	-2.938e-5	33 nodes / direct method

Table 9.1.5: exE: Verification results of the centrifugal force problem

Case name	Number of elements	NASTRAN	ABAQUS	Predicated value : $\delta_{max} = 2.635^{-3}$	Remarks
E231	40	2.410e-3	2.616e-3	2.650e-3	33 nodes / plane stress status problem

Case name	Number of elements		Predicated value : $\delta_{max} = 2.635^{-3}$		Remarks
E232	40	2.447e-3	2.627e-3	2.628e-3	105 nodes / plane stress status problem
E241	20	2.386e-3	2.622e-3	2.624e-3	33 nodes / plane stress status problem
E242	20	2.387e-3	2.627e-3	2.629e-3	85 nodes / plane stress status problem
E341	240	2.708e-3	2.579e-3	2.625e-3	99 nodes
E342	240	2.639e-3	2.614e-3	2.638e-3	525 nodes
E351	80	2.642e-3	2.598e-3	2.625e-3	99 nodes
E352	80	2.664e-3	2.617e-3	2.616e-3	381 nodes
E361	40	2.611e-3	2.603e-3	2.603e-3	99 nodes
E362	40	2.623e-3	2.616e-3	2.616e-3	220 nodes
E731	40	-	-	2.619e-3	33 nodes / direct method
E741	20	-	-	2.622e-3	33 nodes / direct method

Table 9.1.6: exF: Verification results of the thermal stress load problem

Case name	Number of elements	NASTRAN	Predicated Value : $\delta_{max} = 1.000^{-2}$	ABAQUS	FrontISTR	Remarks
F231	40	-	1.016e-2	1.007e-2	1.007e-2	33 nodes / plane stress status problem
F232	40	-	1.007e-2	1.007e-2	1.007e-2	105 nodes / plane stress status problem
F241	20	-	1.010e-2	1.010e-2	1.010e-2	33 nodes / plane stress status problem
F242	20	-	1.006e-2	1.006e-2	1.006e-2	85 nodes / plane stress status problem
F341	240	-	1.047e-2	1.083e-2	1.083e-2	99 nodes
F342	240	-	1.018e-2	1.022e-2	1.022e-2	525 nodes
F351	80	-	1.031e-2	1.062e-2	1.062e-2	99 nodes
F352	80	-	1.015e-2	1.017e-2	1.017e-2	381 nodes
F361	40	-	1.026e-2	1.026e-2	1.026e-2	99 nodes
F362	40	-	1.016e-2	1.016e-2	1.016e-2	220 nodes

Table 9.1.7: exG: Verification results of the direct method (concentrated load problem)

Case name	Number of elements	Predicated value : $\delta_{max} = -1.000$			Remarks
		NASTRAN	ABAQUS	FrontISTR	
G231	40	-0.338	-0.371	-0.371	33 nodes / plane stress status problem
G232	40	-0.942	-1.002	-1.002	105 nodes / plane stress status problem
G241	20	-0.720	-0.711	-0.711	33 nodes / plane stress status problem
G242	20	-0.910	-1.002	-1.002	85 nodes / plane stress status problem
G341	240	-0.384	-0.384	-0.386	99 nodes
G342	240	-0.990	-0.990	-0.999	52 nodes
G351	80	-0.353	-0.355	-0.351	99 nodes
G352	80	-0.993	-0.993	-0.992	381 nodes
G361	40	-0.954	-0.985	-0.984	99 nodes
G362	40	-0.994	-0.993	-0.993	220 nodes
G731	40	-	-	-0.991	33 nodes / direct method
G741	20	-	-	-0.996	33 nodes / dierct method

1.21.2 Non-linear static analysis

1.21.2.1 (2-1) exnl1: Geometrical non-linear analysis

The verification model of exI is the same as those of exA–G. A schematic diagram of the verification model is shown in Fig. 9.1.3. A geometric non-linear analysis was performed on this model. The verification results are presented in Table 9.1.8.

The non-linear calculation is a ten-step process with an increment value of 0.1 P and a final load of 1.0 P.



Fig. 9.1.3: Verification model

Table 9.1.8: exI: Verification results (maximum deflection amount log)

Case Name	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	1.0	Linear Solution
I231	-	-	-	-	-	-	-	-	-	-	-
I232	-	-	-	-	-	-	-	-	-	-	-
I241	-	-	-	-	-	-	-	-	-	-	-
I242	-	-	-	-	-	-	-	-	-	-	-
I341	0.039	0.077	0.116	0.154	0.193	0.232	0.270	0.309	0.348	0.386	0.386
I342	0.099	0.200	0.300	0.400	0.499	0.599	0.698	0.797	0.896	0.995	0.999
I351	0.035	0.070	0.105	0.141	0.176	0.211	0.246	0.281	0.316	0.351	0.351
I352	0.099	0.198	0.298	0.397	0.496	0.595	0.693	0.792	0.890	0.987	0.992
I361	0.070	0.139	0.209	0.278	0.348	0.417	0.487	0.556	0.625	0.694	0.984
I362	0.099	0.197	0.298	0.397	0.496	0.595	0.694	0.793	0.891	0.988	0.993

1.21.2.2 (2-2) exnl2: Elastoplasticity deformation analysis

Based on the test conducted at National Agency for Finite Element Methods and Standards (NAFEMS; U.K.): Test NL1, this verification problem was verified by elastoplastic deformation analysis that incorporated geometric non-linearity and multiple hardening rules. The elastoplastic deformation analysis model is shown in Fig. 9.1.4.

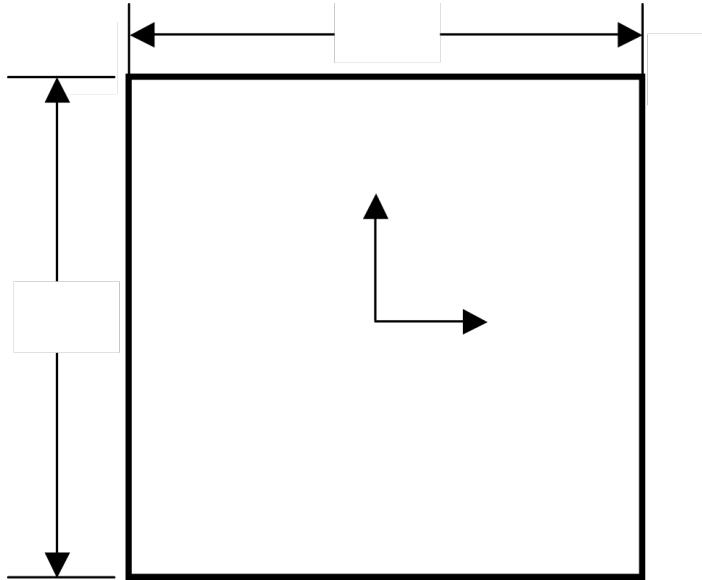


Fig. 9.1.4: Elastoplasticity deformation analysis Model

(1) Verification conditions:

Item	Value
Material	Mises elastoplastic material
Young's Modulus	$E = 250GPa$
Poisson's Ratio	$\nu = 0.25$
Initial yield stress	$5MPa$
Initial yield strain	0.25×10^{-4}
Isotropic hardening factor	$H_i = 0$ or $62.5GPa$

(2) Boundary conditions

Item	Boundary conditions	Value
step 1	Forced displacement in nodes 2 and 3	$u_x = 0.2500031251 * 10^{-4}$
step 2	Forced displacement in nodes 2 and 3	$u_x = 0.25000937518 * 10^{-4}$
step 3	Forced displacement in nodes 3 and 4	$u_y = 0.2500031251 * 10^{-4}$
step 4	Forced displacement in nodes 3 and 4	$u_y = 0.25000937518 * 10^{-4}$
step 5	Forced displacement in nodes 2 and 3	$u_x = -0.25000937518 * 10^{-4}$
step 6	Forced displacement in nodes 2 and 3	$u_x = -0.2500031251 * 10^{-4}$
step 7	Forced displacement in nodes 3 and 4	$u_y = -0.25000937518 * 10^{-4}$
step 8	Forced displacement in nodes 3 and 4	$u_y = -0.2500031251 * 10^{-4}$

All the nodes that are not mentioned here are fully constrained. The theoretical solution for this problem is presented as follows:

Strain ($\times 10^{-4}$) $[\varepsilon_x, \varepsilon_y, \varepsilon_z]$	Equivalent Stress (MPa) $[H_i = 0 H_k = 0; H_i = 62.5 H_k = 0]$
0.25, 0, 0	5.0; 5.0
0.50, 0, 0	5.0; 5.862
0.50, 0.25, 0	5.0; 5.482
0.50, 0.50, 0	5.0; 6.362

Strain ($\times 10^{-4}$) [$\varepsilon_x, \varepsilon_y, \varepsilon_z$]	Equivalent Stress (MPa) [$H_i = 0 H_k = 0; H_i = 62.5 H_k = 0$]
0.25, 0.50, 0	5.0; 6.640
0, 0.50, 0	5.0; 7.322
0, 0.25, 0	3.917; 4.230
0, 0, 0	5.0; 5.673

The results of the calculations are as follows:

Strain ($\times 10^{-4}$) [$\varepsilon_x, \varepsilon_y, \varepsilon_z$]	Equivalent Stress (MPa) [$H_i = 0 H_k = 0; H_i = 62.5 H_k = 0$]
0.25, 0, 0	5.0 (0.0%); 5.0 (0.0%)
0.50, 0, 0	5.0 (0.0%); 5.862 (0.0%)
0.50, 0.25, 0	5.0 (0.0%); 5.482 (0.0%)
0.50, 0.50, 0	5.0 (0.0%); 6.362 (-0.05%)
0.25, 0.50, 0	5.0 (0.0%); 6.640 (-0.21%)
0, 0.50, 0	5.0 (0.0%); 7.322 (-0.34%)
0, 0.25, 0	3.824 (-2.4%); 4.230 (-2.70%)
0, 0, 0	5.0 (0.0%); 5.673 (5.673 (-2.50%))

1.21.3 Contact analysis (1)

Based on the contact patch test problem (CGS-4) from NAFEMS (U.K.), this verification problem tests the finite sliding contact problem function with friction. The contact analysis model is shown in Fig. 9.1.5

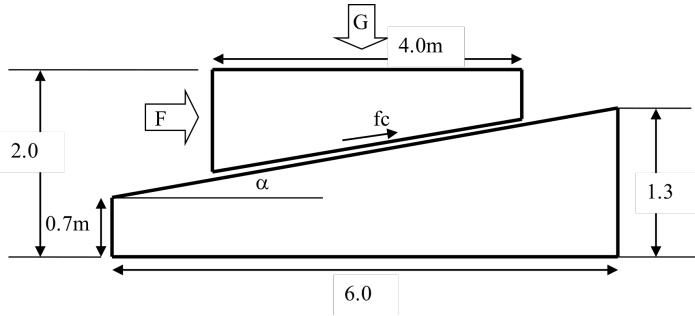


Fig. 9.1.5: Contact analysis model

The equilibrium condition of this problem is as follows:

$$F \cos \alpha - G \sin \alpha = \pm f_c$$

In the adhesive friction stage, the frictional force is as follows:

$$f_c = E_t \Delta u$$

In the sliding friction stage, the frictional force is as follows:

$$f_c = \mu(G \cos \alpha + F \sin \alpha)$$

The comparison between the calculation results and the analysis solution is as follows.

μ	F/G Analysis Solution	F/G Calculation Results
0.0	0.1	0.1
0.1	0.202	0.202
0.2	0.306	0.306
0.3	0.412	0.412

1.21.4 Contact analysis (2): Hertz contact problem

In this verification, the Hertz contact problem between a cylinder of infinite length and an infinite plane was analyzed. The cylinder's radius was $R=8$ mm, and the deformable body's Young's modulus E and Poisson's ratio μ were 1100 MPa and 0.0, respectively. Moreover, assuming that the contact area was much smaller than the cylinder's radius, and also considering the symmetry of the problem, the analysis was performed with a quarter model of the cylinder.

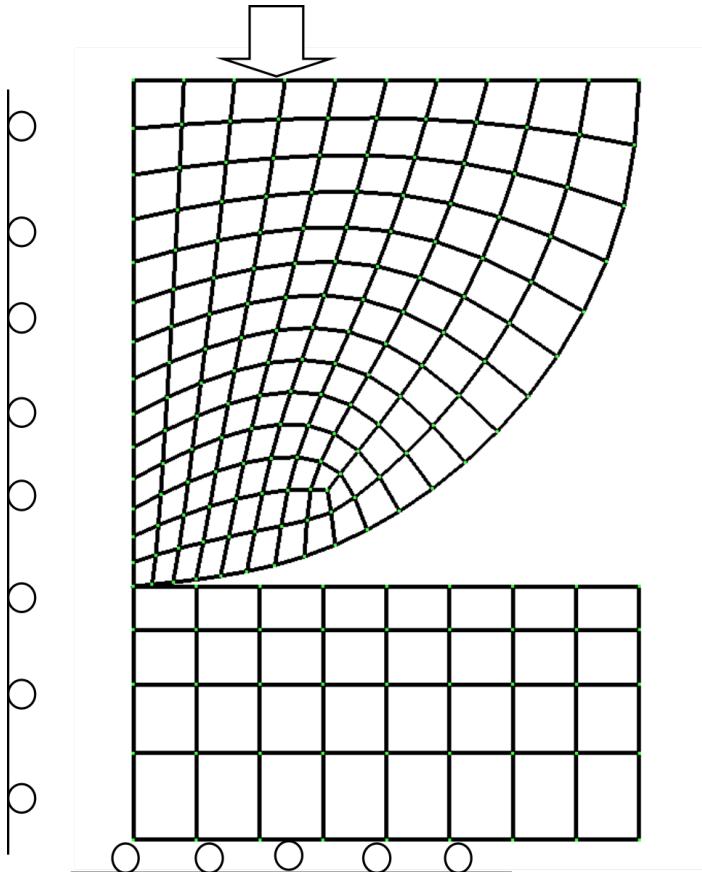


Fig. 9.1.6: Hertz contact problem analysis model

1.21.4.1 (1) Verification results of contact radius

The theoretical formula to calculate the contact radius is as follows:

$$a = \sqrt{\frac{4FR}{\pi E^*}}$$

where

$$E^* = \frac{E}{2(1 - \mu^2)}$$

With the actual calculation, when the pressure is $F = 100$, the contact radius is $a = 1.36$.

