

Research and Development for Next-generation Information Technology of
Ministry of Education, Culture, Sports, Science and Technology
"Research and Development of Innovative Simulation Software"

CISS Free Software

FrontISTR

Ver. 3.6

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 "CISS Free Software" by using this software at no charge. You shall conclude a contract separately when you use this software for the purpose of profit-making business. This software is protected by the copyright law and the other related laws, regarding unspecified issues in our license agreement and contract, or the condition without either license agreement or contract.

Corresponding Clerks:

(Engagement) The Foundation for the Promotion of Industrial Science (F.P.I.S)
4-6-1 Komaba, Meguro-ku, Tokyo 153-8505 JAPAN

(Management) Center for Research on Innovative Simulation Software,
Institute of Industrial Science (IIS), the University of Tokyo
4-6-1 Komaba, Meguro-ku, Tokyo 153-8505 JAPAN
Fax : +81-3-5452-6662
E-mail : software@ciss.iis.u-tokyo.ac.jp

Contents

1. Introduction	1
2. Notes for Use in this Release	1
3. Analysis Procedure	2
3.1 Analysis by Sequential Processing	2
3.1.1 Execution Flow	2
3.1.2 Preparation of Input File	2
3.1.3 Execution Procedure	3
3.1.4 Description of Output File	4
3.2 Analysis by Parallel Processing	6
3.2.1 Execution Flow	6
3.2.2 Preparation of Input File	7
3.2.3 Execution Procedure	8
3.2.4 Description of Output File	9
4. Example of Analysis	11
4.1 Static Analysis (Elasticity)	11
4.1.1 Analysis Object	11
4.1.2 Analysis Contents	11
4.1.3 Analysis Results	12
4.2 Static Analysis (Elasticity, Parallel)	13
4.3 Static Analysis (Hyperelasticity Part 1)	13
4.3.1 Analysis Object	13
4.3.2 Analysis Content	13
4.3.3 Analysis Results	14
4.4 Static Analysis (Hyperelasticity Part 2)	15
4.4.1 Analysis Object	15
4.4.2 Analysis Content	15
4.4.3 Analysis Results	16
4.5 Static Analysis (Elastoplasticity Part 1)	18
4.5.1 Analysis Object	18
4.5.2 Analysis Content	18
4.5.3 Analysis Results	19
4.6 Static Analysis (Elastoplasticity Part 2)	20
4.6.1 Analysis Object	20
4.6.2 Analysis Content	20
4.6.3 Analysis Results	21
4.7 Static Analysis (Viscoelasticity)	23
4.7.1 Analysis Object	23

4.7.2	Analysis Content.....	23
4.7.3	Analysis Results.....	23
4.8	Static Analysis (Creep).....	25
4.8.1	Analysis Object.....	25
4.8.2	Analysis Content.....	25
4.8.3	Analysis Results.....	25
4.9	Contact Analysis (Part 1).....	27
4.9.1	Analysis Object.....	27
4.9.2	Analysis Content.....	27
4.9.3	Analysis Results.....	28
4.10	Contact Analysis (Part 2).....	29
4.10.1	Analysis Object.....	29
4.10.2	Analysis Content.....	29
4.10.3	Analysis Results.....	30
4.11	Contact Analysis (Part 3).....	31
4.11.1	Analysis Object.....	31
4.11.2	Analysis Contents	31
4.11.3	Analysis Results.....	32
4.12	Linear Dynamic Analysis.....	33
4.12.1	Analysis Object.....	33
4.12.2	Analysis Contents	33
4.12.3	Analysis Results.....	34
4.13	Nonlinear Dynamic Analysis.....	35
4.13.1	Analysis Object.....	35
4.13.2	Analysis Content.....	35
4.13.3	Analysis Results.....	35
4.14	Nonlinear Contact Dynamic Analysis.....	36
4.14.1	Analysis Object.....	36
4.14.2	Analysis Content.....	37
4.14.3	Analysis Results.....	38
4.15	Eigenvalue Analysis	39
4.15.1	Analysis Object.....	39
4.15.2	Analysis Content.....	39
4.15.3	Analysis Results.....	39
4.16	Heat Conduction Analysis	41
4.16.1	Analysis Object.....	41
4.16.2	Analysis Content.....	41
4.16.3	Analysis Results.....	42
4.17	Frequency Response Analysis	42
4.17.1	Analysis Object.....	43

4.17.2	Analysis Content.....	43
4.17.3	Analysis Results.....	44

1. Introduction

This guide describes the analysis implementation guidelines using a large-scale structural analysis program using the finite element method FrontISTR based on examples. In addition, these examples intend for FrontISTR Ver.3.6.

2. Notes for Use in this Release

There are two versions of FrontISTR included in this release.

(1) FrontISTR Ver.3.6

All the functions of FrontISTR can be used in versions constructed with HEC-MW Ver.2.7. However, there are the following restrictions regarding the contact analysis function.

- When the MUMPS is linked, process parallel can be executed in the analysis by parallel processing. Also, when the Intel MKL is linked, thread parallel can be executed in the analysis by parallel processing. Before executing, set the environmental variables according to the computer environment being used.

(2) FrontISTR Ver.4.4

The following functions of FrontISTR can be used in versions constructed with HEC-MW Ver.4.4.

- Elastic Static Analysis
- Nonlinear Static Analysis (except for Contact Analysis)

Each version can be executed with the execution commands which are as follows.

(1) When executing FrontISTR Ver.3.6

hecmw_part1、 fistr1、 hecmw_vis1

(2) When executing FrontISTR Ver.4.4

hecmw_part2、 fistr2、 hecmw_vis2

3. Analysis Procedure

3.1 Analysis by Sequential Processing

3.1.1 Execution Flow

The execution flow by sequential processing of a single processor using FrontISTR is shown in Figure 3.1.1.

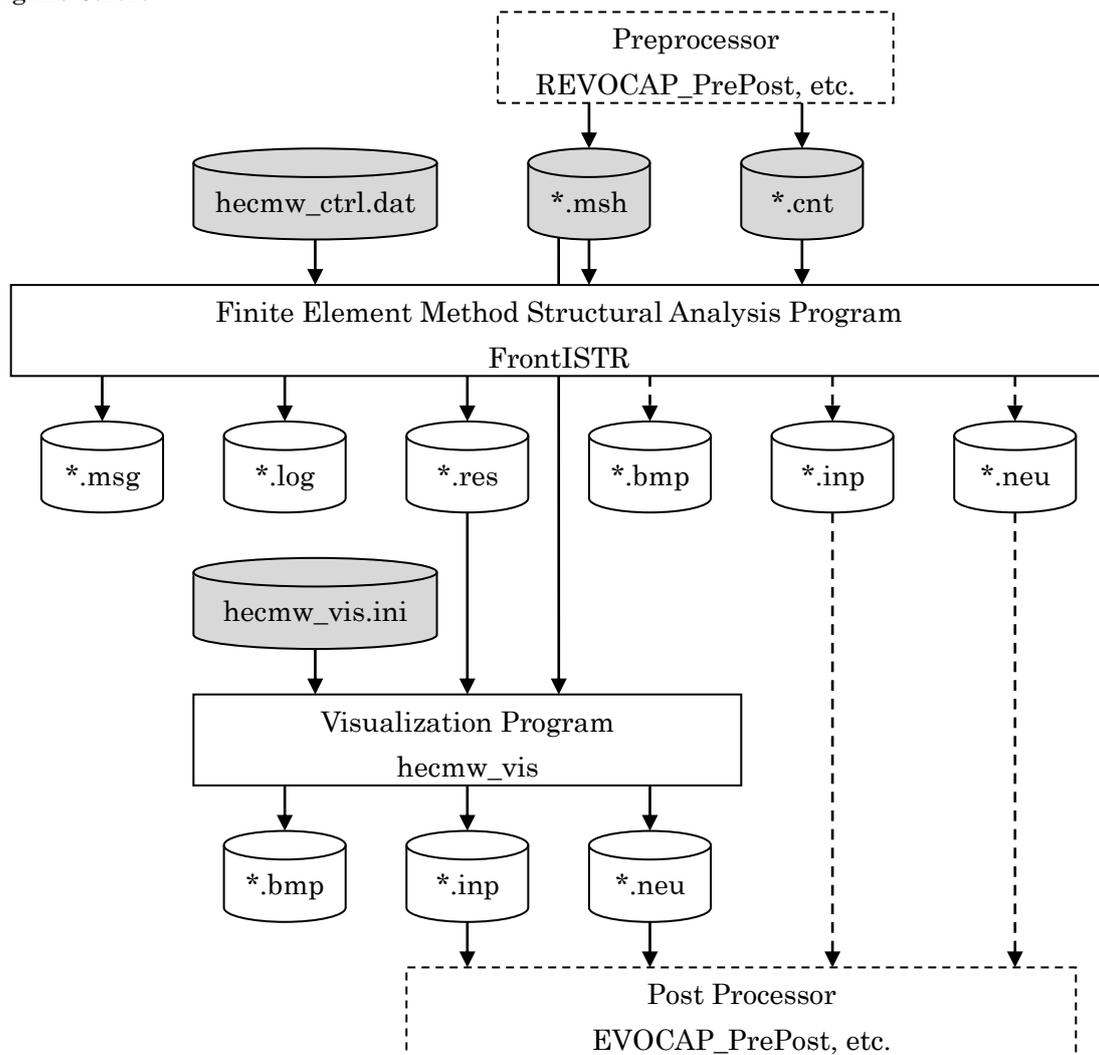


Figure 3.1.1: Execution Flow by Sequential Processing

3.1.2 Preparation of Input File

(1) Overall control data (Ext. dat)

The input file and the analysis results output file of the mesh data and analysis control data are specified in this file. The fixed file name is hecmw_ctrl.dat.

An example of the overall control data is shown in the following. In this example, FrontSTR reads the single domain mesh data model.msh and analysis control data model.cnt, and

writes the analysis output data model.res.0.1. Moreover, hecmw_vis reads the single domain mesh data model.msh and analysis results data model.res.0.1, and writes the model_vis_psf.0000. (Ext.) corresponding to the specified output. For details, refer to Chapter 5 of the User's Manual.

```
#
# for solver
#
!MESH, NAME=fstrMSH, TYPE=HECMW-ENTIRE
model.msh
!CONTROL, NAME=fstrCNT
model.cnt
!RESULT, NAME=fstrRES, IO=OUT
model.res
!RESULT, NAME=vis_out, IO=OUT
model_vis
```

(2) Single domain mesh data (Ext. msh)

The overall mesh configuration applicable for analysis, material data, group data used in the analysis control data and etc., are defined in this file. For details, refer to Chapter 6 of the User's Manual.

(3) Analysis control data (Ext. cnt)

The analysis classification, displacement boundary conditions, load boundary conditions and etc., are defined in this file. The solver control data and the visualizer control data are also specified in this file. An example of the analysis control data is shown in Chapter 3. For details, refer to Chapter 7 of the User's Manual.

(4) Visualization control data (Ext. ini)

The control data of hecmw_vis is specified in this file. The default file name is hecmw_vis.ini. An example of the visualization control data is shown in the following. In this example, an unstructured mesh type data for MicroAVS (Ext. inp) is output. For details, refer to Section 7.3.3 and Section 7.4.7 of the User's Manual.

```
!VISUAL, method=PSR, visual_start_step=1, visual_interval_step=1, visual_end_step=1
!surface_num = 1
!surface 1
!output_type = complete_avs
```

3.1.3 Execution Procedure

Execute FrontISTR with the following command line in the directory where the input file is saved.

```
$ fistr1
```

Two methods can be used to execute the visualization. The first method is used when executing as a post process of FrontISTR. By specifying !WRITE, VISUAL in the analysis control data, the visualization will automatically be executed. In this case, it is necessary to describe the visualization control data included in the analysis control data.