Fig. 9.1.7 shows the equivalent nodal force of the contact point. The contact radius is obtained by extrapolating this nodal force distribution.

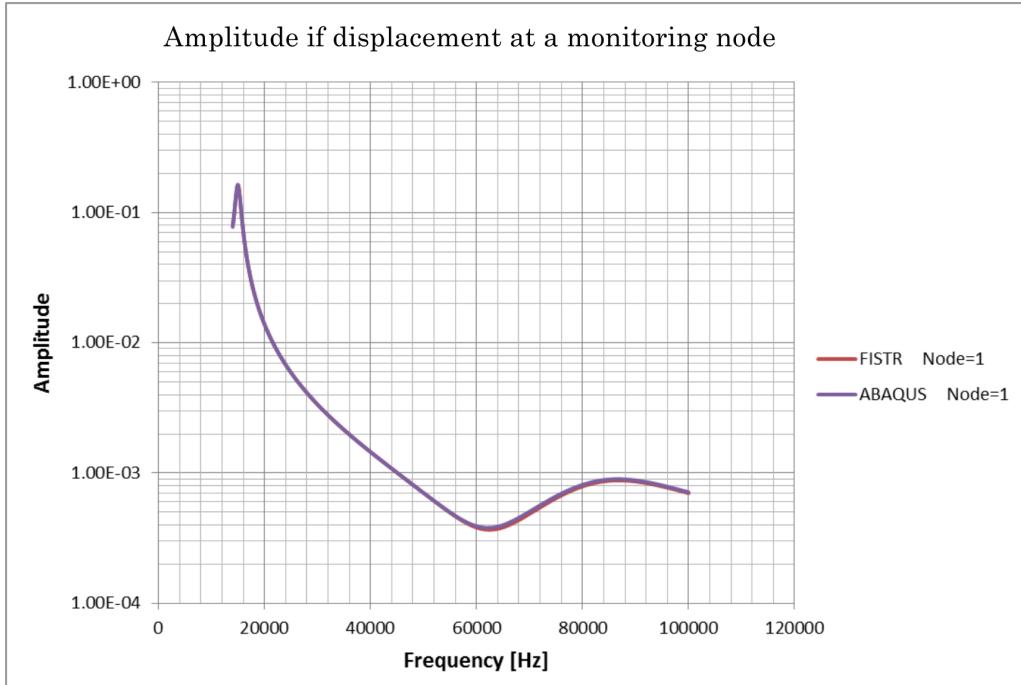


Fig. 9.1.7: Equivalent nodal force distribution of the contact point

1.21.4.2 (2) Verification results of maximum shear stress

With the theoretical solution, when the contact position is $z = 0.78a$, the maximum shear stress is as follows:

$$\tau_{max} = 0.30 \sqrt{\frac{FE^*}{\pi R}}$$

With the actual calculation conditions, it becomes

$$\tau_{max} = 14.2$$

The actual result obtained was

$$\tau_{max} = 15.6$$

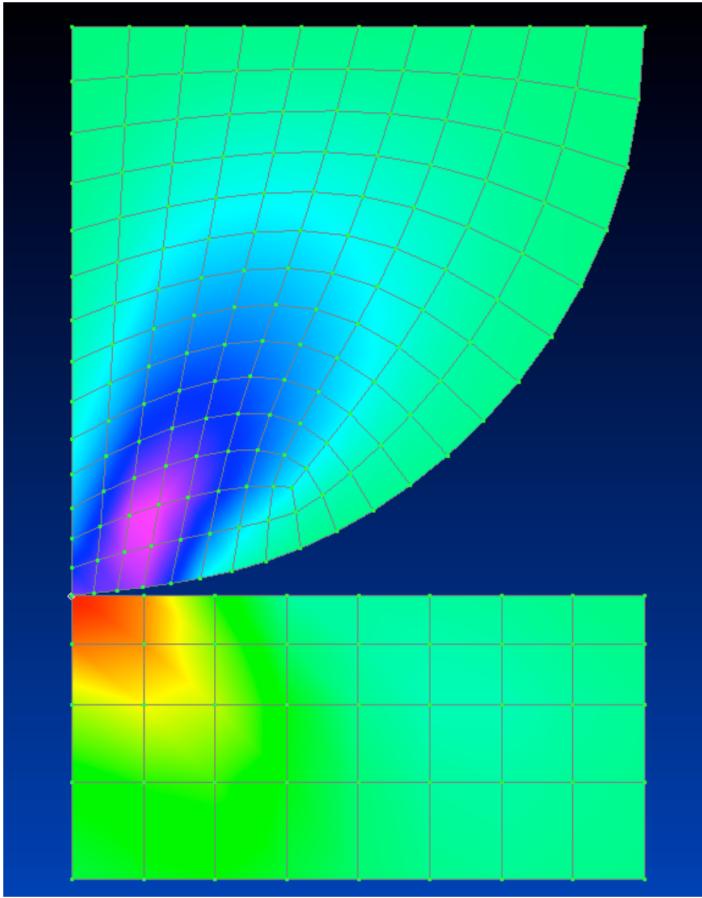


Fig. 9.1.8: Shear stress distribution (maximum value = 15.6)

1.21.5 (3) Eigenvalue analysis

The verification models of exJ and exK are the same as those of exA–G. A schematic diagram of the verification model is shown in Fig. 9.1.9. An eigenvalue analysis was performed for this model to determine the primary, secondary, and tertiary eigenvalues. The iteration and direct method solvers were used for exJ and exK, respectively. Furthermore, the verification results are presented in Tables 9.1.9–9.1.12.



Fig. 9.1.9: Verification Model

The vibration eigenvalue of the cantilever is determined by the following equations:

Primary :

$$n_1 = \frac{1.875^2}{2\pi l^2} \sqrt{\frac{gEI}{\omega}}$$

Secondary :

$$n_2 = \frac{4.694^2}{2\pi l^2} \sqrt{\frac{gEI}{\omega}}$$

Tertiary :

$$n_3 = \frac{7.855^2}{2\pi l^2} \sqrt{\frac{gEI}{\omega}}$$

The property values of the verification model are:

Item	Value
I	10.0mm
E	4000.0kgf/mm ²
l	1.0/12.0mm ⁴
ω	7.85 * 10 ⁻⁶ kgf/mm ³
g	9800.0mm/sec ²

Therefore, the primary to tertiary eigenvalue are as follows:

Mode number	Value
n_1	3.609e3
n_2	2.262e4
n_3	6.335e4

Table 9.1.9: exJ: Iteration method verification results with the primary eigenvalue

Case Name	Number of elements	Predicated value : n1=3.609e3	Remarks		
NASTRAN	FrontISTR				
J231	40	5.861e3	5.861e3	33 nodes / plane stress status problem	
J232	40	3.596e3	3.593e3	105 nodes / plane stress status problem	
J241	20	3.586e3	4.245e3	33 nodes / plane stress status problem	
J242	20	3.590e3	3.587e3	85 nodes / plane stress status problem	
J341	240	5.442e3	5.429e3	99 nodes	
J342	240	3.621e3	3.595e3	525 nodes	
J351	80	3.695e3	4.298e3	99 nodes	
J352	80	3.610e3	3.609e3	381 nodes	
J361	40	3.679e3	3.619e3	99 nodes	
J362	40	3.611e3	3.606e3	220 nodes	

Table 9.1.10: Iteration method verification results of exJ with the secondary eigenvalue

Case name	Number of elements	Predicated value : n2=2.262e4	Remarks		
NASTRAN	FrontISTR				
J231	40	3.350e4	3.351e4	33 nodes / plane stress status problem	
J232	40	2.163e4	2.156e4	105 nodes / plane stress status problem	
J241	20	2.149e4	2.516e4	33 nodes / plane stress status problem	
J242	20	2.149e4	2.143e4	85 nodes / plane stress status problem	
J341	240	3.145e4	3.138e4	99 nodes	
J342	240	2.171e4	2.155e4	525 nodes	
J351	80	2.208e4	2.546e4	99 nodes	
J352	80	2.156e4	2.149e4	381 nodes	
J361	40	2.202e4	2.168e4	99 nodes	
J362	40	2.154e4	2.144e4	220 nodes	

Note: In the three-dimensional (3D) models, the primary and secondary values have equal roots.
Therefore, the secondary value in the table represents the tertiary calculation value.

Table 9.1.11: Direct method verification results of exK with the primary eigenvalue

Case name	Number of elements	Predicated Value : n1=3.609e3		Remarks
		NASTRAN	FrontISTR	
J231	40	5.861e3	5.861e3	33 nodes / plane stress status problem
J232	40	3.596e3	3.593e3	105 nodes / plane stress status problem
J241	20	3.586e3	4.245e3	33 nodes / plane stress status problem
J242	20	3.590e3	3.587e3	85 nodes / plane stress status problem
J341	240	5.442e3	5.429e3	99 nodes
J342	240	3.621e3	3.595e3	525 nodes
J351	80	3.695e3	4.298e3	99 nodes
J352	80	3.610e3	3.609e3	381 nodes
J361	40	3.679e3	3.619e3	99 nodes
J362	40	3.611e3	3.606e3	220 nodes
J731	40	-	3.606e3	220 nodes
J741	20	-	3.594e3	220 nodes

Table 9.1.12: Direct method verification results of exK with the secondary eigenvalue

Case name	Number of elements	Predicated value : n2=2.262e4		Remarks
		NASTRAN	FrontISTR	
J231	40	3.350e4	3.351e4	33 nodes / plane stress status problem
J232	40	2.163e4	2.156e4	105 nodes / plane stress status problem
J241	20	2.149e4	2.516e4	33 nodes / plane stress status problem
J242	20	2.149e4	2.143e4	85 nodes / plane stress status problem
J341	240	3.145e4	3.138e4	99 nodes
J342	240	2.171e4	2.155e4	525 nodes
J351	80	2.208e4	2.546e4	99 nodes
J352	80	2.156e4	2.149e4	381 nodes
J361	40	2.202e4	2.168e4	99 nodes
J362	40	2.154e4	2.144e4	220 nodes
J731	40	-	2.156e4	220 nodes
J741	20	-	2.153e4	220 nodes

Note: In the 3D models, the primary and secondary values have equal roots. Therefore, the secondary value in the table represents the tertiary calculation value.

1.21.6 (4) Heat conduction analysis

The common conditions for the steady-state heat conduction analysis are shown in Fig. 9.1.10. The individual conditions for the verification cases exM-exT are shown in Fig. 9.1.11. The mesh division used here is equivalent to that of exA.

The verification results (temperature distribution table) of each case are presented in Tables 9.1.13–9.1.20.

$$\begin{aligned} \text{Length between AB} &= L = 10.0m \\ \text{Cross-sectional area} &= A = 1.0mm^2 \end{aligned}$$

Temperature dependency of thermal conductivity

Thermal conductivity $\lambda(W/mK)$	Temperature ($^{\circ}C$)
50.0	0.0
35.0	500.0
20.0	1000.0

Fig. 9.1.10: Verification conditions of steady-heat conduction analysis

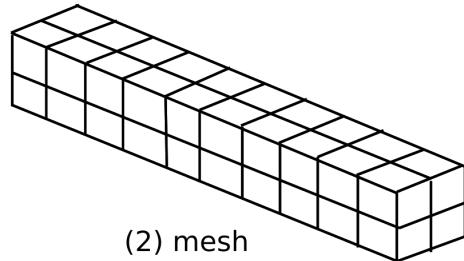
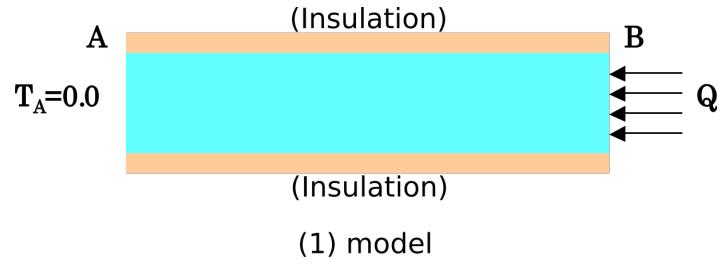
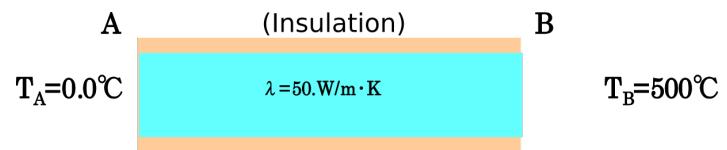
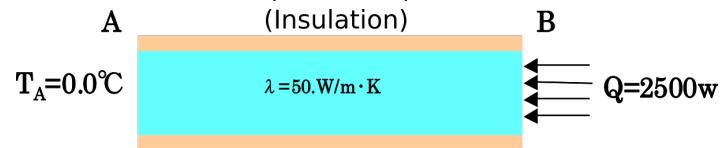