When executing the visualization after the execution of FrontISTR is completed, first specify !WRITE, RESULT in the analysis control data, and then execute FrontISTR.

After the execution of FrontISTR is completed, execute hecmw_vis by the following command line in the directory where the input file and analysis results file are saved.

```
$ hecmw_vis1
```

3.1.4 Description of Output File

(1) Analysis results message file (Ext. msg)

Messages, such as the analysis progression process of FrontISTR will be output in this file. One file will be created with one execution, and the fixed file name is FSTR.msg.

(2) Analysis results log file (Ext. log)

The physical quantity analysis results of each node and element of FrontISTR will be output in this file. The analysis results of the maximum, minimum and eigenvalues of the physical quantity will also be output in this file. In the case of dynamic analysis, the analysis results of all the steps will be output in this file. One file will be created with one execution, and the fixed file name is 0.log.

(3) Analysis results file (no Ext.)

The analysis results will be output in this file, when the !WRITE, RESULT option is specified.

The physical quantity analysis results of each node and element of FrontISTR will be output in this file. A file will be created for each step, and the file will be named as follows using the file header specified in the overall control data.

Naming rule: (File header specified by !RESUT).0. (Step No.)

Example: model.res.0.1

(4) Analysis results bitmap file (Ext. bmp)

The analysis results will be output in this file when specified in the visualization control data.

The visualized bitmap data will be output in this file. The file will be named using the file header specified in the overall control data. For details on the naming rules, refer to the hecmw1 document (0803_001f_hecmw_PC_cluster_201_vis.pdf).

(5) Analysis results unstructured mesh type data file (Ext. inp)

The analysis results will be output in this file when specified in the visualization control data.

The post process can be executed with REVOCAP_PrePost, MicroAVS and etc. using this file. The file will be named as follows, using the file header specified in the overall control data.

Naming rule: (File header specified by !RESUT)_psf.(Step No).inp

Example: model_vis_psf.0000.inp

(6) Analysis results neutral file (Ext. neu)

The analysis results will be output in this file when specified in the visualization control data.

The post process can be executed with Femap using this file. The file will be named as follows, using the file header specified in the overall control data.

Naming rule: (File header specified by !RESUT)_psf.(Step No.).neu

Example: model_vis_psf.0000.neu

Note: In addition to the above, the FSTR.dbg file will be output; however, since this is for debugging, it is normally not necessary to refer to this file.

3.2 Analysis by Parallel Processing

3.2.1 Execution Flow

The execution flow by parallel processing of a multiprocessor using FrontISTR is shown in Figure 3.2.1.

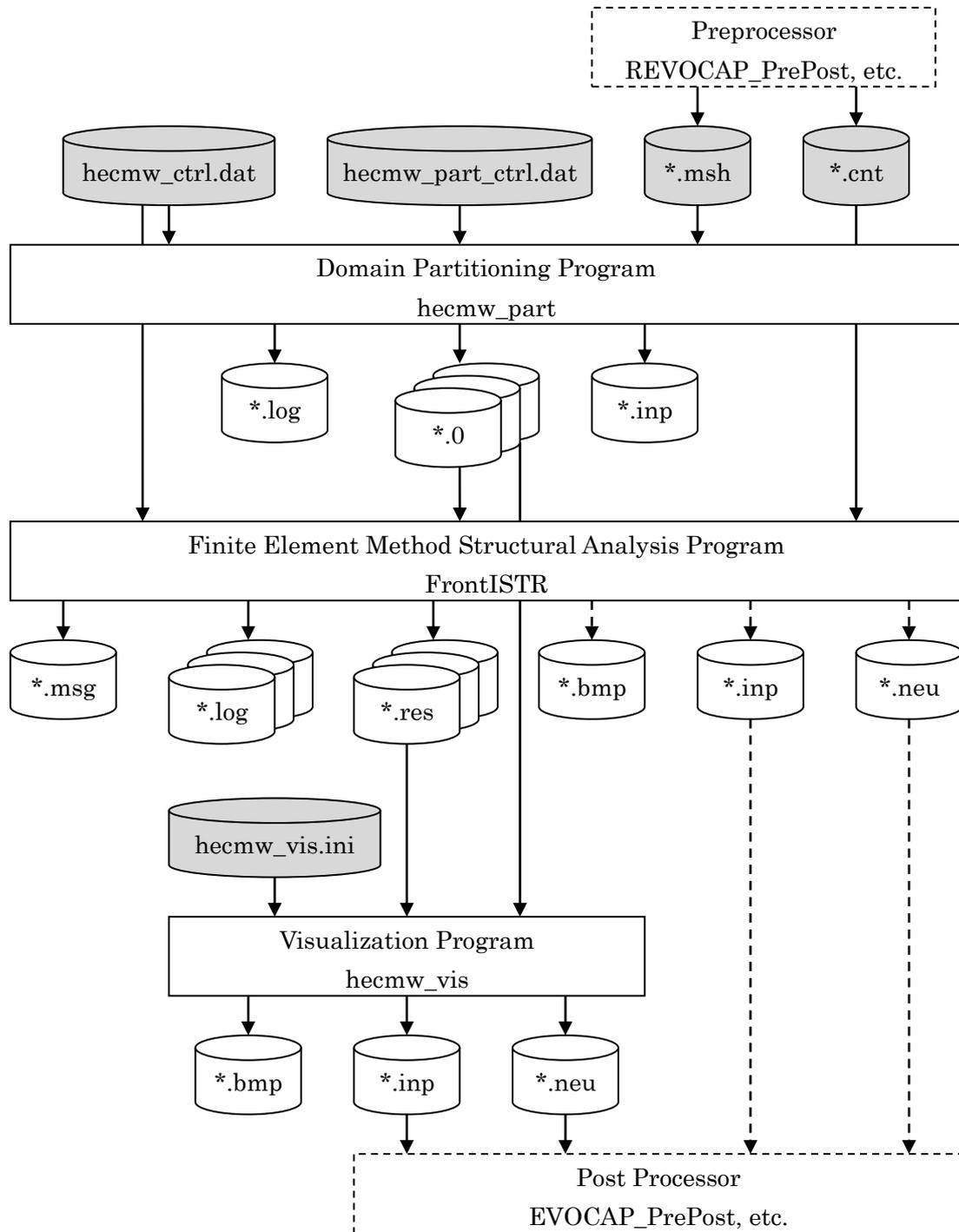


Figure 3.2.1: Execution Flow by Parallel Processing

3.2.2 Preparation of Input File

(1) Overall control data (Ext. dat)

The input file and the analysis results output file of the mesh data and analysis control data are specified in this file. The fixed file name is `hecmw_ctrl.dat`.

An example of the overall control data is shown in the following. In this example, first, `hecmw_part` reads the single domain mesh data `model.msh`, and writes the distributed domain mesh data `model_8.0~n`. `FrontSTR` reads the distributed domain mesh data `model_8.0~n` and the analysis control data `model.cnt`, and writes the analysis results data `model.res.0~n.1`. Moreover, `hecmw_vis` reads the distributed domain mesh data `model_8.0~n` and the analysis results data `model.res.0~n.1`, and writes the `model_vis_psf.0000`. (Ext.) corresponding to the specified output. For details, refer to Chapter 5 of the User's Manual.

```
#
# for partitioner
#
!MESH, NAME=part_in, TYPE=HECMW-ENTIRE
model.msh
!MESH, NAME=part_out, TYPE=HECMW-DIST
model_8
#
# for solver
#
!MESH, NAME=fstrMSH, TYPE=HECMW-DIST
Model_8
!CONTROL, NAME=fstrCNT
model.cnt
!RESULT, NAME=fstrRES, IO=OUT
model.res
!RESULT, NAME=vis_out, IO=OUT
model_vis
```

(2) Single domain mesh data (Ext. msh)

The overall mesh configuration applicable for analysis, material data, group data used in the analysis control data and etc., are defined in this file. For details, refer to Chapter 6 of the User's Manual.

(3) Analysis control data (Ext. cnt)

The analysis classification, displacement boundary conditions, load boundary conditions and etc., are defined in this file. The solver control data and the visualizer control data are also specified in this file. An example of the analysis control data is shown in Chapter 3. For details, refer to Chapter 7 of the User's Manual.

(4) Domain partitioning utility control data (Ext. dat)

The control data of `hecmw_part` is specified in this file. The fixed file name is

hecmw_part_ctrl.dat. An example of the domain partitioning utility control data is shown in the following. In this example, a single domain is partitioned into 8 domains by the domain decomposition method PMETIS. Moreover, file model_8.inp will be output to display the mesh after the domains are partitioned. For details, refer to the hecmw1 document (0803_001x_hecmw_part_201_users.pdf).

```
!PARTITION,TYPE=NODE-BASED,METHOD=PMETIS,DOMAIN=8,UCD=model_8.inp
```

(5) Visualization control data (Ext. ini)

The control data of hecmw_vis is specified in this file. The default file name is hecmw_vis.ini. An example of the visualization control data is shown in the following. In this example, an unstructured mesh type data for MicroAVS (Ext. inp) is output. For details, refer to Section 7.3.3 and Section 7.4.7 of the User's Manual.

```
!VISUAL, method=PSR, visual_start_step=1, visual_interval_step=1, visual_end_step=1  
!surface_num = 1  
!surface 1  
!output_type = complete_avs
```

3.2.3 Execution Procedure

Execute hecmw_part with the following command line in the directory where the input file is saved.

```
$ hecmw_part1
```

Execute FrontISTR with the following command line in the directory where the input file is saved. In addition, in the execution procedure of the MPI process, it is necessary to make corrections according to each environment.

```
$ mpirun -np 8 fistr1
```

Two methods can be used to execute the visualization. The first method is used when executing as a post process of FrontISTR. By specifying `!WRITE, VISUAL` in the analysis control data, the visualization will automatically be executed. In this case, it is necessary to describe the visualization control data included in the analysis control data.

When executing the visualization after the execution of FrontISTR is completed, first specify `!WRITE, RESULT` in the analysis control data, and then execute FrontISTR.

After the execution of FrontISTR is completed, execute hecmw_vis by the following command line in the directory where the input file and analysis results file are saved. In addition, in the execution procedure of the MPI process, it is necessary to make corrections according to each environment.

```
$ mpirun -np 8 hecmw_vis1
```

3.2.4 Description of Output File

(1) Domain partitioning utility log file (Ext. log)

Messages, such as the analysis progression process of hecmw_part will be output in this file. The fixed file name is hecmw_part.log.

(2) Distributed domain mesh file (no Ext.)

The mesh configuration partitioned into domains, material data, group data used in the analysis control data and etc, will be output in this file. A file will be created for each distributed domains, and the file will be named as follows using the file header specified in the overall control data.

Naming rule: (File header specified by !MESH).(Distributed domain number)

Example: model_8.0 ~ model_8.7

(3) File for domain partitioning mesh display (Ext. inp)

The unstructured mesh type data to display the mesh partitioned into domains will be output in this file. This can be displayed by MicroAVS, etc.

(4) Analysis results message file (Ext. msg)

Messages, such as the analysis progression process of FrontISTR will be output in this file. One file will be created with one execution, and the fixed file name is FSTR.msg.