Figure 2: Heat conduction analysis

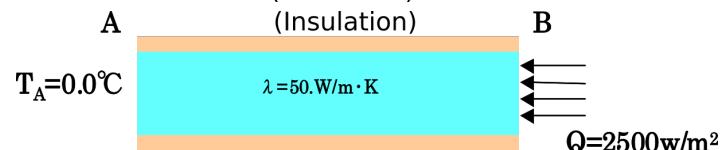
exM: Linear material



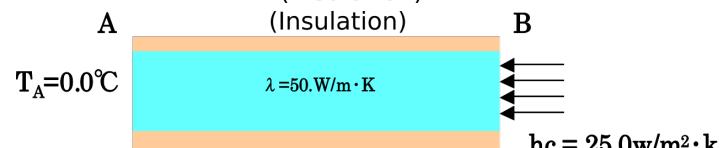
exN: Specified temperature problem



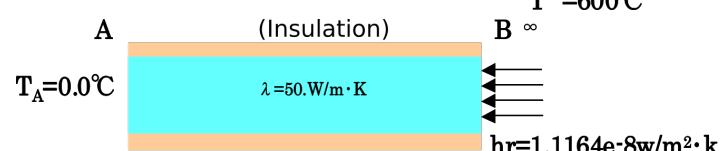
exO: Concentrated heat flux problem



exP: Distributed heat flux problem



exQ: Convective heat transfer problem



exR: Radiant heat transfer problem

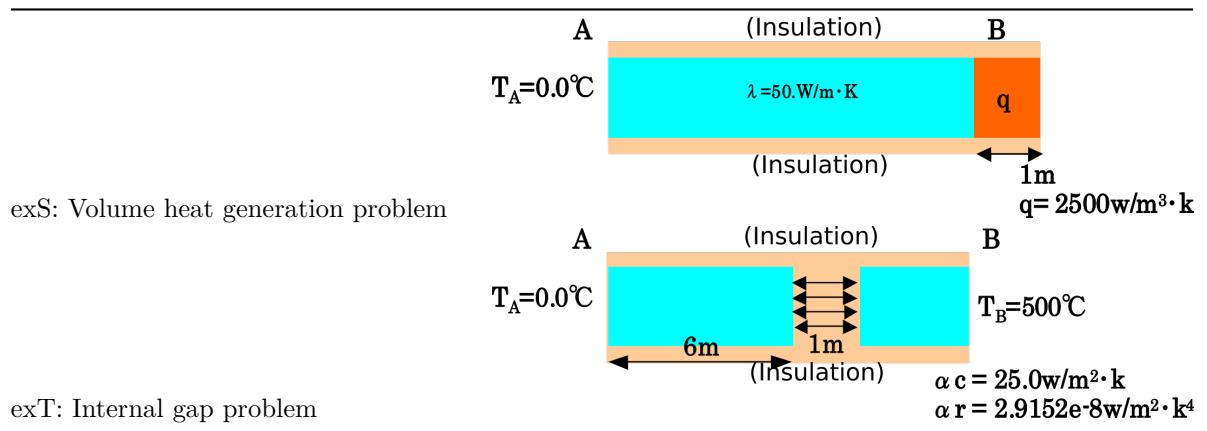


Fig. 9.1.11: Analysis conditions for each verification case

Table 9.1.13: Verification results of exM (steady-state calculation of linear material)

Case name	Element type	Number of elements/nodes	Distance from edge A (m)	Edge A	2.0	4.0	6.0	8.0	Edge B
M361A	361	40 33	0.0		100.0	200.0	300.0	400.0	500.0
M361B	361	40 105	0.0		100.0	200.0	300.0	400.0	500.0
M361C	361	20 33	0.0		100.0	200.0	300.0	400.0	500.0
M361D	361	20 85	0.0		100.0	200.0	300.0	400.0	500.0
M361E	361	240 99	0.0		100.0	200.0	300.0	400.0	500.0
M361F	361	24 525	0.0		100.0	200.0	300.0	400.0	500.0
M361G	361	80 99	0.0		100.0	200.0	300.0	400.0	500.0

Table 9.1.14: Verification results of exN (specified temperature problem)

Case name	Element type	Number of elements/nodes	Distance from edge A (m)	Edge A	2.0	4.0	6.0	8.0	Edge B
ABAQUS	361	40 99	0.0		87.3	179.7	278.2	384.3	500.0
N231	231	40 33	0.0		87.2	179.5	278.0	384.1	500.0
N232	232	40 105	0.0		86.0	178.3	276.8	382.9	500.0
N241	241	20 33	0.0		87.3	179.7	278.2	384.3	500.0
N242	242	20 85	0.0		87.3	179.7	278.2	384.3	500.0
N341	341	240 99	0.0		87.3	179.7	278.2	384.3	500.0
N342	342	24 525	0.0		87.9	179.9	278.0	383.6	500.0
N351	351	80 99	0.0		87.3	179.7	278.2	384.3	500.0
N352	352	80 381	0.0		87.3	179.7	278.2	384.3	500.0
N361	361	40 99	0.0		87.3	179.7	278.2	384.3	500.0
N362	362	40 330	0.0		87.3	179.7	278.2	384.3	500.0
N731	731	40 33	0.0		87.3	179.7	278.2	384.3	500.0
N741	741	20 33	0.0		87.3	179.7	278.2	384.3	500.0

Table 9.1.15: Verification results of exO (concentrated heat flux problem)

Case name	Element type	Number of elements/nodes	Distance from edge A (m)	Edge A	2.0	4.0	6.0	8.0	Edge B
ABAQUS	361	40 99	0.0		103.2	213.7	333.3	464.8	612.0
O231	231	40 33	0.0		103.2	213.7	333.3	464.8	612.0
O232	232	40 105	0.0		103.2	213.7	333.3	464.8	612.0
O241	241	20 33	0.0		103.2	213.7	333.3	464.8	612.0

Case name	Element type	Number of elements/nodes	Distance from edge A (m)	103.2	213.7	333.4	465.2	618.0
O242	242	20 85	0.0	-	-	-	-	-
O341	341	240 99	-	-	-	-	-	-
O342	342	24 525	0.0	104.4	214.9	334.7	466.3	614.0
O351	351	80 99	-	-	-	-	-	-
O352	352	80 381	0.0	103.2	213.7	333.3	465.0	624.0
O361	361	40 99	0.0	103.2	213.7	333.3	464.8	612.0
O362	362	40 330	0.0	103.2	213.7	333.4	465.5	623.0
O731	731	40 33	0.0	103.2	213.7	333.3	464.8	612.0
O741	741	20 33	0.0	103.2	213.7	333.3	464.8	612.0

Table 9.1.16: Verification results of exP (distribution heat flux problem)

Case name	Element type	Number of elements/nodes	Distance from edge A (m)	Edge A	2.0	4.0	6.0	8.0	Edge
ABAQUS	361	40 99	0.0	-	103.2	213.7	333.3	464.8	612.0
P231	231	40 33	0.0	-	103.2	213.7	333.3	464.8	612.0
P232	232	40 105	0.0	-	103.2	213.7	333.3	464.8	612.0
P241	241	20 33	0.0	-	103.2	213.7	333.3	464.8	612.0
P242	242	20 85	0.0	-	103.2	213.7	333.3	464.8	612.0
P341	341	240 99	-	-	-	-	-	-	-
P342	342	24 525	0.0	-	103.2	213.7	333.3	464.8	612.0
P351	351	80 99	-	-	-	-	-	-	-
P352	352	80 381	0.0	-	103.2	213.7	333.3	464.8	612.0
P361	361	40 99	0.0	-	103.2	213.7	333.3	464.8	612.0
P362	362	40 330	0.0	-	103.2	213.7	333.4	465.5	612.0
P731	731	40 33	0.0	-	103.2	213.7	333.3	464.8	612.0
P741	741	20 33	0.0	-	103.2	213.7	333.3	464.8	612.0

Table 9.1.17: Verification results of exQ (convective heat transfer problem)

Case name	Element type	Number of elements/nodes	Distance from edge A (m)	Edge A	2.0	4.0	6.0	8.0	Edge
ABAQUS	361	40 99	0.0	-	89.2	183.8	284.8	393.9	513.2
Q231	231	40 33	0.0	-	89.2	183.8	284.8	393.9	513.2
Q232	232	40 105	0.0	-	89.2	183.8	284.8	393.9	513.2
Q241	241	20 33	0.0	-	89.2	183.8	284.8	393.9	513.2
Q242	242	20 85	0.0	-	89.2	183.8	284.8	393.9	513.2
Q341	341	240 99	-	-	-	-	-	-	-
Q342	342	24 525	0.0	-	89.2	183.8	284.8	393.9	513.2
Q351	351	80 99	-	-	-	-	-	-	-
Q352	352	80 381	0.0	-	89.2	183.8	284.8	393.9	513.2
Q361	361	40 99	0.0	-	89.2	183.8	284.8	393.9	513.2
Q362	362	40 330	0.0	-	89.2	183.8	284.8	393.9	513.2
Q731	731	40 33	0.0	-	89.2	183.8	284.8	393.9	513.2
Q741	741	20 33	0.0	-	89.2	183.8	284.8	393.9	513.2

Table 9.1.18: Verification results of exR (radiation heat transfer problem)

Case name	Element type	Number of elements/nodes	Distance from edge A (m)	Edge A	2.0	4.0	6.0	8.0	Edge
ABAQUS	361	40 99	0.0	-	89.5	184.4	285.8	395.3	515.2
R231	231	40 33	0.0	-	89.5	184.4	285.8	395.3	515.2

Case name	Element type	Number of elements/nodes	Distance from edge A (m)						
R232	232	40 105	0.0		89.5	184.4	285.8	395.3	515.2
R241	241	20 33	0.0		89.5	184.4	285.8	395.3	515.2
R242	242	20 85	0.0		89.5	184.4	285.8	395.3	515.2
R341	341	240 99	-		-	-	-	-	-
R342	342	24 525	0.0		89.5	184.4	285.8	395.3	515.2
R351	351	80 99	-		-	-	-	-	-
R352	352	80 381	0.0		89.5	184.4	285.8	395.3	515.2
R361	361	40 99	0.0		89.5	184.4	285.8	395.3	515.2
R362	362	40 330	0.0		89.5	184.4	285.8	395.3	515.2
R731	731	40 33	0.0		89.5	184.4	285.8	395.3	515.2
R741	741	20 33	0.0		89.5	184.4	285.8	395.3	515.2

Table 9.1.19: Verification results of exS (volume heat generation problem)

Case name	Element type	Number of elements/nodes	Distance from edge A (m)						
			Edge A	2.0	4.0	6.0	8,0	Edge	
ABAQUS	361	40 99	0.0	103.2	213.7	333.3	464.8	612.0	
S231	231	40 33	0.0	103.2	213.7	333.3	464.8	612.0	
S232	232	40 105	0.0	103.2	213.7	333.3	464.8	612.0	
S241	241	20 33	0.0	103.2	213.7	333.3	464.8	612.0	
S242	242	20 85	0.0	103.2	213.7	333.3	464.8	612.0	
S341	341	240 99	-	-	-	-	-	-	-
S342	342	24 525	0.0	103.2	213.7	333.3	464.8	612.0	
S351	351	80 99	-	-	-	-	-	-	-
S352	352	80 381	0.0	103.2	213.7	333.3	464.8	612.0	
S361	361	40 99	0.0	103.2	213.7	333.3	464.8	612.0	
S362	362	40 330	0.0	103.2	213.7	333.3	464.8	612.0	
S731	731	40 33	0.0	103.2	213.7	333.3	464.8	612.0	
S741	741	20 33	0.0	103.2	213.7	333.3	464.8	612.0	

Table 9.1.20: Verification results of exT (internal gap problem)

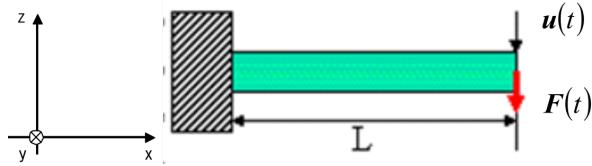
Case name	Element type	Number of elements/nodes	Distance from edge A (m)						
			Edge A	2.0	4.0	6.0	8,0	Edge	
ABAQUS	361	40 99	0.0	88.6	182.4	282.6	387.7	500.0	
S231	231	40 33	0.0	88.6	182.4	282.6	387.7	500.0	
S232	232	40 105	0.0	88.6	182.4	282.6	387.7	500.0	
S241	241	20 33	0.0	88.6	182.4	282.6	387.7	500.0	
S242	242	20 85	0.0	88.6	182.4	282.6	387.7	500.0	
S341	341	240 99	-	-	-	-	-	-	-
S342	342	24 525	0.0	88.6	182.4	282.6	387.7	500.0	
S351	351	80 99	-	-	-	-	-	-	-
S352	352	80 381	0.0	88.6	182.4	282.6	387.7	500.0	
S361	361	40 99	0.0	88.6	182.4	282.6	387.7	500.0	
S362	362	40 330	0.0	88.6	182.4	282.6	387.7	500.0	
S731	731	40 33	0.0	88.6	182.4	282.6	387.7	500.0	
S741	741	20 33	0.0	88.6	182.4	282.6	387.7	500.0	

1.21.7 (5) Linear dynamic analysis

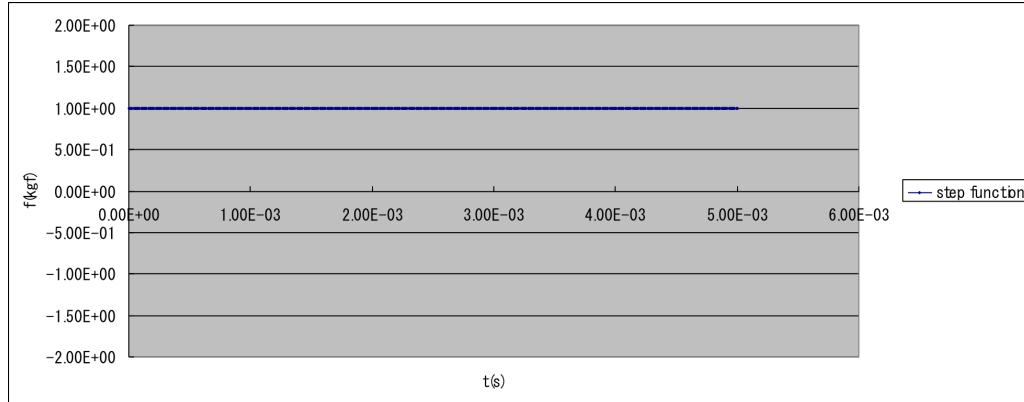
In exW, a linear dynamic analysis was performed on the same mesh-divided cantilever that was discussed earlier in the Elastic static analysis subsection, with the verification conditions shown in Fig. 9.1.12. The objective was to verify the impact of time increment on the results with the same mesh division. Both the implicit and explicit

methods of dynamic analysis and the element types 361 and 342 were used. The verification results are presented in Table 9.1.22 and shown in Figs. 9.1.13–9.1.15.

Analysis Model



Analysis Model



Time history of external force F

The theoretical solution for vibration point displacement is as follows:

$$F(t) = F_0 I(t)$$

where

$$F_0 : \text{Constant vector}$$

$$I(t) = \begin{cases} 0, & t < 0 \\ 1, & 0 \leq t \end{cases}$$

$$u(t) = \frac{F_0 l^3}{EI} \sum_{i=1}^{\infty} i = 1 \frac{1 - \cos \omega_i t}{\lambda_i^4} \left\{ \cosh \lambda_i - \cos \lambda_i - \frac{\cosh \lambda_i + \cos \lambda_i}{\sin \lambda_i + \sinh \lambda_i} (\sinh \lambda_i - \sin \lambda_i) \right\}^2$$

Fig. 9.1.12: Verification conditions of linear dynamic analysis

Verification conditions:

Length	L	10.0 mm
Cross-sectional width	a	1.0 mm
Cross-sectional height	b	1.0 mm
Young's modulus	E	4000.0 kgf/mm ²
Poisson's ratio	ν	0.3
Density	ρ	1.0E-09 kgf s ² /mm ³
Gravitational acceleration	g	9800.0 mm/s ²
External force	F_0	1.0 kgf

Element	Hexahedral linear element
Tetrahedral secondary element	

Solution	Implicit method
Parameter γ of Newmark- β method	1/2
Parameter β of Newmark- β method	1/4
Explicit method	
Damping	N/A

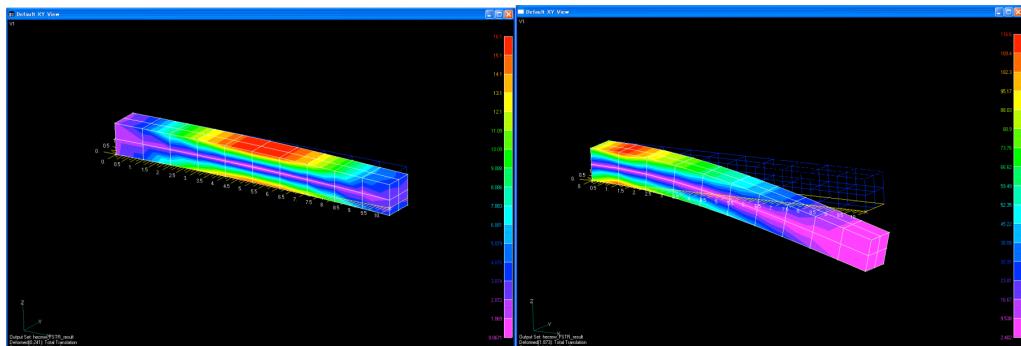
Table 9.1.21: Verification conditions of the linear dynamic analysis (continuation)

Case Name	Element Type	No. of Nodes	No. of Elements	Solution	Time Increment Δt [sec]
W361_c0_in36in2_t1		99	40	Implicit method	1.0E-06
W361_c0_in36in2_t2		99	40	Implicit method	1.0E-05
W361_c0_in36in2_t3		99	40	Implicit method	1.0E-04
W361_c0_ex36in2_t1		99	40	Explicit method	1.0E-08
W361_c0_ex36in2_t2		99	40	Explicit method	1.0E-07
W361_c0_ex36in2_t3		99	40	Explicit method	1.0E-06
W342_c0_in34in2_t1		525	240	Implicit method	1.0E-06
W342_c0_in34in2_t2		525	240	Implicit method	1.0E-05
W342_c0_in34in2_t3		525	240	Implicit method	1.0E-04
W342_c0_ex34in2_t1		525	240	Explicit method	1.0E-08
W342_c0_ex34in2_t2		525	240	Explicit method	5.0E-08
W342_c0_ex34in2_t3		525	240	Explicit method	1.0E-07

Table 9.1.22: Verification results of linear dynamic analysis of exW (cantilever)

Case name	Element type	Number of nodes	Number of elements	Solu-tion	z-direction displacement: u_z (mm) when time $t = 0.002(s)$	Fron-tISTR
W361_c036in_m2_t199		40		Theoretical solution repeated to sextic equation		
W361_c036in_m2_t299		40		Im-plicit method	1.9753	1.9302
W361_c036in_m2_t399		40		Im-plicit method	1.9753	1.8686
W361_c036k_m2_t199		40		Im-plicit method	1.9753	0.3794
W361_c036k_m2_t299		40		Ex-plicit method	1.9753	1.9302
W361_c036k_m2_t399		40		Ex-plicit method	1.9753	1.9247

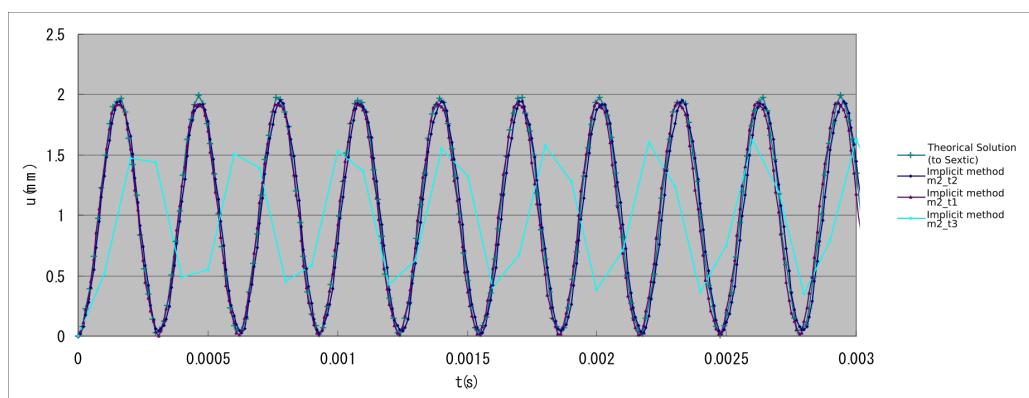
Case name	Element type	Number of nodes	Number of elements	Solu-tion	z-direction displacement: $u_z(mm)$ when time $t = 0.002(s)$	
W361_c036k_m2_t399			40	Ex-plicit method	1.9753	Diver-gence
W342_c034h_m2_t525			240	Im-plicit method	1.9753	1.9431
W342_c034h_m2_t325			240	Im-plicit method	1.9753	1.8719
W342_c034h_m2_t3525			240	Im-plicit method	1.9753	0.3873
W342_c034k_m2_t1525			240	Ex-plicit method	1.9753	1.9359
W342_c034k_m2_t2525			240	Ex-plicit method	1.9753	1.9358
W342_c034k_m2_t3525			240	Ex-plicit method	1.9753	Diver-gence



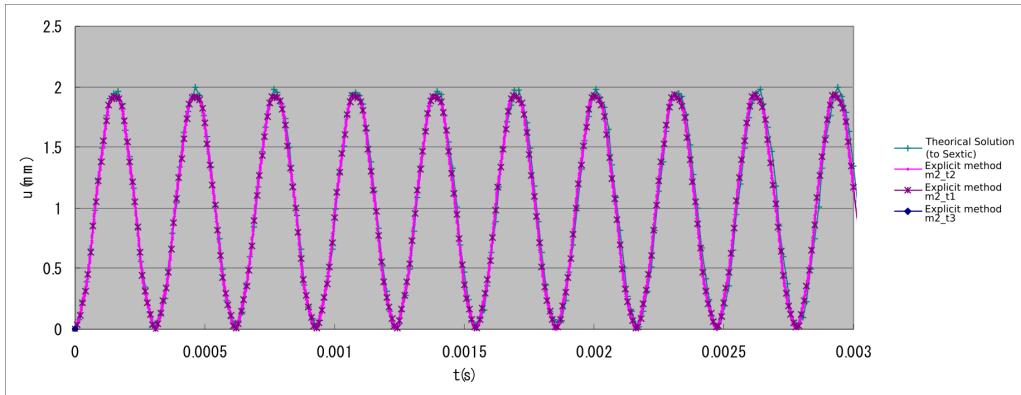
(a) $t=2.0\text{E-}03(\text{s})$

(b) $t=4.0\text{E-}03(\text{s})$

Fig. 9.1.13: Deformation diagram and equivalent stress distribution of the cantilever (W361_c0_im_m2_t2)

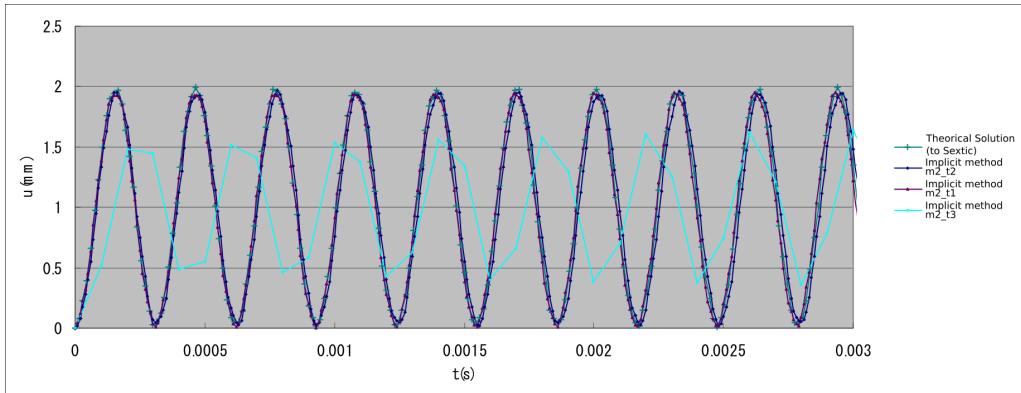


(a) Element Type 361 : Implicit method

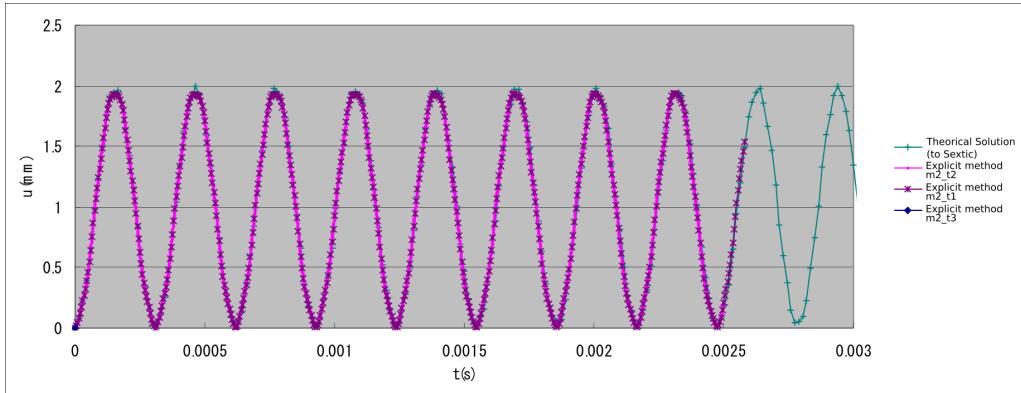


(b) Element Type 361 : Explicit method

Fig. 9.1.14: Time history of vibration point displacement u_z



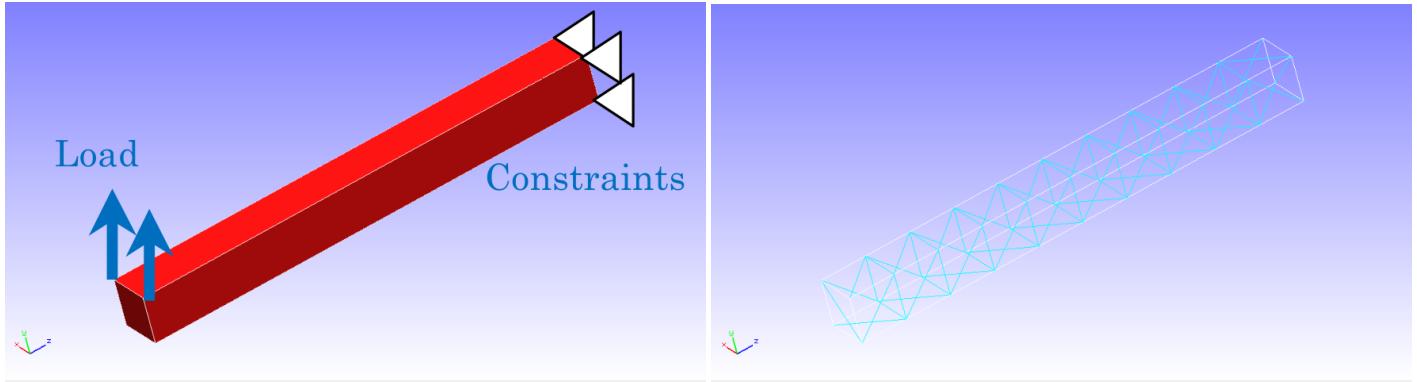
(a) Element Type 342 : Implicit method



(b) Element Type 342 : Explicit method

1.21.8 (6) Frequency response analysis

In this verification, a frequency response analysis was performed on a cantilever, and the results were compared with those obtained using the ABAQUS software. The analysis model and verification conditions are presented below:



Analysis conditions:

Young's modulus	E	210000 N/mm^2
Poisson's ratio	ν	0.3
Density	ρ	$7.89E - 09 \text{ t/mm}^3$
Gravitational acceleration	g	9800.0 mm/s^2
Load	F_0	1.0 N
Parameter of Rayleigh damping	R_m	0.0
Parameter of Rayleigh damping	R_k	$7.2E - 07$

Fig. 9.1.15 : Analysis model (tetrahedral primary element (126 elements and 55 nodes))

The eigenvalues up to the fifth order and the frequency response of the vibration points obtained from eigenvalue analysis are as follows:

mode	FrontISTR	ABAQUS
1	14952	14952
2	15002	15003
3	84604	84539
4	84771	84697
5	127054	126852

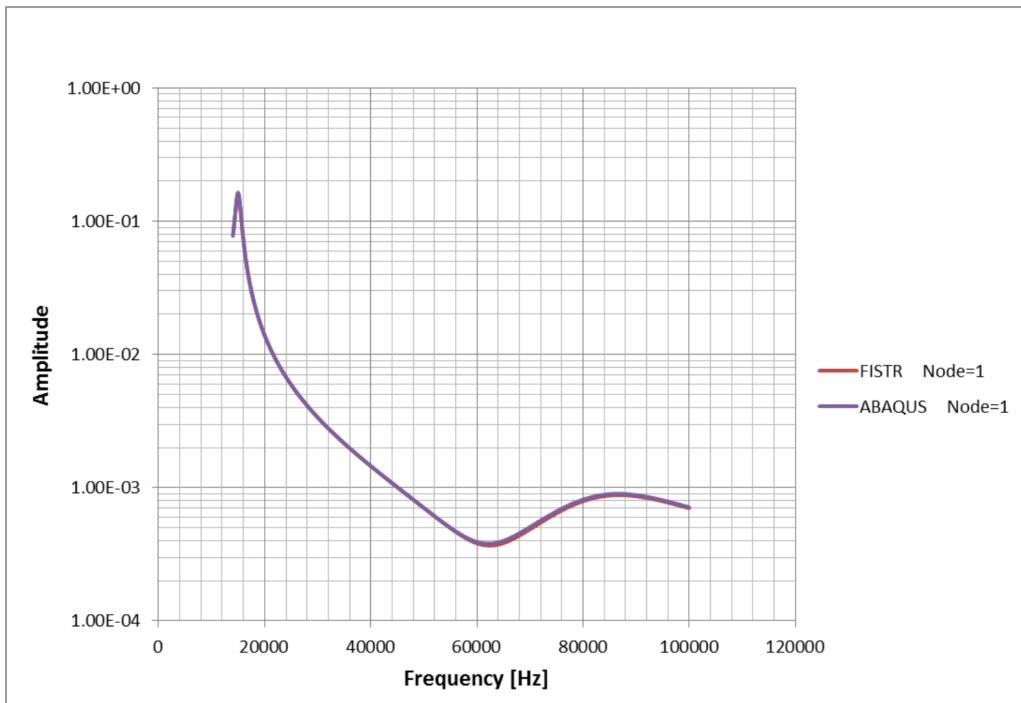


Fig. 9.1.16 : Frequency dependence of displacement strength of vibration points

1.22 Actual Model Examples for Elastic Static Analysis

1.22.1 Analysis Model

A list of actual model verification examples for elastic static analysis are presented in Table 9.2.1. The different shapes of the models are shown in Figs. 9.2.1–9.2.5 (some models are excluded). The examples of the element types 731 and 741 require a separate direct method solver.