(5) Analysis results log file (Ext. log)

The physical quantity analysis results of each node and element of FrontISTR will be output in this file. The analysis results of the maximum, minimum and eigenvalues of the physical quantity will also be output in this file. In the case of dynamic analysis, the analysis results of all the steps will be output in this file. A file will be created for each distributed domain, and the fixed file name is n.log (n is the distributed domain number).

(6) Analysis results file (no Ext.)

The analysis results will be output in this file, when the !WRITE, RESULT option is specified.

The physical quantity analysis results of each node and element of FrontISTR will be output in this file. A file will be created for each distributed domain and step, and the file will be named as follows using the file header specified in the overall control data.

Naming rule: (File header specified by !RESUT).(Distributed domain number).(Step number)

Example: model_8.res.0.1 ~ model_8.res.7.1

(7) Analysis results bitmap file (Ext. bmp)

The analysis results will be output in this file when specified in the visualization control data.

The visualized bitmap data will be output in this file. The file will be named using the file header specified in the overall control data. For details on the naming rules, refer to the hecmw1 document (0803_001f_hecmw_PC_cluster_201_vis.pdf).

(8) Analysis results unstructured mesh type data file (Ext. inp)

The analysis results will be output in this file when specified in the visualization control data.

The post process can be executed with REVOCAP_PrePost, MicroAVS and etc. using this file. The file will be named as follows, using the file header specified in the overall control data.

Naming rule: (File header specified by !RESUT)_psf.(Step No.).inp

Example: model_vis_psf.0000.inp

(9) Analysis results neutral file (Ext. neu)

The analysis results will be output in this file when specified in the visualization control data.

The post process can be executed with Femap using this file. The file will be named as follows, using the file header specified in the overall control data.

Naming rule: (File header specified by !RESUT)_psf.(Step No).neu

Example: model_vis_psf.0000.neu

Note: In addition to the above, the FSTR.dbg.0 ~ n files will be output; however, since these are for debugging, it is normally not necessary to refer to these files.

4. Example of Analysis

4.1 Static Analysis (Elasticity)

Data of tutorial/01_elastic_hinge/ is used for the implementation of this analysis.

4.1.1 Analysis Object

A hinge component is the object of the analysis. The shape is shown in Figure 4.1.1, and the mesh data is shown in Figure 4.1.2. Quadratic tetrahedral elements are used for the mesh, and the scale of the mesh consists of 49,871 elements and 84,056 nodes.

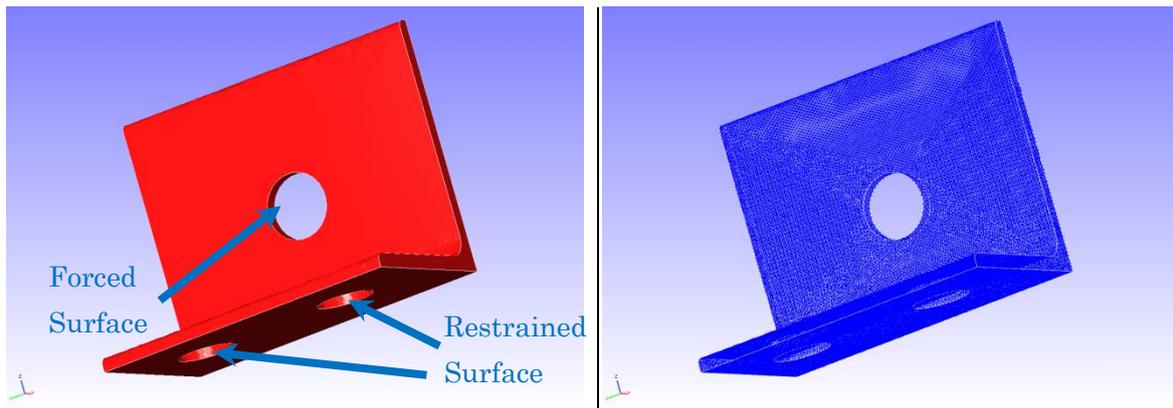


Figure 4.1.1: Shape of Hinge Component

Figure 4.1.2: Mesh Data of Hinge Component

4.1.2 Analysis Contents

A stress analysis is implemented, where the displacement of the restrained surface shown in Figure 4.1.1 is restrained, and a concentrated load is applied to the forced surface. The analysis control data is shown in the following.

```
# Control File for FISTR
## Analysis Control
!VERSION
  3
!SOLUTION, TYPE=STATIC
!WRITE, RESULT
!WRITE, VISUAL
## Solver Control
### Boundary Conditon
!BOUNDARY
  BND0, 1, 3, 0.000000
!BOUNDARY
  BND1, 1, 3, 0.000000
!CLOAD
  CLO, 1, 1.00000
### Material
!MATERIAL, NAME=STEEL
!ELASTIC
  210000.0, 0.3
!DENSITY
  7.85e-6
```

```

### Solver Setting
!SOLVER, METHOD=CG, PRECOND=1, ITERLOG=YES, TIMELOG=YES
10000, 2
1.0e-8, 1.0, 0.0

```

4.1.3 Analysis Results

A contour figure of the Mises stress was created by REVOCAP_PrePost, and is shown in Figure 4.1.3. Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

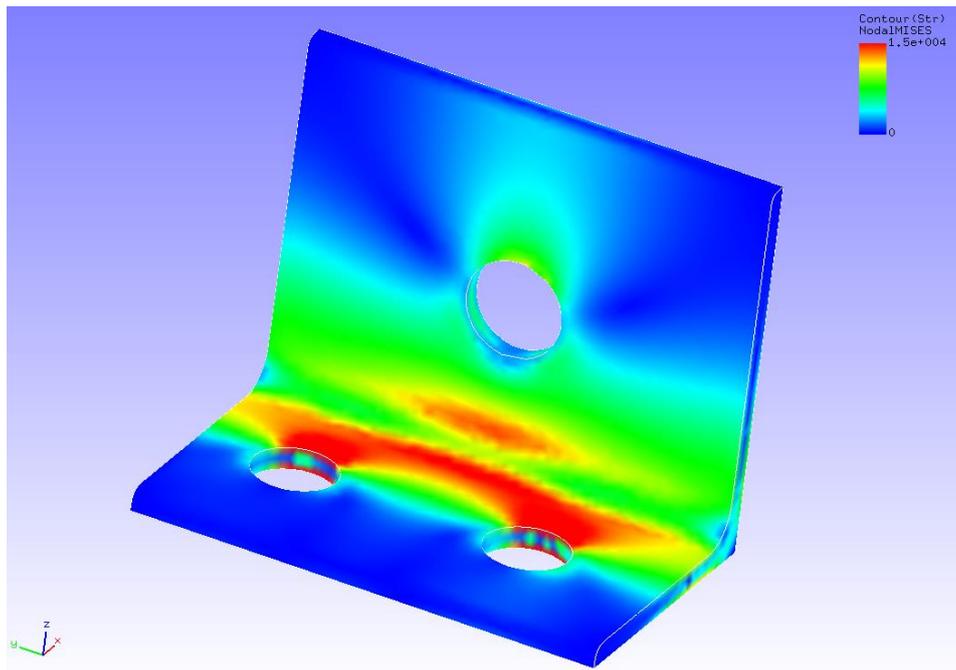


Figure 4.1.3: Analysis Results of Mises Stress

```

#### Result step= 1
##### Local Summary :Max/IdMax/Min/IdMin####
//U1 3.9115E+00 82452 -7.1083E-02 65233
//U2 7.4504E-03 354 -5.8813E-02 696
//U3 5.9493E-02 84 -5.8751E-01 61080
//E11 1.3777E-01 130 -1.3653E-01 77625
//E22 4.9199E-02 61 -5.4370E-02 102
//E33 6.8634E-02 51036 -6.1176E-02 30070
//E12 7.1556E-02 27808 -6.8093E-02 27863
//E23 5.3666E-02 56 -5.4347E-02 82
//E13 7.2396E-02 36168 -9.6621E-02 130
//S11 3.8626E+04 130 -3.6387E+04 28580
//S22 1.6628E+04 130 -1.5743E+04 28580
//S33 1.6502E+04 30033 -1.5643E+04 28580
//S12 5.7795E+03 27808 -5.4998E+03 27863
//S23 4.3345E+03 56 -4.3896E+03 82
//S13 5.8474E+03 36168 -7.8040E+03 130
//SMS 2.8195E+04 77625 1.2755E+00 75112

```

4.2 Static Analysis (Elasticity, Parallel)

Data of tutorial/02_elastic_hinge_parallel/ is used to implement the analysis of Section 4.1 in four-parallel.

4.3 Static Analysis (Hyperelasticity Part 1)

Data of tutorial/ 03_hyperelastic_cylinder/ is used to implement this analysis.

4.3.1 Analysis Object

The object for analysis is a 1/8 model of a cylinder. The shape is shown in Figure 4.3.1, and the mesh data is shown in Figure 4.3.2. Hexahedral linear elements are used for the mesh, and the scale of the mesh consists of 432 elements and 629 nodes.

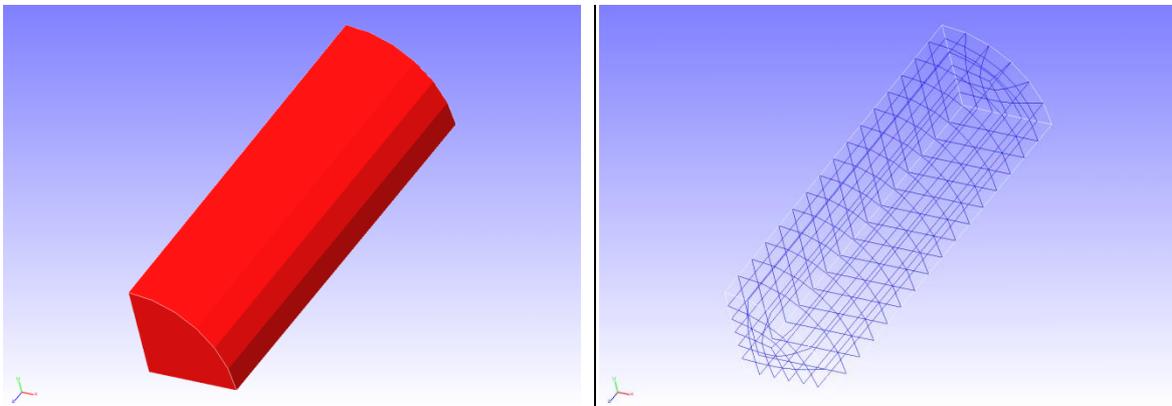


Figure 4.3.1: Shape of Cylinder (1/8 Model) Figure 4.3.2: Mesh Data of Cylinder (1/8 model)

4.3.2 Analysis Content

Stress analysis is implemented where tension displacement is applied to the cylinder in the axial direction. The Mooney-Rivlin model is used for the constitutive equation of the material of the hyperelasticity. The analysis control data is shown in the following.

```
# 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
!HYPERELASTIC, TYPE=MOONEY-RIVLIN
0.1486, 0.4849, 0.0789
### Solver Setting
!SOLVER, METHOD=CG, PRECOND=1, ITERLOG=YES, TIMELOG=YES
10000, 2
1.0e-8, 1.0, 0.0
```

4.3.3 Analysis Results

As analysis results of the 5th sub step, a deformed figure applied with a contour of the Mises stress was created by REVOCAP_PrePost, and is shown in Figure 4.3.3. Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