Table 9.2.1: Actual model verification examples for elastic static analysis

Case name	Element type	Verification model	Number of nodes	Freedom frequency
EX01A	342	Connecting rod (100,000 nodes)	94,074	282,222
EX01B	342	Connecting rod (330,000 nodes)	331,142	993,426
EX02	361	Block with hole	37,386	112,158
EX03	342	Turbine blade	10,095	30,285
EX04	741	Cylindrical shell	10,100	60,600
EX05A	731	Wine glass (coarse)	7,240	43,440
EX05B	731	Wine glass (midium)	48,803	292,818
EX05C	731	Wine glass (fine)	100,602	603,612

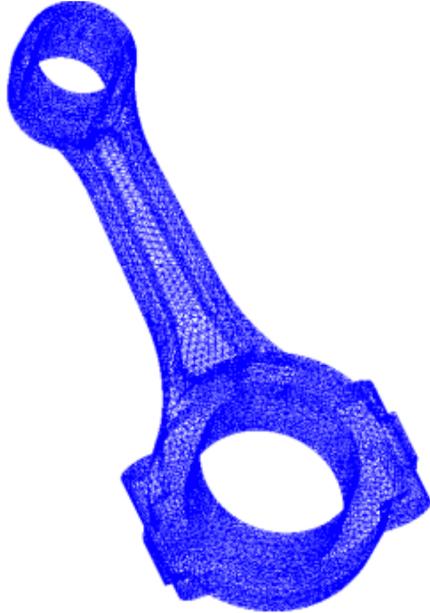


Fig. 9.2.1: Connecting Rod (EX01A)

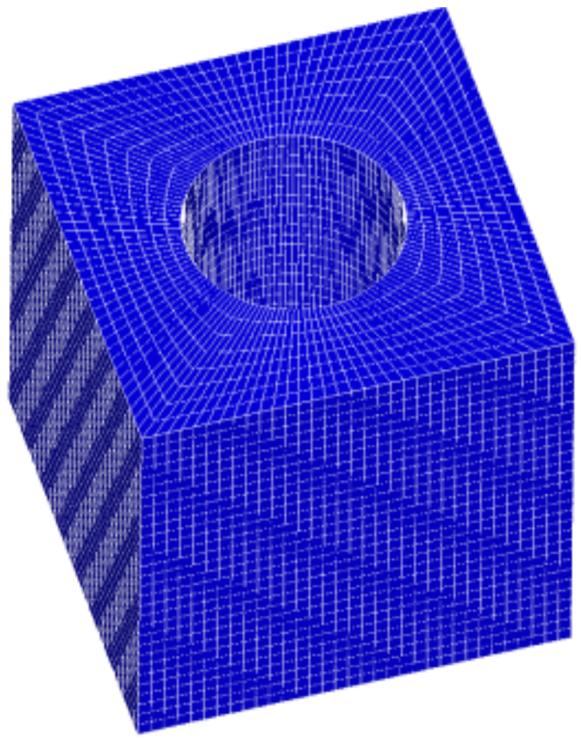


Fig. 9.2.2: Perforated block (EX02)

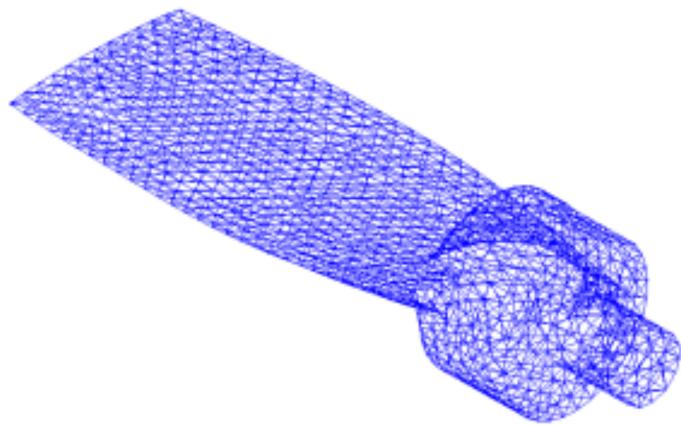


Fig. 9.2.3: Turbine blade (EX03, EX06)

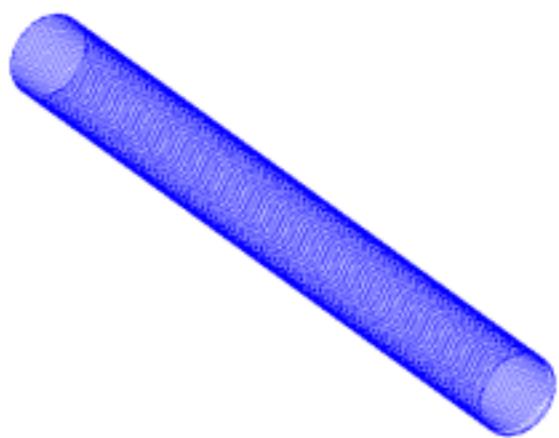


Fig. 9.2.4: Cylindrical shell (EX04, EX09)

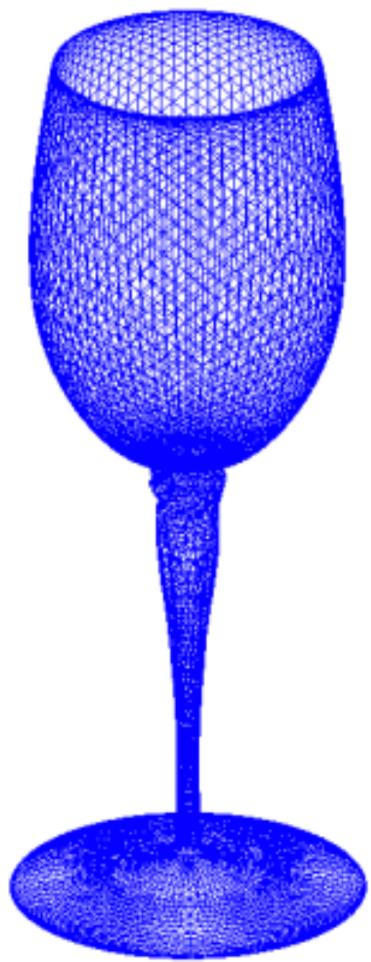


Fig. 9.2.5: Wine Glass (EX05, EX10A)

1.22.2 Analysis results

1.22.2.1 Example of analysis results

Examples of the analysis results are shown in Figs. 9.2.6–9.2.9.

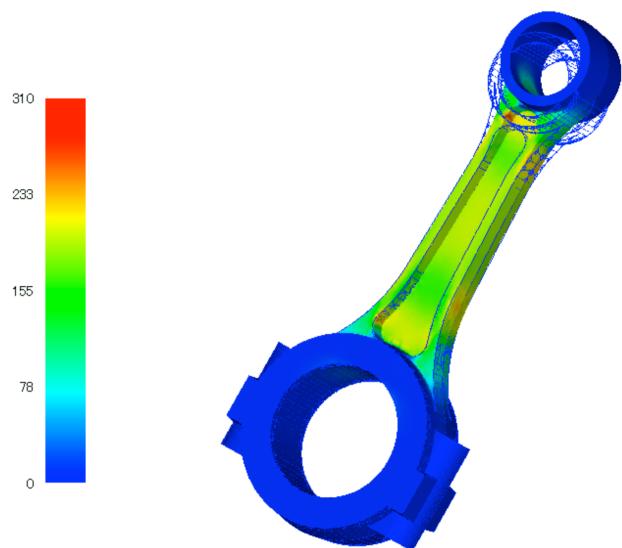


Fig. 9.2.6: EX01 analysis results (Mises stress and deformation diagram (10 times))

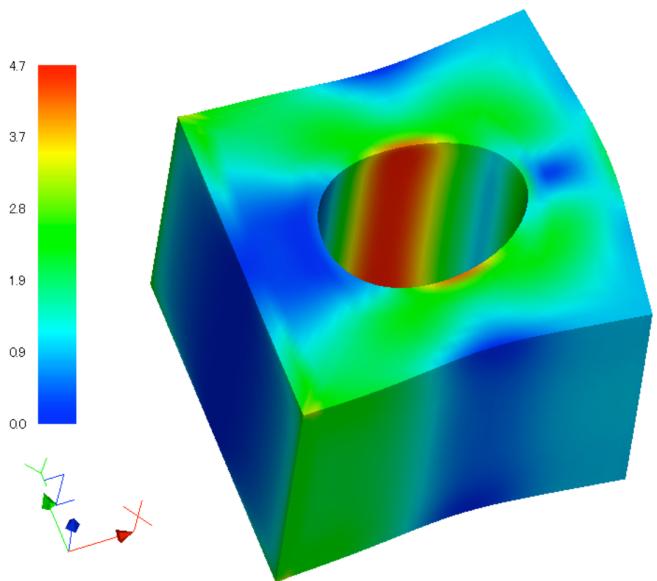


Fig. 9.2.7: EX02 analysis results (Mises stress and deformation diagram (100 times))

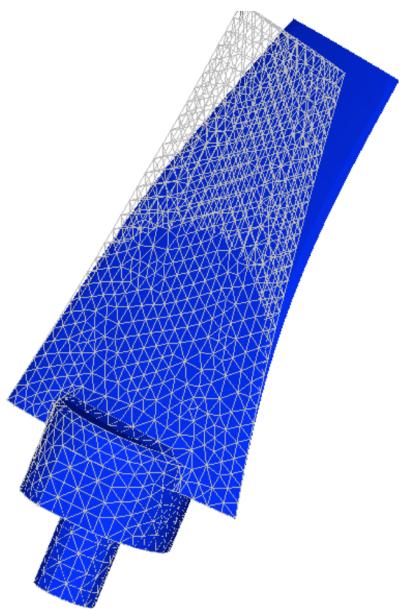
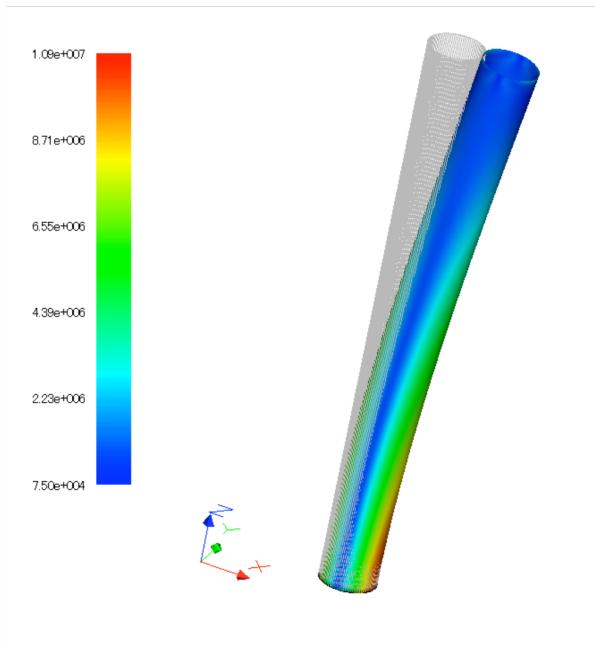


Fig. 9.2.8: EX03 analysis results (deformation diagram (10 times))



9.2.9: EX04 Analysis Results (Deformed Figure (100 times))

Fig. 9.2.9: EX04 analysis results (deformation diagram (100 times))

1.22.2.2 Verification Results of Analysis Performance with Example EX02

An analysis was performed with the commercial software ABAQUS using a model equivalent to the verification example model EX02 (perforated block). A comparison of the maximum and minimum values of the stress components with the results of FrontISTR is shown in Fig. 9.2.10. It can be seen that the stress components are very close to each other.

The effect of area division on stress distribution was also analyzed. The division was performed according to the RCB method, i.e., the model was halved in each of the X, Y, and Z axial directions, creating eight areas in total. Fig. 9.2.11 shows the division, while Fig. 9.2.12 shows the stress distribution of the analysis results with a single area and with the area divided into eight areas.

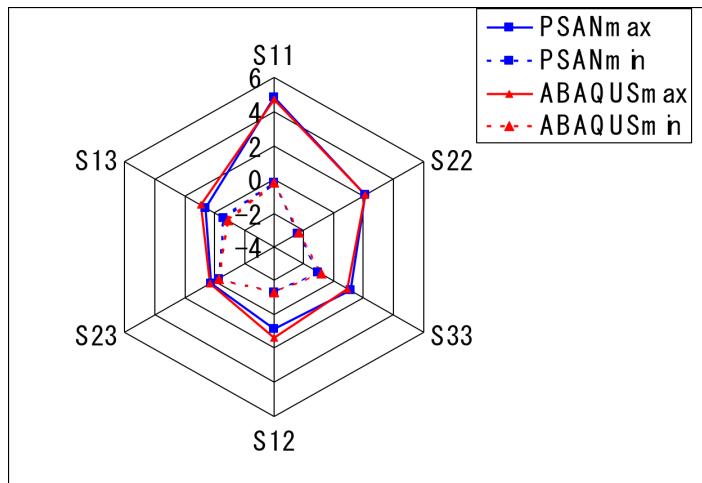


Fig. 9.2.10: Comparison of the stress components of EX02 with the commercial software

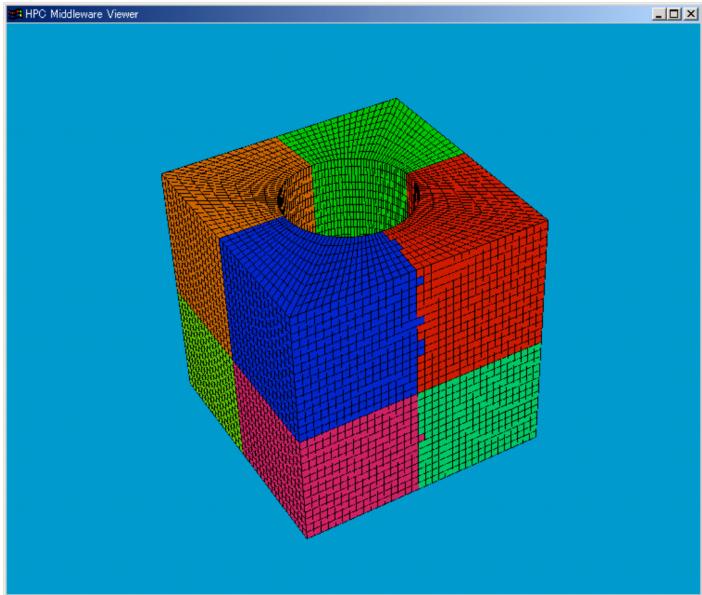


Fig. 9.2.11: Result of the division of EX02 in eight areas by the RCB method

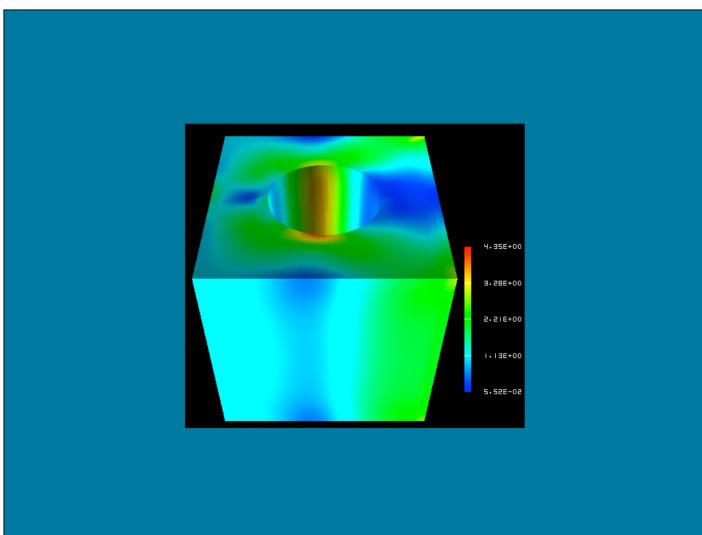


Fig. 9.2.12: No difference between the stress distribution of the analysis results with a single area and with the area divided into eight areas

Furthermore, a comparison of the execution time with the settings of the HEC-MW solver used is presented in Table 9.2.2. Fig. 9.2.13 shows the convergence history until the solution was found.

Table 9.2.2: Comparison of execution time with HEC-MW solvers

Solver	Execution Time(s)
CGI	38.79
CGscale	52.75
BCGS	60.79
CG8	6.65

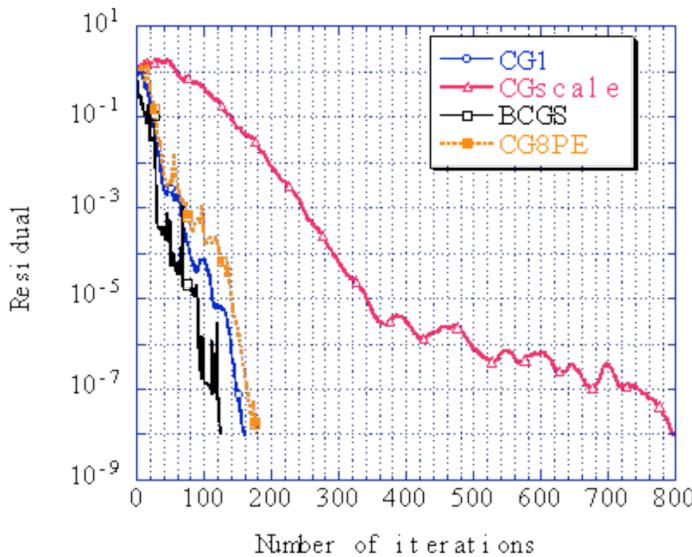


Fig. 9.2.13: Comparison of convergence history with the HEC-MW solver (convergence threshold: 1.0×10^{-8})

1.22.2.3 Comparison of calculation time with verification example EX01A

The increase rate of the calculation speed because of area division was verified with the example EX01A (connecting rod.) The test was conducted with a Xeon 2.8 GHz 24 node cluster computer, and the results are shown in Fig. 9.2.14. This figure shows that the calculation speed increases proportionally to the number of areas.

The difference in the calculation time because of the computer environment was also analyzed. The results are presented in Table 9.2.3.

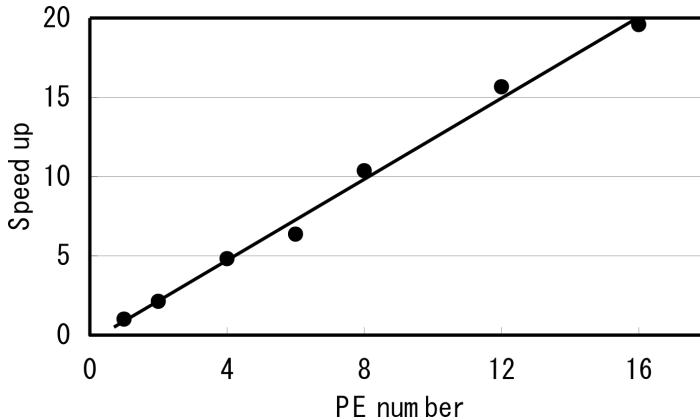


Fig. 9.2.14: Speed-increasing effect because of area division

Table 9.2.3: Comparison of calculation time with different computers (one CPU)

CPU	Frequency [GHz]	OS	CPU Time [sec]	solver time [sec]
Xeon	2.8	Linux	850	817
Pentium III	0.866	Win2000	2008	1980
Pentium M	0.760	WinXP	1096	1070
Pentium 4	2.0	WinXP	802	785
Pentium 4	2.8	WinXP	738	718
Celeron	0.700	Win2000	2252	2215
Pentium 4	2.4	WinXP	830	804

1.23 Actual Model Examples for Eigenvalue Analysis

1.23.1 Analysis model

A list of examples of the actual verification models for eigenvalue analysis is presented in Table 9.3.1. The model shapes of EX07 (turbine rotor) and EX08 (spring) are also shown in Figs. 9.3.1 and 9.3.2. The other model shapes are the same as those of the examples previously discussed in the elastic static analysis, which has the same verification content. The examples of the element types 731 and 741 require a separate direct method solver.

Table 9.3.1: Examples of actual model verification for eigenvalue analysis

Case Name	Element Type	Verification Model	No. of Nodes	No. of Degrees of Freedom
EX06	342	Turbine blade	10,095	30,285
EX07	361	Turbine rotor	127,440	382,320
EX08	342	Spring	78,771	236,313
EX09	741	Cylindrical shell	10,100	60,600
EX10A	731	Wine glass (coarse)	7,240	43,440
EX10B	731	Wine glass (medium)	48,803	292,818

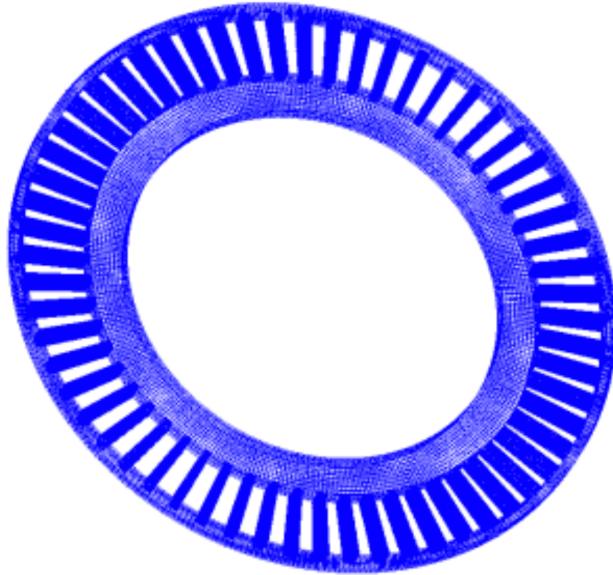


Fig. 9.3.1: Turbine rotor (EX07)

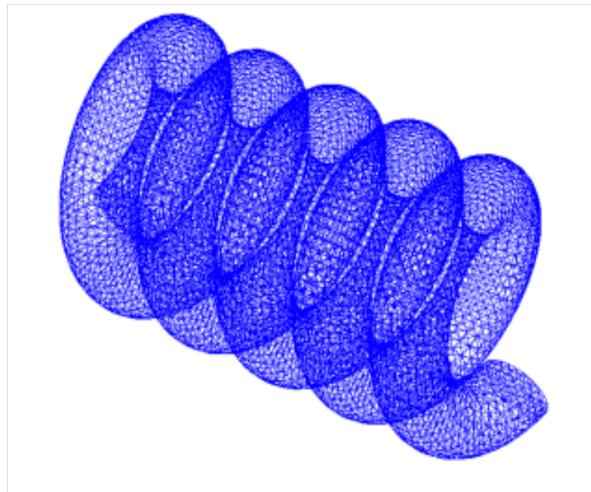


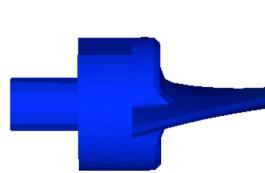
Figure 9.3.2: Spring (EX08)

Fig. 9.3.2: Spring (EX08)

1.23.2 Analysis Results

The vibration mode and natural frequency are shown in the following.

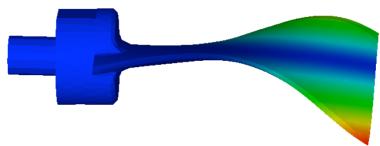
1.23.2.1 (1) EX06 Turbine blade



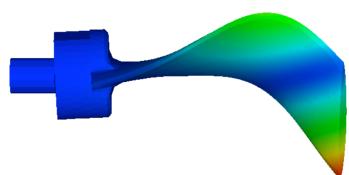
(a) Mode 1 (1170 kHz)



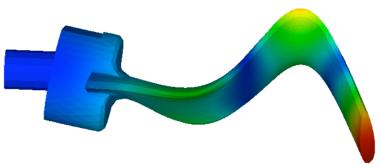
(b) Mode 2(3250kHz)



(c) Mode 3(4130kHz)



(d) Mode 4(4140kHz)



(e) Mode 5(8210kHz)

Fig. 9.3.3: EX06, turbine blade vibration mode

1.23.2.2 (2) EX07 Turbine rotor

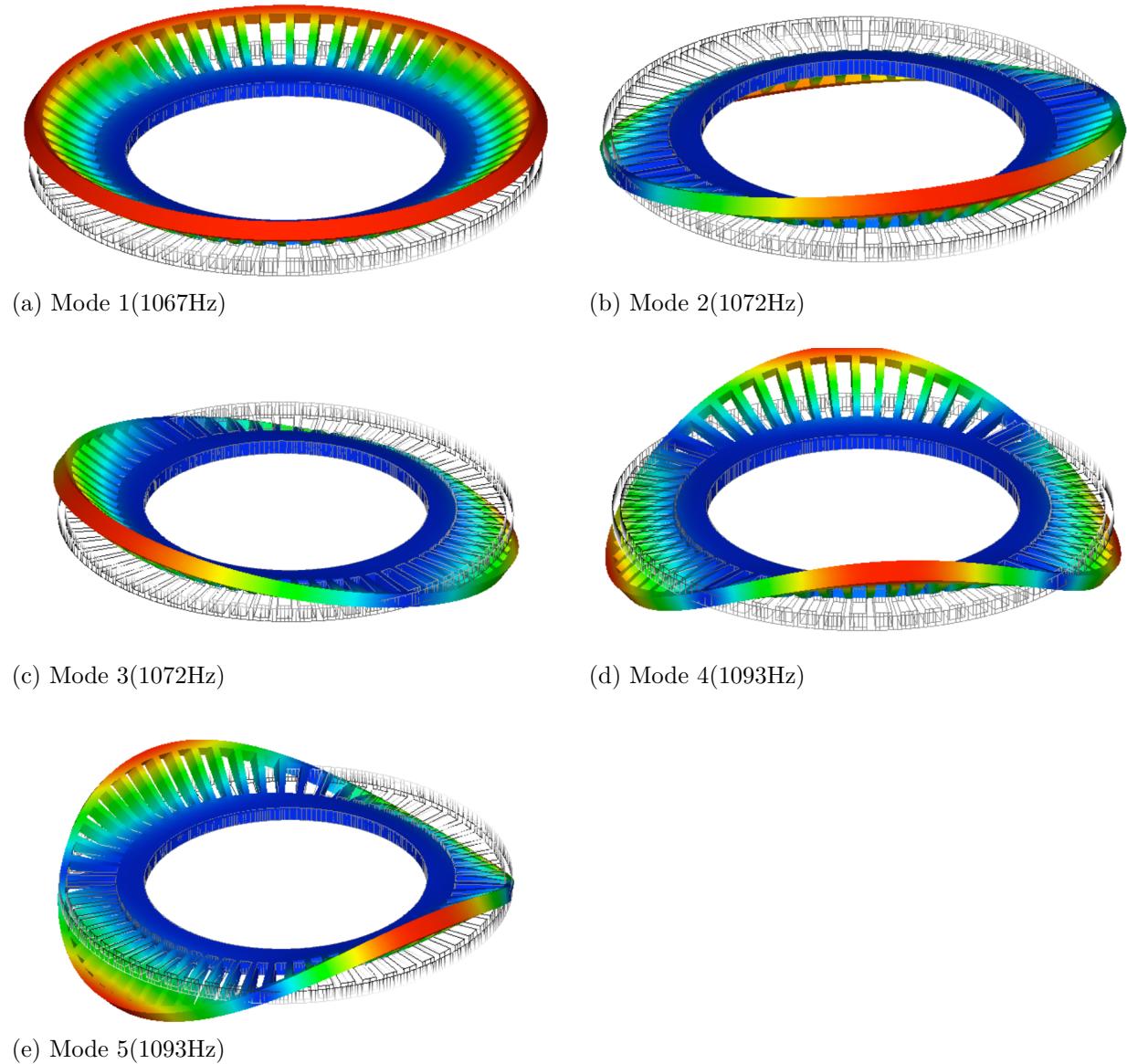
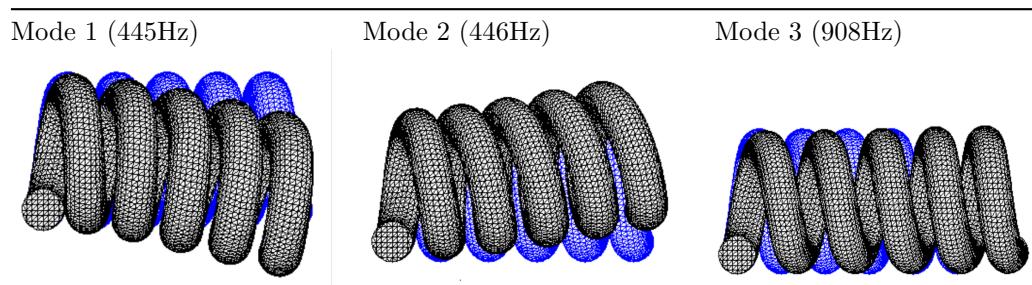