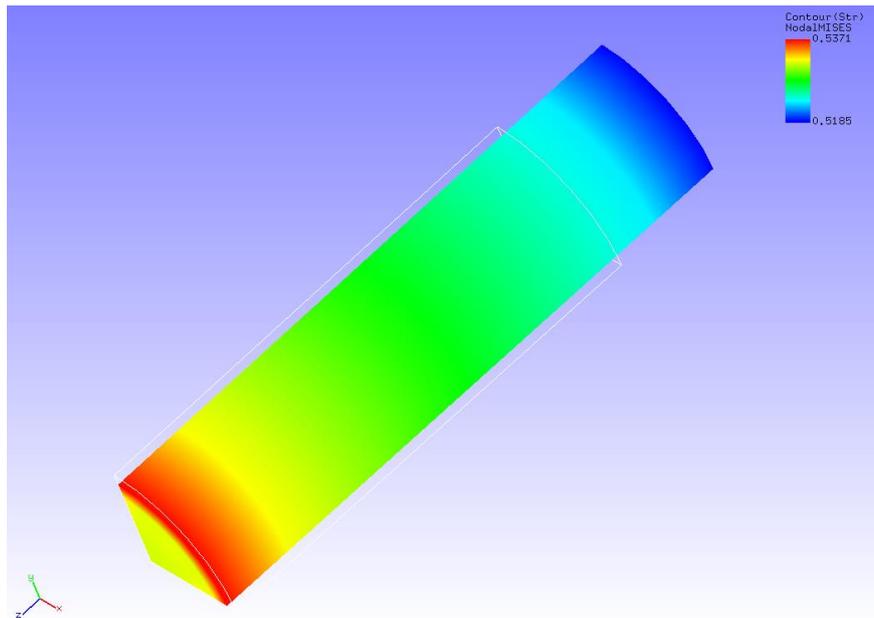


Figure 4.3.3: Analysis Results of Deformation and Mises Stress

```

#### Result step=      5
##### Local Summary :Max/IdMax/Min/IdMin#####
//U1  0.0000E+00      1 -6.7543E-01      7
//U2  0.0000E+00      1 -6.7543E-01     13
//U3  0.0000E+00      1 -7.0000E+00     38
//E11 -9.6960E-02     38 -1.0234E-01      7
//E22 -9.6960E-02     50 -1.0234E-01     13
//E33  3.0653E-01     13  2.8767E-01     38
//E12  6.9417E-04     53 -7.0552E-04     10
//E23  5.8123E-08     39 -3.2652E-03     86
//E13  5.8123E-08     49 -3.2652E-03     93
//S11  5.8544E-03     38 -6.3700E-03      7
//S22  5.8544E-03     50 -6.3701E-03     13
//S33  5.3515E-01     35  5.2022E-01     64
//S12  1.5492E-03     53 -1.6314E-03     10
//S23  1.7965E-07     38 -2.1555E-03     86
//S13  1.7965E-07     50 -2.1555E-03     93
//SMS  5.3711E-01     10  5.1849E-01     53

```

4.4 Static Analysis (Hyperelasticity Part 2)

Data of tutorial/ 04_hyperelastic_spring/ is used to implement this analysis.

4.4.1 Analysis Object

A spring is the object of the analysis. The shape is shown in Figure 4.4.1, and the mesh data is shown in Figure 4.4.2. Quadratic tetrahedral elements are used for the mesh, and the scale of the mesh consists of 46,454 elements and 78,771 nodes.

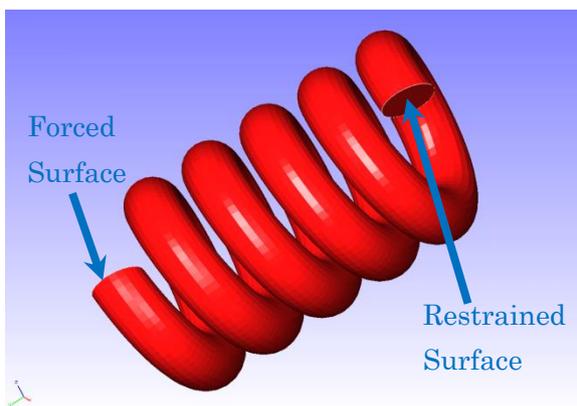


Figure 4.4.1: Shape of Spring

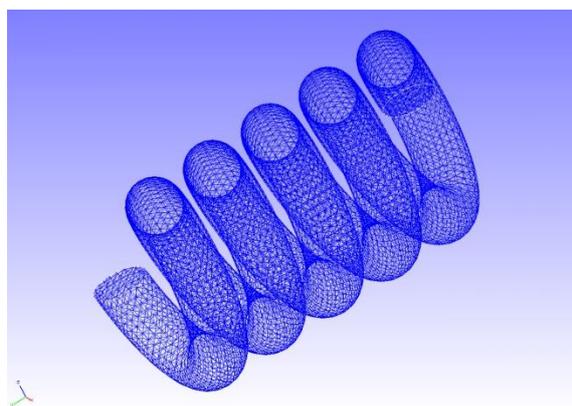


Figure 4.4.2: Mesh Data of Spring

4.4.2 Analysis Content

A stress analysis is implemented, where the displacement of the restrained surface shown in Figure 4.4.1 is restrained, and a displacement is applied to the forced surface. The Arruda-Boyce model is used for the constitutive equation of the material of the hyperelasticity. The analysis control data is shown in the following.

```

# Control File for FISTR
## Analysis Control
!VERSION
  3
!SOLUTION, TYPE=NLSTATIC
!WRITE, RESULT
!WRITE, VISUAL
## Solver Control
### Boundary Conditon
!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, 2
  1.0e-8, 1.0, 0.0

```

4.4.3 Analysis Results

A deformed figure applied with a displacement contour was created by REVOCAP_PrePost, and is shown in Figure 4.4.3. Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

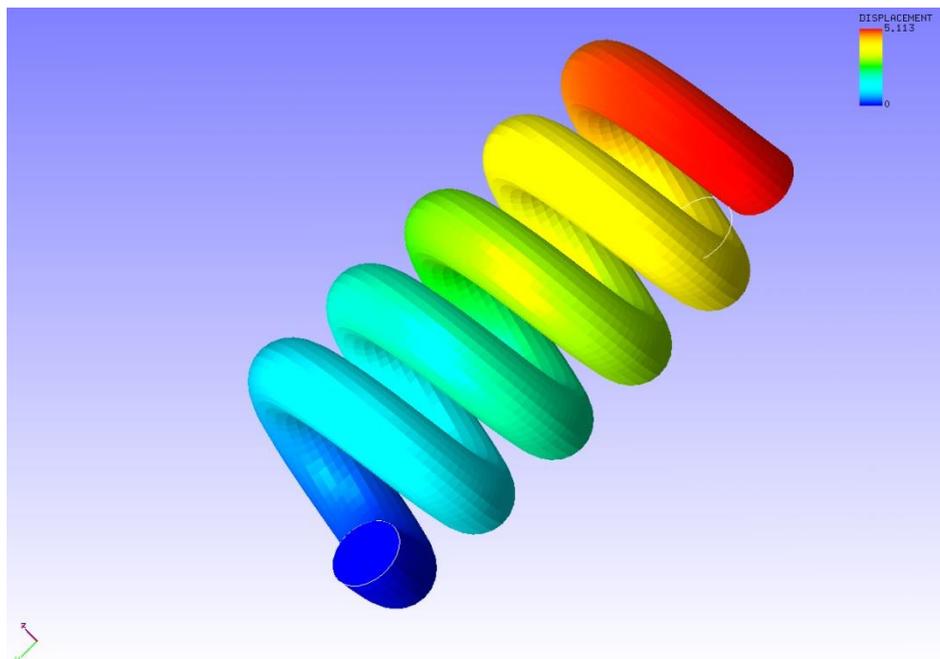


Figure 4.4.3: Analysis Results of Deformation and Displacement

```

#### Result step=      1
##### Local Summary :Max/IdMax/Min/IdMin#####
//U1  2. 8588E-01      42179 -2. 6512E-01      22274
//U2  2. 2657E-02       6381 -5. 0291E+00      22825
//U3  7. 4573E-02       7058 -9. 5095E-01      48324
//E11 4. 8291E-03       2851 -4. 2788E-03       3429
//E22 2. 4161E-03      55960 -1. 4539E-03      44761
//E33 5. 3256E-03      25260 -4. 6858E-03      27938
//E12 1. 3574E-02      56003 -1. 3081E-02      45120
//E23 2. 8679E-02      48353 -1. 8970E-02      48322
//E13 1. 0897E-02      47938 -9. 1054E-03      27344
//S11 5. 1605E-02       2814 -5. 0895E-03      10408
//S22 5. 0635E-02      55965 -3. 6174E-03      45307
//S33 4. 9662E-02      39836 -5. 1017E-03       4949
//S12 1. 2059E-02      56003 -1. 1865E-02      45120
//S23 2. 6123E-02      48353 -1. 7281E-02      56868
//S13 1. 0133E-02      47938 -8. 2330E-03      27344
//SMS 4. 9365E-02      48353  3. 2148E-04      64553

```

4.5 Static Analysis (Elastoplasticity Part 1)

Data of tutorial/ 05_plastic_cylinder / is used to implement this analysis.

4.5.1 Analysis Object

The same 1/8 model cylinder as the static analysis (hyperelasticity part 1) in Section 4.3 is the object of the analysis.

4.5.2 Analysis Content

The necking phenomenon of the cylinder by plastic deformation is analyzed. The Mises model is used for the yield function. The analysis control data is shown in the following.

```
# Control File for FISTR
## Analysis Control
!VERSION
 3
!SOLUTION, TYPE=NLSTATIC
!WRITE, RESULT, FREQUENCY=10
!WRITE, VISUAL, FREQUENCY=10
## 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=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, 2
 1.0e-8, 1.0, 0.0
```

4.5.3 Analysis Results

As analysis results of the 35th sub step, a deformed figure applied with a contour of the Mises stress was created by REVOCAP_PrePost, and is shown in Figure 4.5.1. Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