Fig. 9.3.4: EX07, Turbine rotor vibration mode

1.23.2.3 EX08 Spring



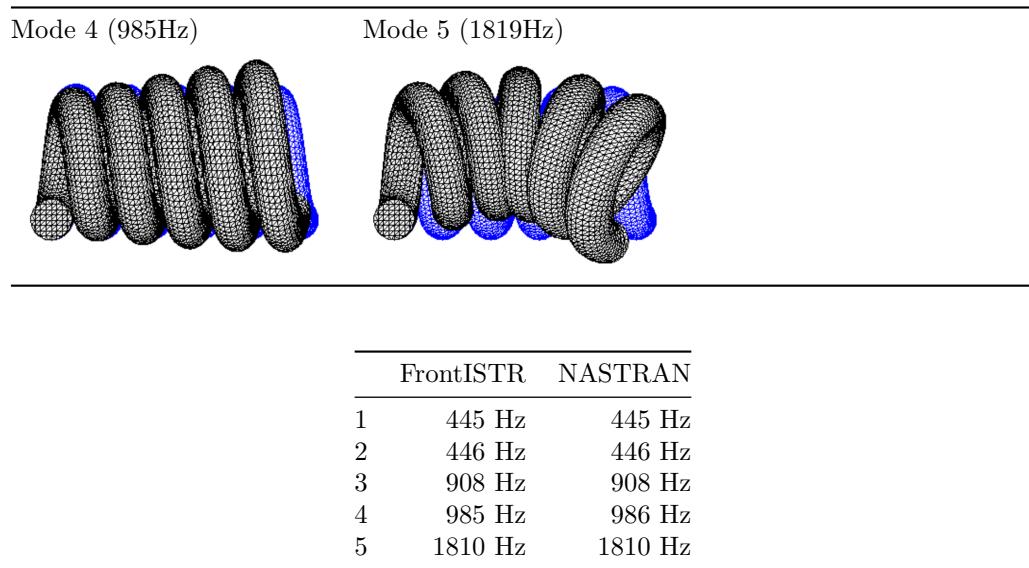
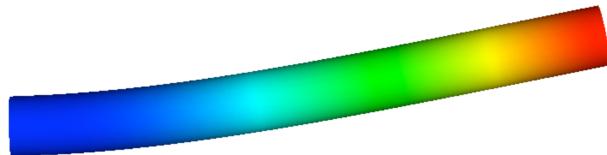
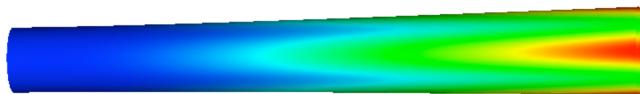


Fig. 9.3.5: EX08, spring vibration mode

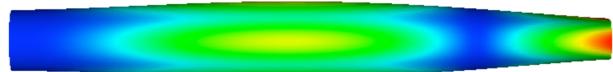
1.23.2.4 (4) EX09 Cylindrical shell



(a) Mode 1, 2 (109 Hz)



(b) Mode 3,4(570Hz)



(c) Mode 5(615Hz)

Fig. 9.3.6: EX09, cylindrical shell vibration mode

1.23.2.5 EX10A Wine glass

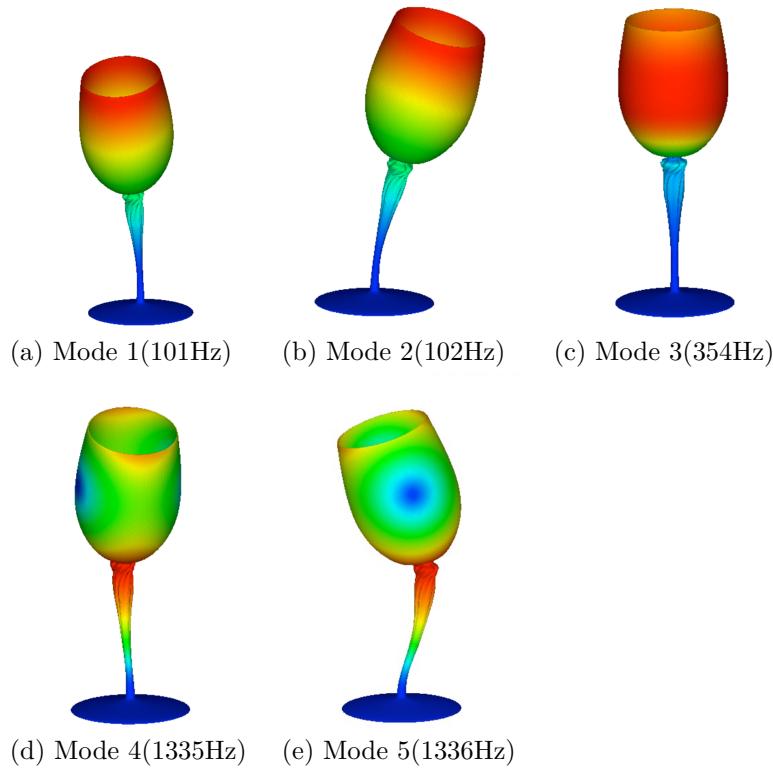


Fig. 9.3.7: EX10A Wine Glass Vibration Mode

1.24 Actual Model Examples for Heat Conduction Analysis

1.24.1 Analysis model

The heat conduction analysis was performed with a used nuclear fuel transport container as an actual model. For this analysis, three verification examples were set, each with a model of different mesh roughness, as presented in Table 9.4.1. The shapes of the models are shown in Figs. 9.4.1–9.4.4.

Table 9.4.1: Examples of actual verification models for heat conduction analysis

Case name	Element type	Verification model	Number of nodes	Freedom frequency
EX21A	361	Used unclear fuel transport container	88,938	79,920
EX21B	361		309,941	289,800
EX21C	361		1,205,765	1,159,200

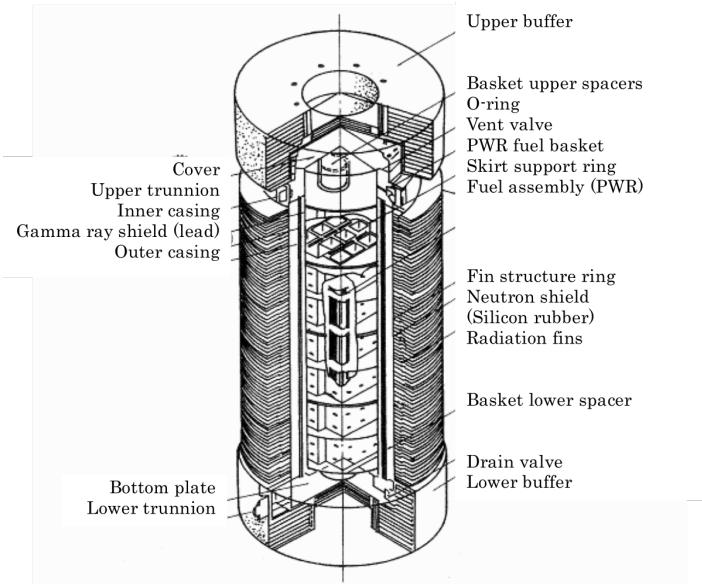


Fig. 9.4.1: Used nuclear fuel transport container

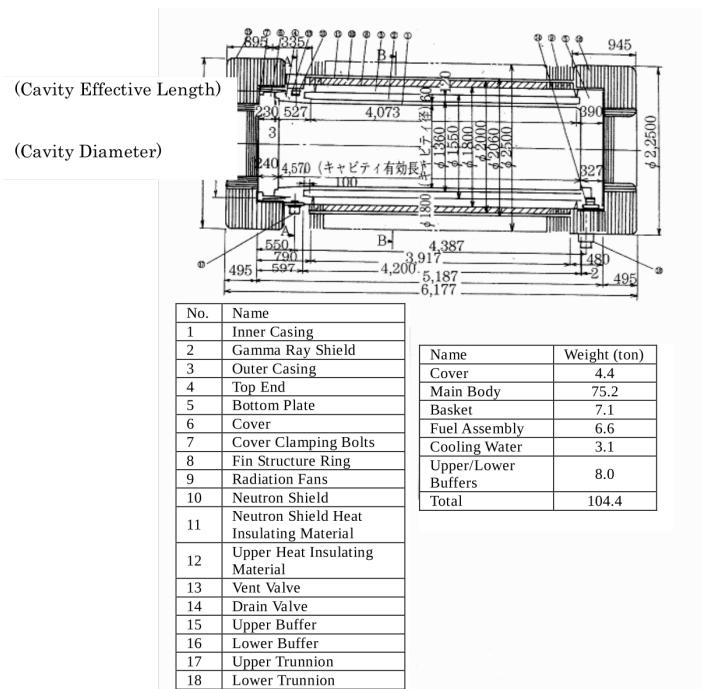


Fig. 9.4.2: Dimensions of the used nuclear fuel transport container

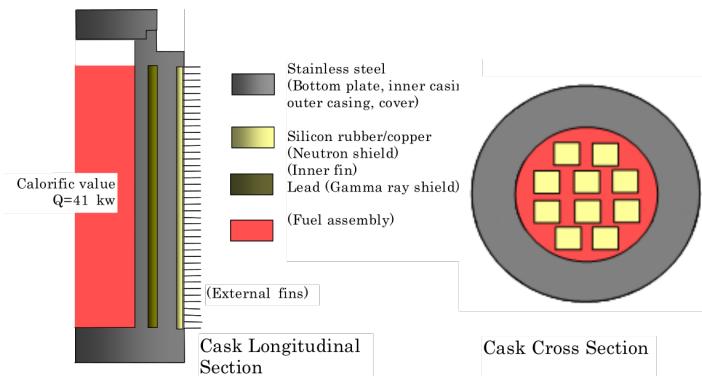


Fig. 9.4.3: Model's schematic diagram

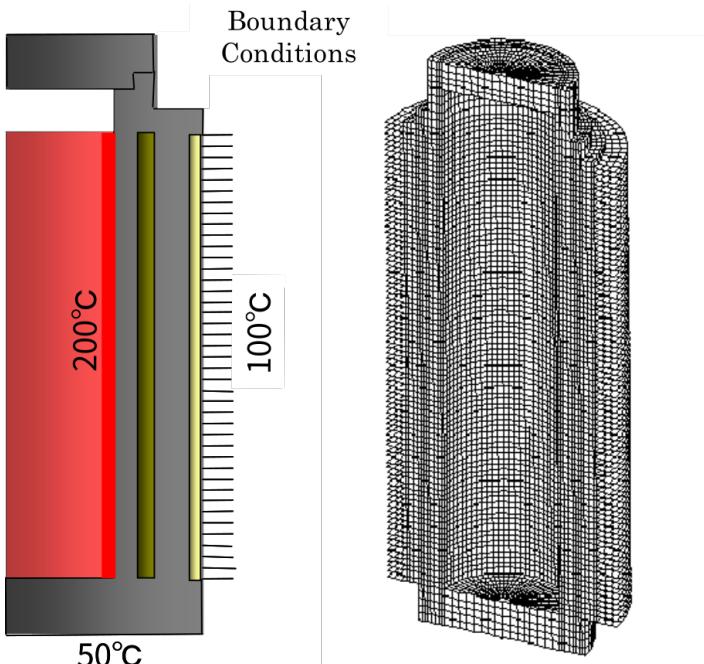


Fig. 9.4.4: Model's boundary conditions and mesh division diagram (EX21A)

1.24.2 Analysis Results

Examples of analysis results are shown in Figs. 9.4.5–9.4.7.