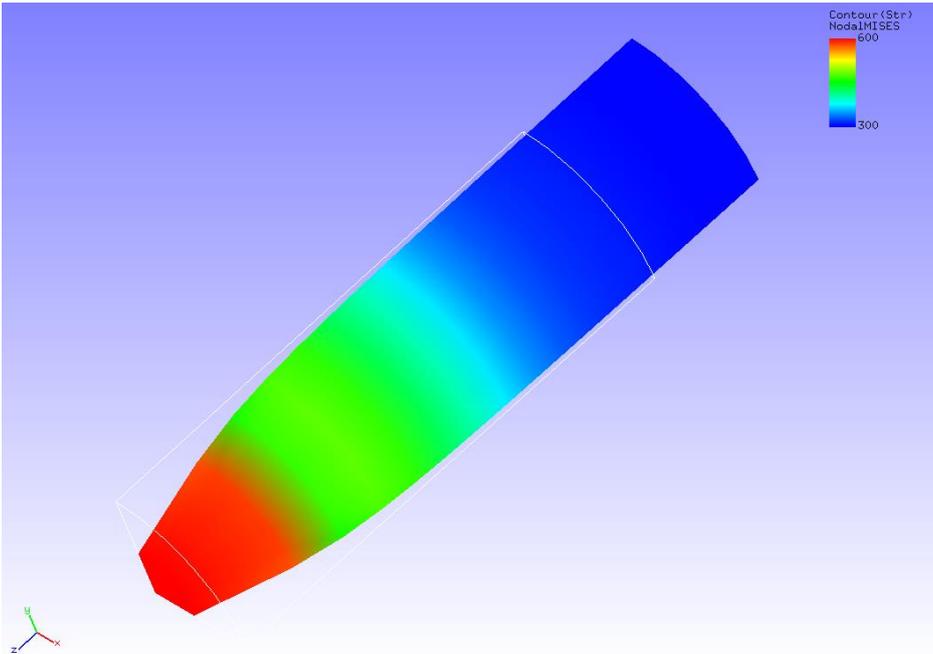


Figure 4.5.1: Analysis Results of Deformation and Mises Stress

```

#### Result step= 40
##### Local Summary :Max/IdMax/Min/IdMin####
//U1 0.0000E+00 1 -3.5930E+00 7
//U2 0.0000E+00 1 -3.5930E+00 13
//U3 0.0000E+00 1 -7.0000E+00 38
//E11 -3.9417E-02 38 -6.5298E-01 16
//E22 -3.9417E-02 50 -6.5298E-01 4
//E33 1.3083E+00 1 7.9614E-02 50
//E12 6.9553E-02 10 -1.7556E-02 368
//E23 1.5953E-02 90 -7.1473E-01 13
//E13 1.5954E-02 89 -7.1473E-01 7
//S11 1.9746E+02 86 -3.6807E+02 192
//S22 1.9746E+02 93 -3.6807E+02 192
//S33 9.1649E+02 1 -6.4716E+01 191
//S12 6.3257E-01 53 -1.2521E+02 406
//S23 5.7963E+01 191 -2.0766E+02 89
//S13 5.7963E+01 191 -2.0766E+02 90
//SMS 7.9001E+02 13 2.1264E+02 189

```

4.6 Static Analysis (Elastoplasticity Part 2)

Data of tutorial/ 06_plastic_can / is used to implement this analysis.

4.6.1 Analysis Object

The object for analysis is a 1/2 model of a can. The shape is shown in Figure 4.6.1, and the mesh data is shown in Figure 4.6.2. Quadratic tetrahedral elements are used for the mesh, and the scale of the mesh consists of 7,236 elements and 14,119 nodes.

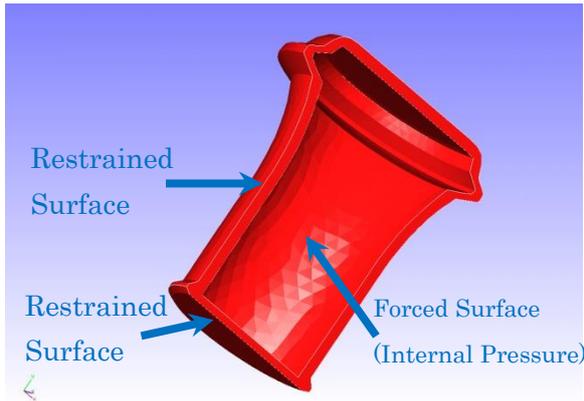


Figure 4.6.1: Shape of Can

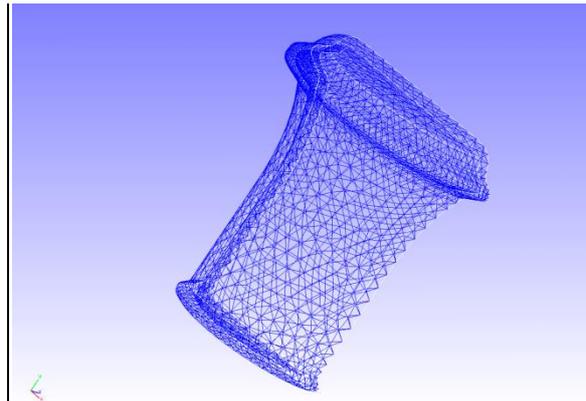


Figure 4.6.2: Mesh Data of Can

4.6.2 Analysis Content

A stress analysis is implemented, where the displacement of the restrained surface shown in Figure 4.6.1 is restrained, and a distributed load was applied to the forced surface inside of the can. The Drucker-Prager model is used for the yield function. The analysis control data is shown in the following.

```
# Control File for FISTR
## Analysis Control
!VERSION
3
!SOLUTION, TYPE=NLSTATIC
## Solver Control
### Boundary Conditon
!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
DLO, 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, 2
1.0e-8, 1.0, 0.0
```

4.6.3 Analysis Results

As analysis results of the 10th sub step, a deformed figure applied with a contour of the Mises stress was created by REVOCAP_PrePost, and is shown in Figure 4.6.3. The deformation magnification is set to 30. Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

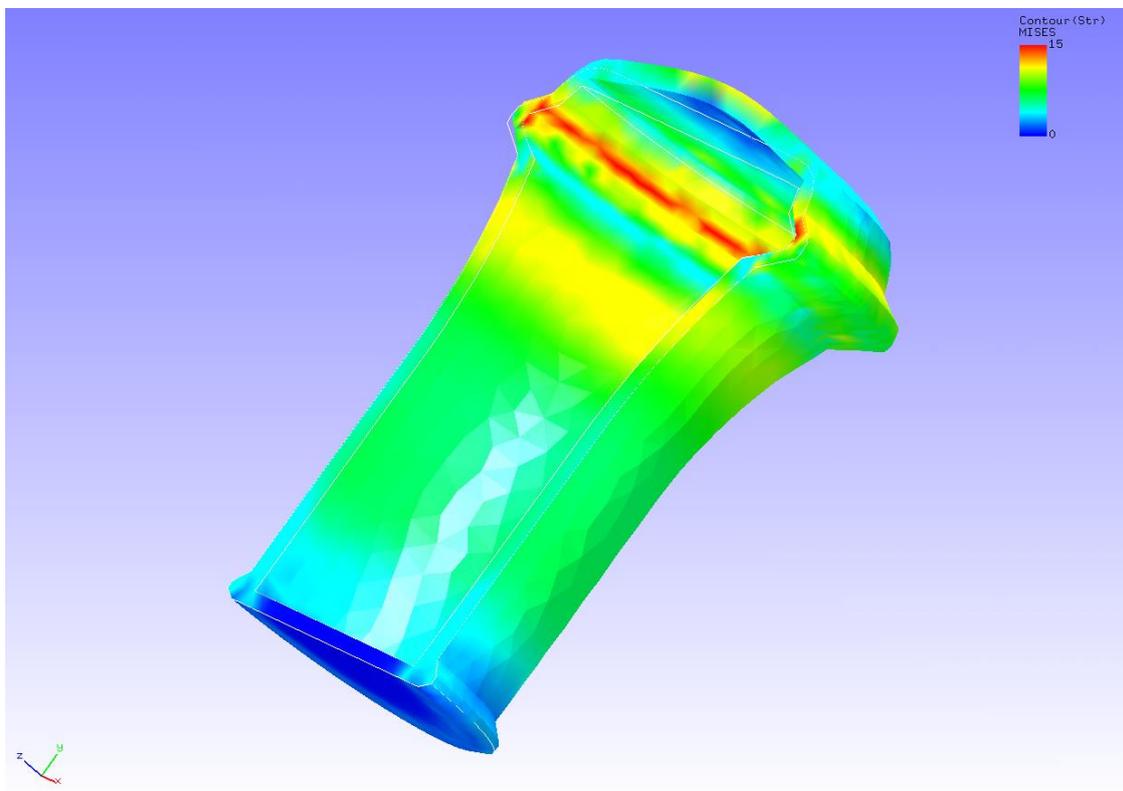


Figure 4.6.3: Analysis Results of Deformation and Mises Stress

```

#### Result step= 10
##### Local Summary :Max/IdMax/Min/IdMin#####
//U1 1. 6235E+00 1600 -1. 6188E+00 11901
//U2 1. 9319E+01 6877 -4. 5377E-01 7096
//U3 1. 6152E+00 7016 -1. 5121E+00 6934
//E11 9. 9346E-04 11242 -6. 5987E-04 1404
//E22 1. 5038E-03 13972 -5. 4264E-04 2367
//E33 9. 8561E-04 6833 -6. 4870E-04 7000
//E12 1. 6845E-03 2698 -1. 7200E-03 11906
//E23 1. 7107E-03 6749 -1. 4474E-03 13509
//E13 1. 2130E-03 12475 -1. 1219E-03 11342
//S11 2. 7825E+01 1086 -1. 9473E+01 2363
//S22 3. 7931E+01 13972 -1. 4575E+01 2367
//S33 2. 7377E+01 1086 -1. 9776E+01 13082
//S12 1. 6847E+01 2698 -1. 7201E+01 11906
//S23 1. 7109E+01 6749 -1. 4474E+01 13509
//S13 1. 2124E+01 12475 -1. 1214E+01 11342
//SMS 3. 7533E+01 2834 2. 7585E-04 7333

```

4.7 Static Analysis (Viscoelasticity)

Data of tutorial/ 07_viscoelastic_cylinder / is used to implement this analysis.

4.7.1 Analysis Object

The same 1/8 model cylinder as in the static analysis (hyperelasticity part 1) in Section 4.3 is the object of the analysis.

4.7.2 Analysis Content

Stress relaxation analysis is implemented where tension displacement is applied to the cylinder in the axial direction. The analysis control data is shown in the following.

```
# Control File for FISTR
## Analysis Control
!VERSION
3
!SOLUTION, TYPE=NLSTATIC
!WRITE, VISUAL
!WRITE, RESULT
## 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, 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, 2
1.0e-8, 1.0, 0.0
```

4.7.3 Analysis Results

A deformed figure applied with a contour of the Mises stress was created by REVOCAP_PrePost, and is shown in Figure 4.7.1. This is the analysis results after 2 seconds (10th step). Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

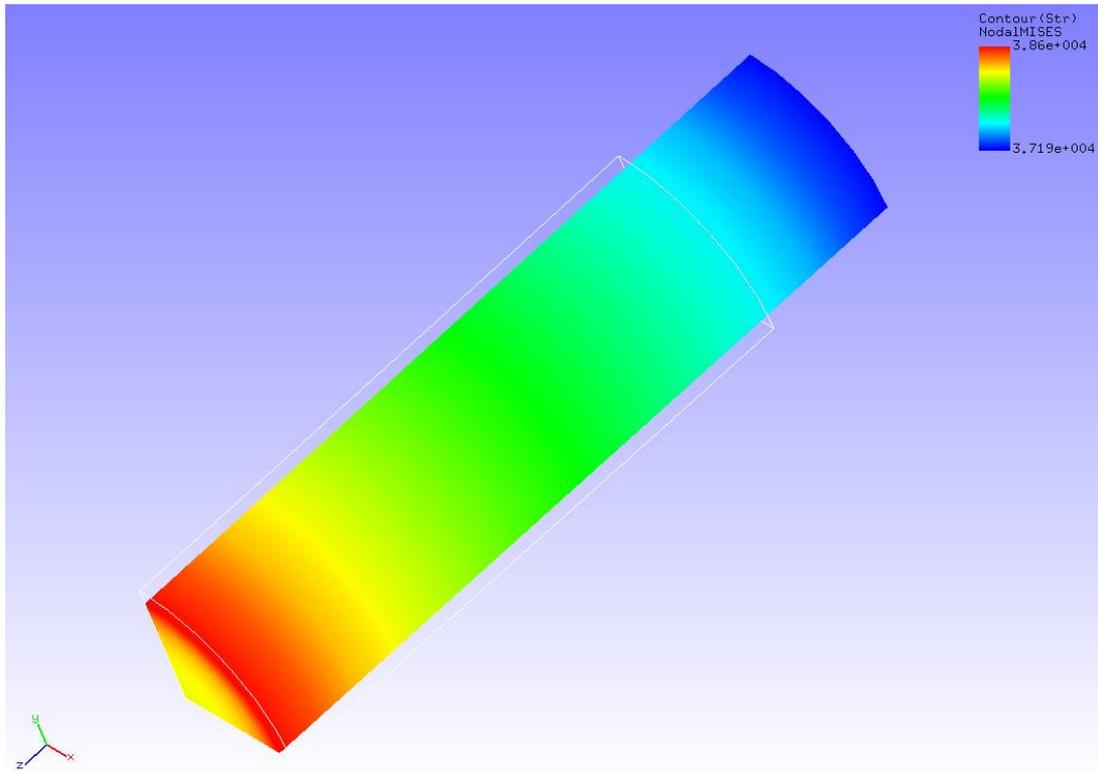


Figure 4.7.1: Analysis Results of Deformation and Mises Stress

```

#### Result step= 10
##### Local Summary :Max/IdMax/Min/IdMin#####
//U1 0.0000E+00      1 -7.4531E-01      91
//U2 0.0000E+00      1 -7.4531E-01      88
//U3 0.0000E+00      1 -7.0000E+00      38
//E11 -1.0763E-01      38 -1.1244E-01       7
//E22 -1.0763E-01      50 -1.1244E-01      13
//E33 3.0270E-01      13 2.9129E-01       50
//E12 9.8113E-04      53 -9.9997E-04       10
//E23 1.1878E-04      72 -3.2869E-03       84
//E13 1.1878E-04      64 -3.2869E-03       95
//S11 1.4135E+02      13 -1.3699E+02       50
//S22 1.4135E+02       7 -1.3699E+02       38
//S33 3.8691E+04      13 3.7107E+04       50
//S12 4.6701E+01      53 -4.7594E+01       10
//S23 5.2254E+00      72 -1.5313E+02       84
//S13 5.2254E+00      64 -1.5313E+02       95
//SMS 3.8602E+04      13 3.7194E+04       50

```

4.8 Static Analysis (Creep)

Data of tutorial/ 08_creep_cylinder / is used to implement this analysis.

4.8.1 Analysis Object

The same 1/8 model cylinder as in the static analysis (hyperelasticity part 1) in Section 4.3 is the object of the analysis.

4.8.2 Analysis Content

Creep behavioral analysis is implemented where tension displacement is applied to the cylinder in the axial direction. The analysis control data is shown in the following.

```
# 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, 2
1.0e-8, 1.0, 0.0
```

4.8.3 Analysis Results

As analysis results of the 5th sub step, a deformed figure applied with a contour of the Mises stress was created by REVOCAP_PrePost, and is shown in Figure 4.8.1. Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

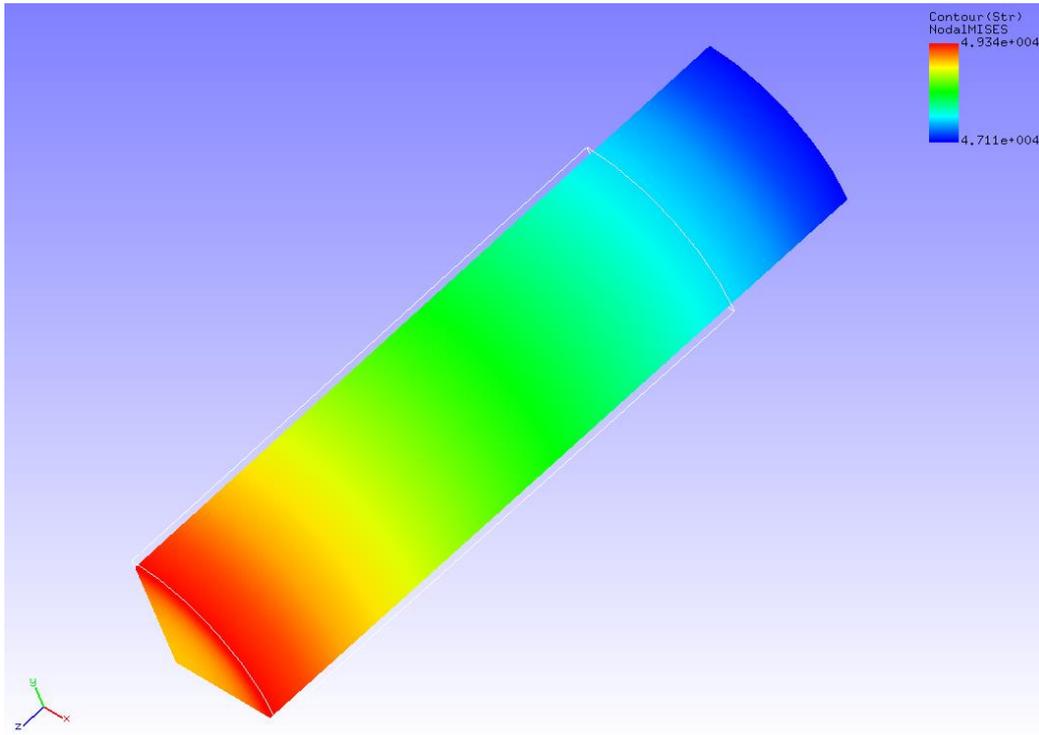


Figure 4.8.1: Analysis Results of Deformation and Mises Stress

```

#### Result step=      5
##### Local Summary :Max/IdMax/Min/IdMin#####
//U1  0.0000E+00      1 -4.1832E-01      91
//U2  0.0000E+00      1 -4.1832E-01      88
//U3  0.0000E+00      1 -7.0000E+00      38
//E11 -6.5815E-02      38 -6.9387E-02      7
//E22 -6.5815E-02      50 -6.9387E-02      13
//E33  2.3854E-01      13  2.2765E-01      38
//E12  5.4317E-04      53 -5.5746E-04      10
//E23  8.9875E-05      72 -2.2085E-03      84
//E13  8.9875E-05      64 -2.2085E-03      95
//S11  1.1317E+02      14 -1.1102E+02      49
//S22  1.1317E+02      6  -1.1102E+02      39
//S33  4.9374E+04      13  4.7081E+04      38
//S12  4.3566E+01      53 -4.4697E+01      10
//S23  7.6408E+00      72 -1.6768E+02      84
//S13  7.6408E+00      64 -1.6768E+02      95
//SMS  4.9340E+04      13  4.7114E+04      38

```

4.9 Contact Analysis (Part 1)

Data of tutorial/ 09_contact_hertz / is used to implement this analysis.

4.9.1 Analysis Object

The Hertz contact problem was applied in this analysis. The shape of the analysis object is shown in Figure 4.9.1, and the mesh data is shown in Figure 4.9.2. Hexahedral linear elements are used for the mesh, and the scale of the mesh consists of 168 elements and 408 nodes.

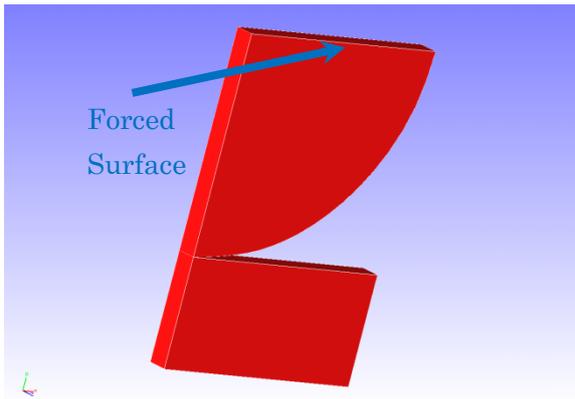


Figure 4.9.1: Shape of Analysis Object

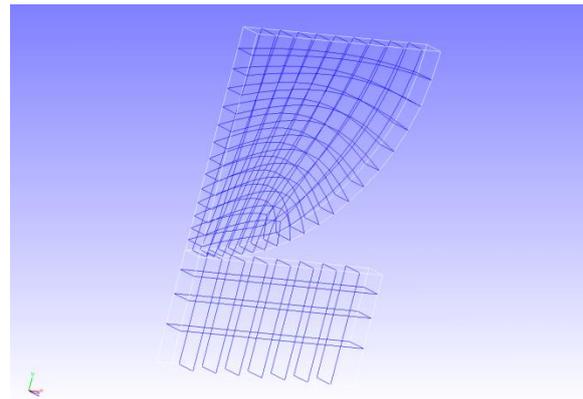


Figure 4.9.2: Mesh Data of Analysis Object

4.9.2 Analysis Content

An extended Lagrange multiplier method is used to implement the contact analysis where forced displacement is applied to the upper surface of a 1/4 model disc in the compression direction. The analysis control data is shown in the following.

```
# Control File for FISTR
## Analysis Control
!VERSION
  3
!SOLUTION, TYPE=NLSTATIC
!WRITE, RESULT
!WRITE, VISUAL
## Solver Control
### Boundary Conditon
!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=ALAGRANGE
!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 Setting
!SOLVER, METHOD=CG, PRECOND=1, ITERLOG=YES, TIMELOG=YES
1000 2

```

4.9.3 Analysis Results

As analysis results of the 5th sub step, a deformed figure applied with a contour of the displacement in the y direction was created by REVOCAP_PrePost, and is shown in Figure 4.9.3. Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

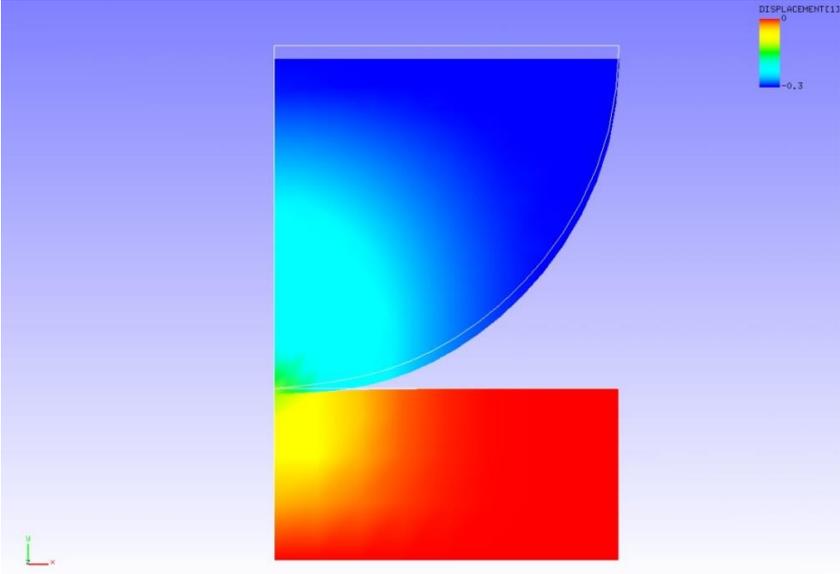


Figure 4.9.3: Analysis Results of Deformation and y Direction Displacement

```

#### Result step=      5
##### Local Summary :Max/IdMax/Min/IdMin#####
//U1  1.1912E-02      70 -3.7167E-02      47
//U2  4.4886E-03     1008 -3.0603E-01     32
//U3  0.0000E+00      1  0.0000E+00      1
//E11 6.4120E-03     1003 -4.1075E-02     50
//E22 1.8765E-03     1012 -5.8752E-02     29
//E33 1.1012E-02     1046 -3.2153E-03     47
//E12 4.9036E-02     1046 -3.9706E-02     30
//E23 1.4957E-15     1047 -8.5554E-15    1000
//E13 7.5696E-15      50 -1.4571E-15    1047
//S11 7.0532E+00     1003 -4.5183E+01     50
//S22 2.0641E+00     1012 -6.4627E+01     29
//S33 1.2113E+01     1046 -3.5369E+00     47
//S12 2.6970E+01     1046 -2.1838E+01     30
//S23 8.2263E-13     1047 -4.7055E-12    1000
//S13 4.1633E-12      50 -8.0141E-13    1047
//SMS 7.6836E+01      30  8.8599E-02      69

```

4.10 Contact Analysis (Part 2)

Data of tutorial/ 10_contact_2tubes/ is used to implement this analysis.