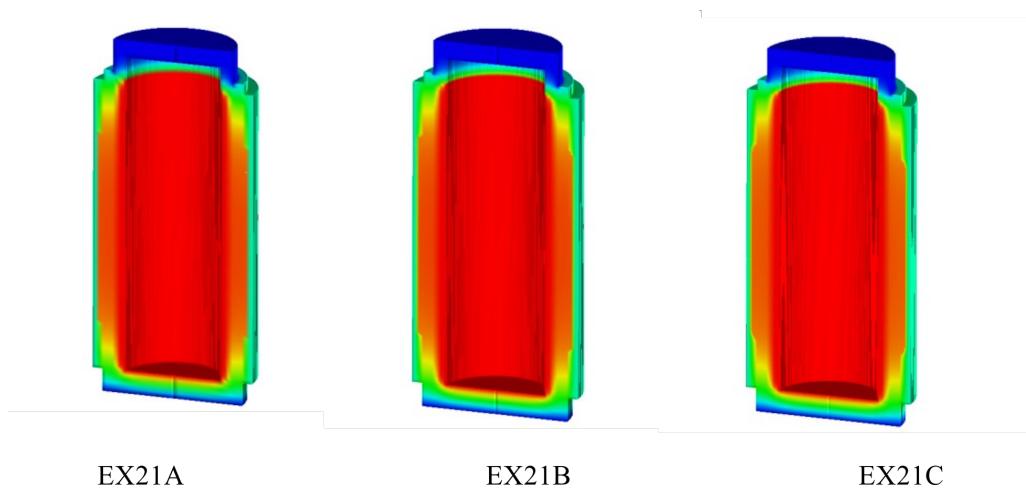


Fig. 9.4.5: Temperature distribution diagram

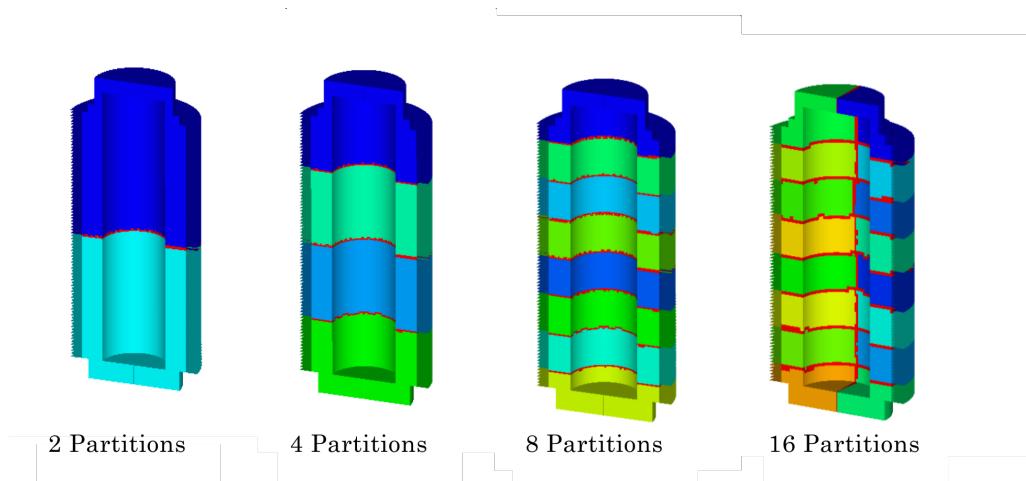


Fig. 9.4.6: Distributed model diagram

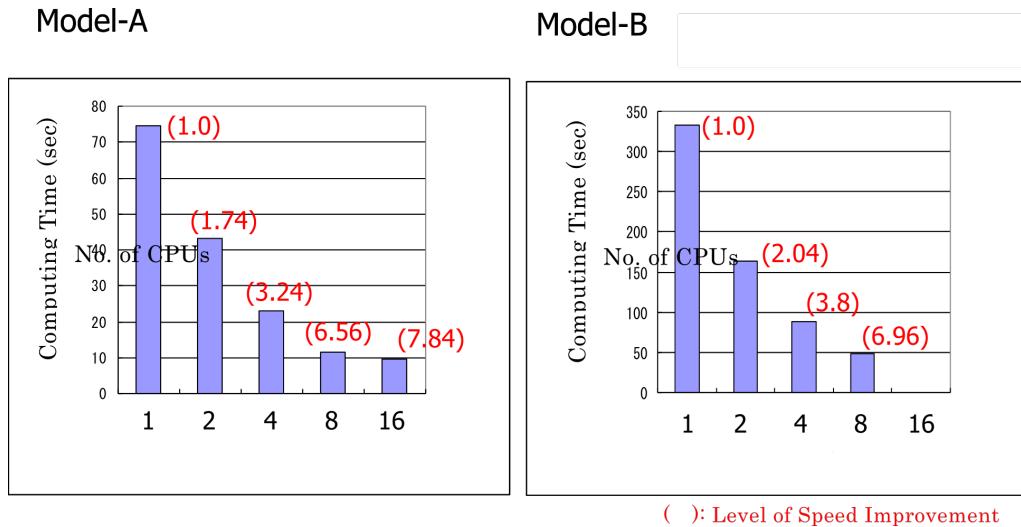


Fig. 9.4.7: Speed improvement degree because of dispersion treatment

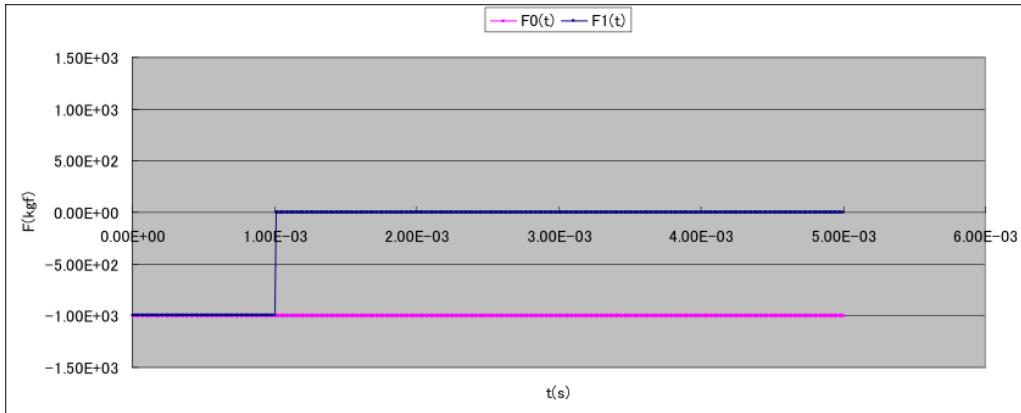
1.25 Actual Model Examples for Linear Dynamic Analysis

1.25.1 Analysis model

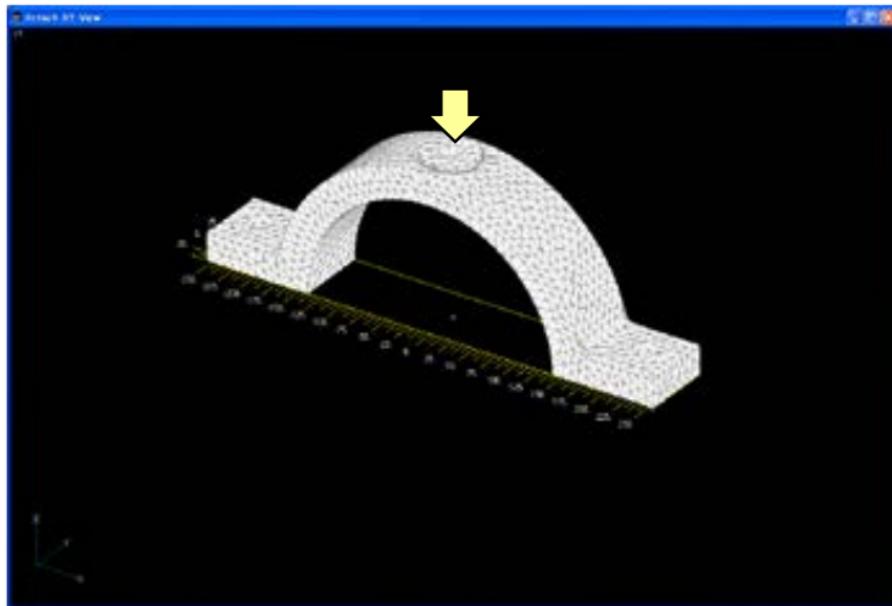
The linear dynamic analysis was performed with the machine parts shown in Fig. 9.5.1 as the actual models. For the analysis model, four cases were considered as verification examples with different load conditions and damping coefficients, as presented in Table 9.5.1.

Table 9.5.1: Verification Example of Actual Model for Linear Dynamic Analysis

Case Name	Element Type	Verification Model	Loading Conditions	Damping Conditions	No. of Nodes	No. of Degrees of Freedom
EX31A	342	Mesh model	Step load (F0)	No	15,214	45,642
EX31B	342		Step load (F0)	Yes	15,214	45,642
EX31C	342		Square wave pulse (F1)	No	15,214	45,642
EX31D	342		Square wave pulse (F1)	Yes	15,214	45,642



Loading Conditions

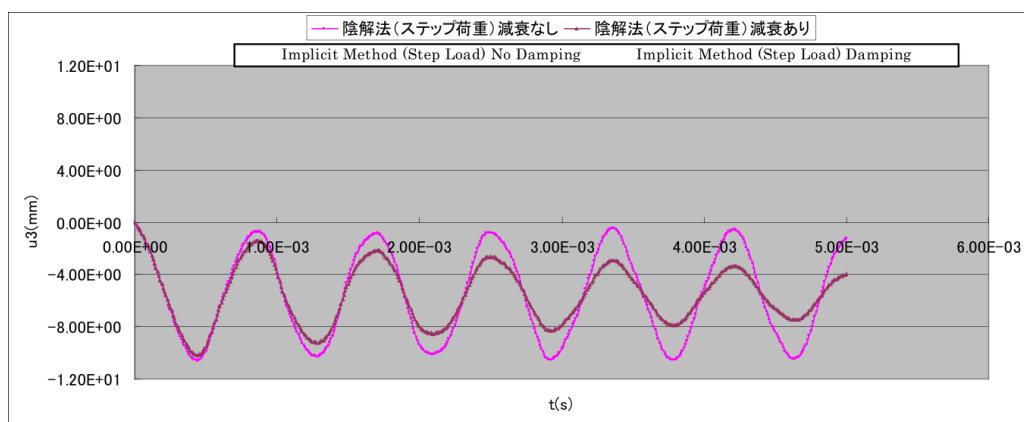


Mesh Figure

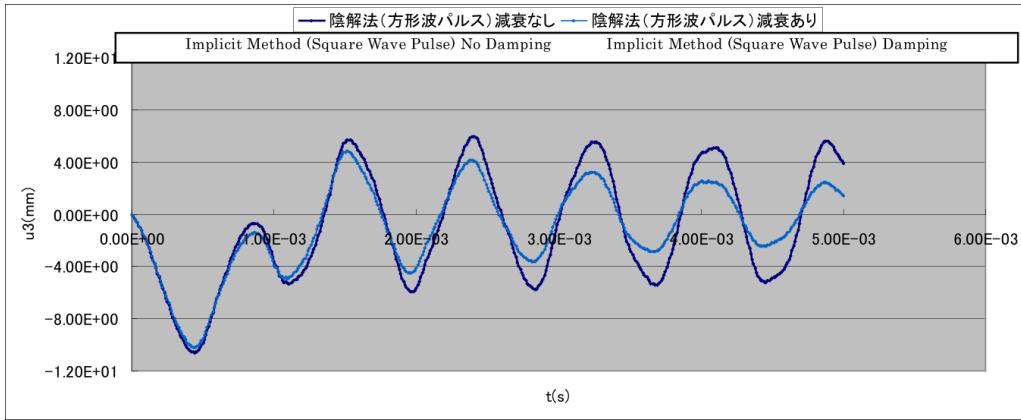
Fig. 9.5.1 : Mesh Model

1.25.2 Analysis results

Examples of the analysis results are shown in Fig. 9.5.2 ~ Fig. 9.5.3.

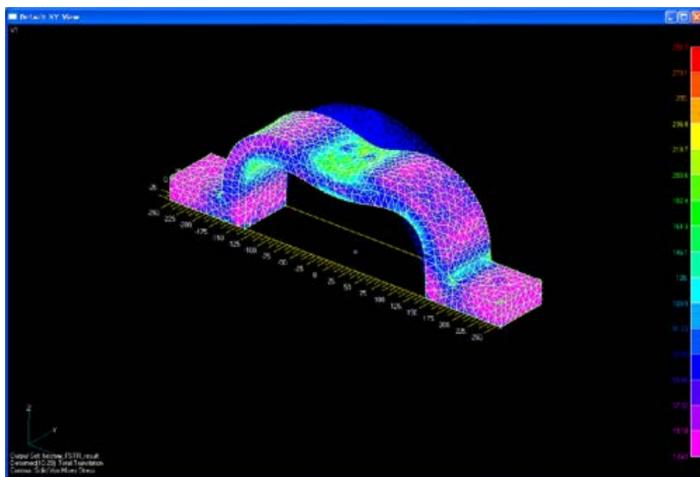


(a) In the case of Step Load

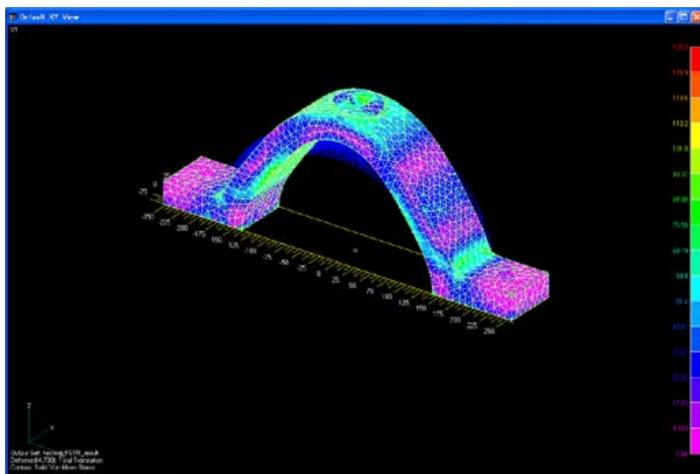


(b) In the case of Square Wave Pulse Load

Fig. 9.5.2: Time history of vibration point displacement u_z



(a) $t=5.0\text{E-}04(\text{s})$



(b) $t=4.0\text{E-}03(\text{s})$

Fig. 9.5.3: Deformation diagram and equivalent stress distribution (deformation ratio of 5.0), EX31C