4.10.1 Analysis Object

A pinched cylindrical problem was applied in this analysis. The shape of the analysis object is shown in Figure 4.10.1, and the mesh data is shown in Figure 4.10.2. Hexahedral linear elements are used for the mesh, and the scale of the mesh consists of 2,888 elements and 4,000 nodes.

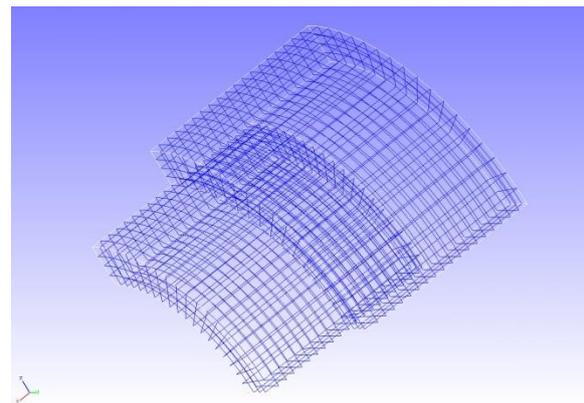
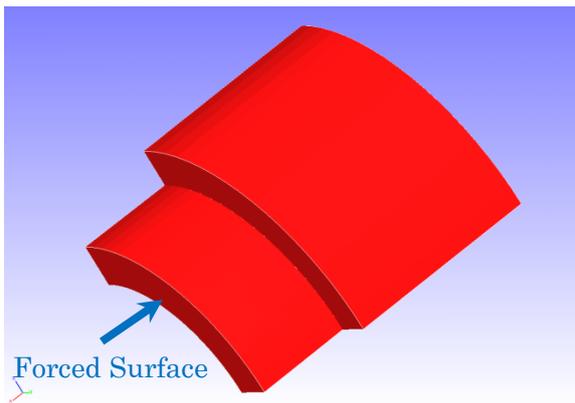


Figure 4.10.1: Shape of Analysis Object

Figure 4.10.2: Mesh Data of Analysis Object

4.10.2 Analysis Content

The Lagrange multiplier method is used to implement the contact analysis where forced displacement is applied to the forced surface shown in Figure 4.10.1 in the pinched direction. The analysis control data is shown in the following.

```
# Control File for FISTR
## Analysis Control
!VERSION
  3
!SOLUTION, TYPE=NLSTATIC
!WRITE, RESULT
!WRITE, VISUAL
## Solver Control
### Boundary Conditon
!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=DIRECTmkI

```

4.10.3 Analysis Results

As analysis results of the 4th sub step, a deformed figure applied with a contour of the Mises stress was created by REVOCAP_PrePost, and is shown in Figure 4.10.3. Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

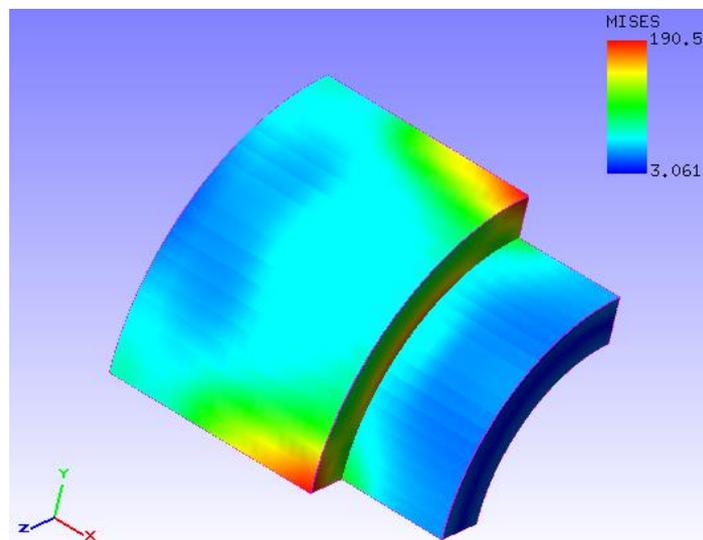


Figure 4.10.3: Analysis Results of Deformation and Mises Stress

```

#### Result step=      4
##### Local Summary :Max/IdMax/Min/IdMin####
//U1  8.6939E-04      32 -1.0021E+00      2006
//U2  8.7641E-03     104 -7.0519E-03      2006
//U3  8.7641E-03      4  -7.0519E-03      1901
//E11 7.5301E-04     1901 -4.1253E-04      105
//E22 9.8422E-04      2  -9.2887E-04     2058
//E33 9.8423E-04     102 -9.2880E-04     3843
//E12 5.3508E-04     133 -2.8307E-04      278
//E23 1.2482E-03     1901 -1.4180E-03      4
//E13 5.3519E-04      33 -2.8312E-04     1678
//S11 7.7141E+01     103 -9.0007E+01      101
//S22 2.0117E+02      2  -2.2938E+02     1905
//S33 2.0117E+02     102 -2.2941E+02     2010
//S12 4.3218E+01     133 -2.2863E+01      278
//S23 1.0082E+02     1901 -1.1453E+02      4
//S13 4.3227E+01      33 -2.2867E+01     1678
//SMS 2.9968E+02     1901 3.1610E+00      2454

```

4.11 Contact Analysis (Part 3)

Data of tutorial/ 11_contact_2beam/ is used to implement this analysis.

4.11.1 Analysis Object

A two beam contact problem is applied in this analysis. The outline of the analysis model is shown in Figure 4.11.1. Hexahedral linear elements are used for the mesh, and the scale of the mesh consists of 80 elements and 252 nodes.

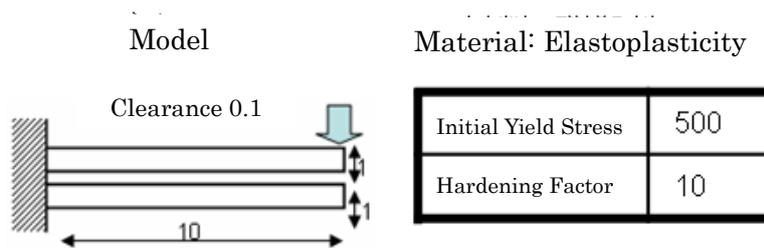


Figure 4.11.1 Outline of Analysis Model

4.11.2 Analysis Contents

The Lagrange multiplier method is used to implement the contact analysis where forced displacement is applied to the front edge surface of the upper beam. The analysis control data is shown in the following.

```
!!  
!! Control File for FISTR  
!!  
!VERSION  
3  
!SOLUTION, TYPE=NLSTATIC  
!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  
BOUNDARY, 1  
CONTACT, 1  
!MATERIAL, NAME=M1  
!ELASTIC  
2.1e+5, 0.3  
!PLASTIC, YIELD=MISES  
500.0, 10.0  
!SOLVER, METHOD=MUMPS
```

4.11.3 Analysis Results

As analysis results of the 100th sub step, a deformed figure applied with a contour of the Mises stress was created by REVOCAP_PrePost, and is shown in Figure 4.11.2. Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

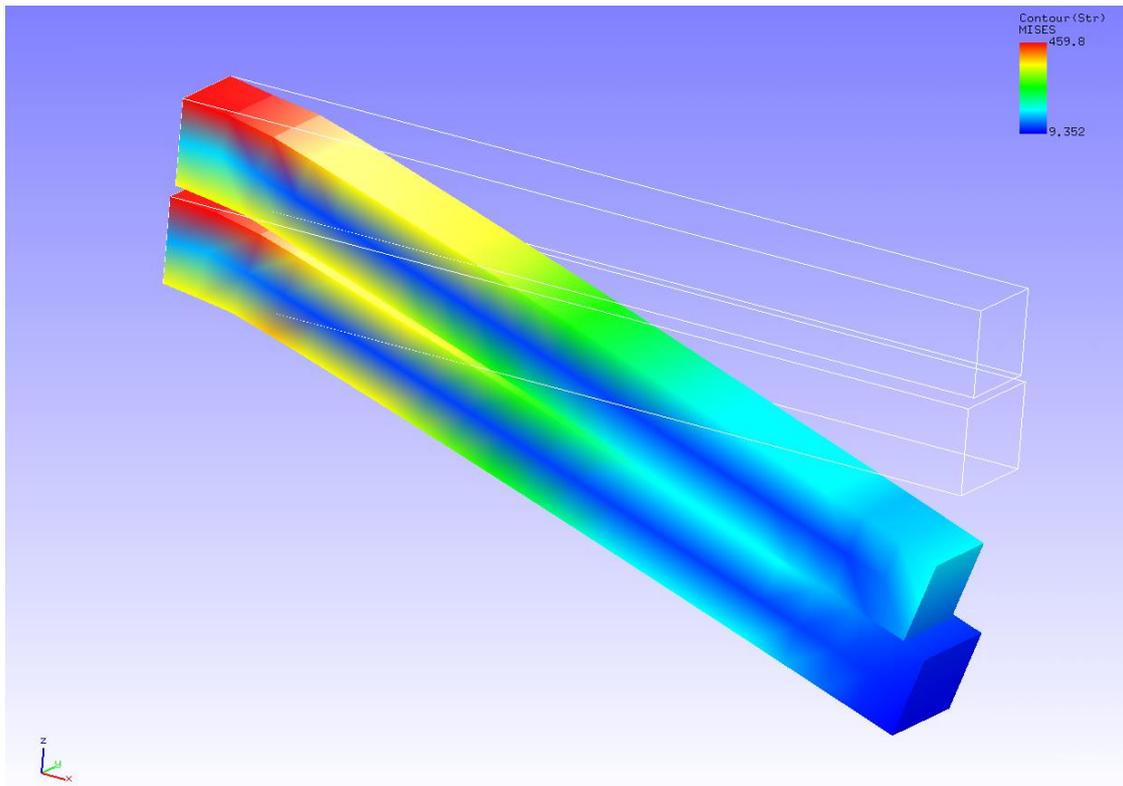


Figure 4.11.2: Analysis Results of Deformation and Mises Stress

```

#### Result step= 100
##### Local Summary :Max/IdMax/Min/IdMin#####
//U1 1.4102E-01      196 -6.1103E-01      6
//U2 4.5722E-02      11 -4.5722E-02      195
//U3 0.0000E+00      1 -3.0000E+00      8
//E11 1.6030E-01      195 -1.3024E-01      49
//E22 5.9705E-02      49 -7.5459E-02      195
//E33 7.3924E-02      152 -8.7395E-02      30
//E12 8.6186E-02      7 -8.6186E-02      192
//E23 9.9009E-02      11 -9.9009E-02      195
//E13 6.0657E-02      90 -1.2889E-01      192
//S11 5.7685E+02      132 -6.3641E+02      152
//S22 1.2740E+02      3 -1.2727E+02      10
//S33 1.4933E+02      3 -1.4146E+02      127
//S12 1.4676E+02      70 -1.4676E+02      235
//S23 1.7885E+02      109 -1.7885E+02      172
//S13 1.6202E+02      90 -2.4814E+02      194
//SMS 6.2476E+02      89 8.3117E+00      2

```

4.12 Linear Dynamic Analysis

Data of tutorial/ 12_dynamic_beam/ is used to implement this analysis.

4.12.1 Analysis Object

A cantilever beam is the object of the analysis. The shape is shown in Figure 4.12.1, and the mesh data is shown in Figure 4.12.2. Quadratic tetrahedral elements are used for the mesh, and the scale of the mesh consists of 240 elements and 525 nodes.

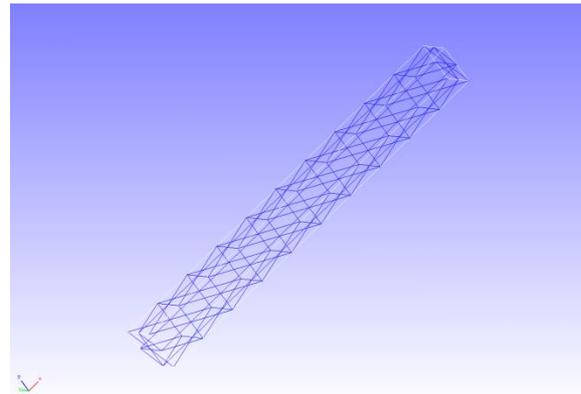
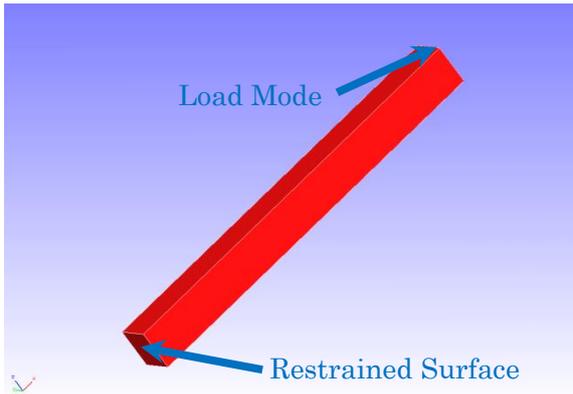


Figure 4.12.1: Shape of Cantilever Beam

Figure 4.12.2: Mesh Data of Cantilever Beam

4.12.2 Analysis Contents

A linear dynamic analysis is implemented, after the displacement of the restrained surface shown in Figure 4.12.1 is restrained, and a concentrated load is applied to the load node. The analysis control data is shown in the following.

```
# 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 Conditon
!BOUNDARY, AMP=AMP1
  FIX, 1, 3, 0.0
!CLOAD, AMP=AMP1
  CL1, 3, -1.0
### Solver Setting
!SOLVER, METHOD=CG, PRECOND=1, ITERLOG=NO, TIMELOG=NO
  10000, 2
  1.0e-06, 1.0, 0.0
```

4.12.3 Analysis Results

A time sequence display of the displacement of the monitoring node (load node, node number 3121) specified by the analysis control data was created in Microsoft Excel, and is shown in Figure 4.12.3. A portion of the displacement output file (dyna_disp_p1.out) of the monitoring node is shown in the following as numeric data of the analysis results.

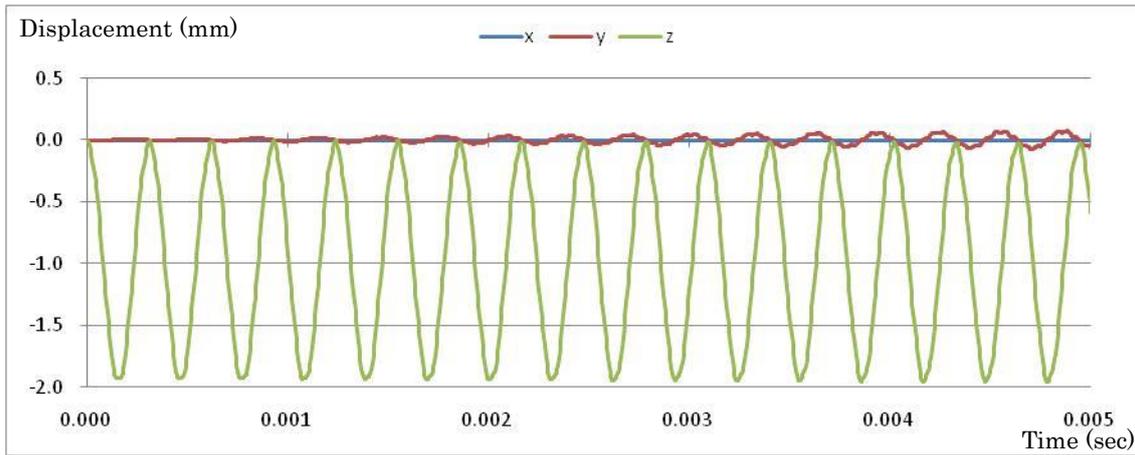


Figure 4.12.3: Displacement Time Sequence of Monitoring Node

0	0.0000E+000	3121	0.0000E+000	0.0000E+000	0.0000E+000
500	5.0000E-006	3121	5.5959E-005	-2.0679E-006	-1.5563E-002
1000	1.0000E-005	3121	5.3913E-005	2.0947E-005	-4.3950E-002
1500	1.5000E-005	3121	7.6105E-005	5.8799E-005	-8.0795E-002
2000	2.0000E-005	3121	6.8543E-006	4.0956E-005	-1.2329E-001
2500	2.5000E-005	3121	5.4725E-005	7.0881E-005	-1.7742E-001
3000	3.0000E-005	3121	6.8226E-005	1.7597E-004	-2.2801E-001
3500	3.5000E-005	3121	4.2923E-005	1.1791E-004	-2.7290E-001
4000	4.0000E-005	3121	-1.2087E-005	1.2552E-004	-3.2393E-001
4500	4.5000E-005	3121	3.4969E-005	-3.4512E-005	-3.8844E-001
5000	5.0000E-005	3121	6.1592E-005	1.2820E-004	-4.6425E-001
5500	5.5000E-005	3121	1.3188E-005	1.9002E-005	-5.4590E-001
6000	6.0000E-005	3121	3.1393E-005	-7.4604E-005	-6.4556E-001
6500	6.5000E-005	3121	9.8931E-005	-1.9078E-004	-7.5561E-001
7000	7.0000E-005	3121	4.2308E-005	1.1593E-004	-8.6826E-001
7500	7.5000E-005	3121	-2.7019E-005	3.0277E-004	-9.6826E-001

4.13 Nonlinear Dynamic Analysis

Data of tutorial/ 13_dynamic_beam_nonlinear / is used to implement this analysis.

4.13.1 Analysis Object

The same cantilever beam as in the linear dynamic analysis in Section 4.12 is the object of the analysis.

4.13.2 Analysis Content

A nonlinear dynamic analysis is implemented, after the displacement of the restrained surface shown in Figure 4.12.1 is restrained, and a concentrated load is applied to the load node. The analysis control data is shown in the following.

```
# 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 Conditon
!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, 2
  1.0e-06, 1.0, 0.0
```

4.13.3 Analysis Results

A time sequence display of the displacement of the monitoring node (load node, node number 3121) specified by the analysis control data was created in Microsoft Excel, and is

shown in Figure 4.13.1. A portion of the displacement output file (dyna_disp_p1.out) of the monitoring node is shown in the following as numeric data of the analysis results.

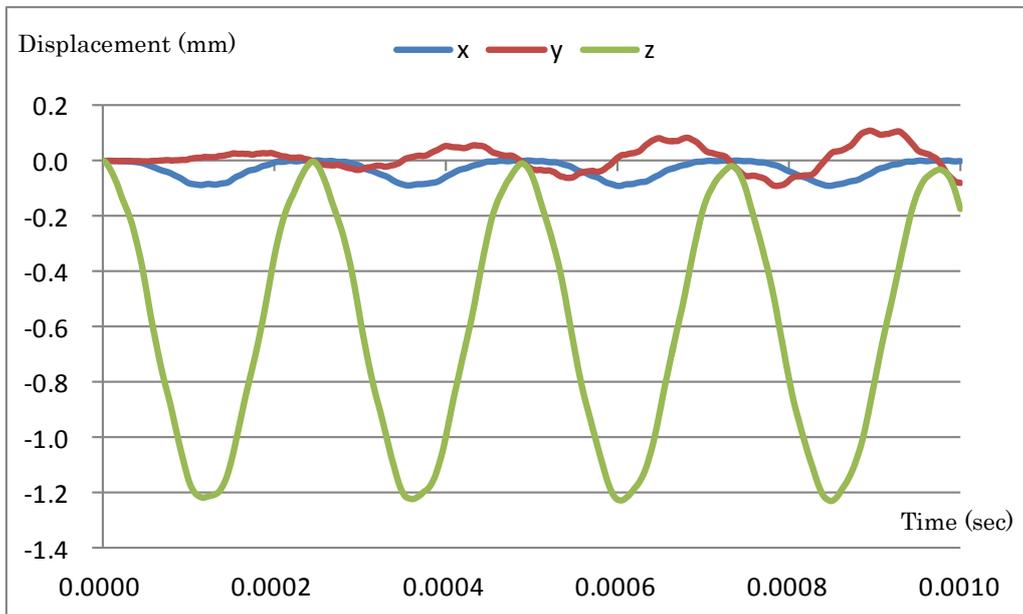


Figure 4.13.1: Displacement Time Sequence of Monitoring Node

0	0.0000E+000	3121	0.0000E+000	0.0000E+000	0.0000E+000
100	1.0000E-006	3121	9.6353E-005	-5.1095E-005	-1.3238E-003
200	2.0000E-006	3121	9.0012E-005	-3.6471E-005	-3.6634E-003
300	3.0000E-006	3121	8.9091E-005	-5.9391E-007	-6.5111E-003
400	4.0000E-006	3121	1.8224E-005	1.8912E-006	-1.0037E-002
500	5.0000E-006	3121	5.1827E-005	-2.5069E-005	-1.4121E-002
600	6.0000E-006	3121	3.6671E-005	2.1807E-005	-1.8473E-002
700	7.0000E-006	3121	-1.7546E-005	6.9216E-006	-2.3308E-002
800	8.0000E-006	3121	-5.2440E-005	1.6820E-006	-2.8491E-002
900	9.0000E-006	3121	-8.5845E-005	3.4707E-005	-3.4008E-002
1000	1.0000E-005	3121	-1.4183E-004	3.5653E-005	-3.9828E-002
1100	1.1000E-005	3121	-2.0256E-004	1.7437E-005	-4.5995E-002
1200	1.2000E-005	3121	-2.3574E-004	3.3228E-005	-5.2387E-002
1300	1.3000E-005	3121	-3.3244E-004	2.3837E-005	-5.9080E-002
1400	1.4000E-005	3121	-4.3976E-004	4.6942E-005	-6.6266E-002
1500	1.5000E-005	3121	-5.2678E-004	1.6307E-004	-7.3148E-002

4.14 Nonlinear Contact Dynamic Analysis

Data of tutorial/ 14_dynamic_plate_contact/ is used to implement this analysis.

4.14.1 Analysis Object

A drop impact analysis of a square plate on a floor is the subject of the analysis. The shape is shown in Figure 4.14.1, and the mesh data is shown in Figure 4.14.2. Hexahedral linear elements are used for the mesh, and the scale of the mesh consists of 8,232 elements and 10,712 nodes.

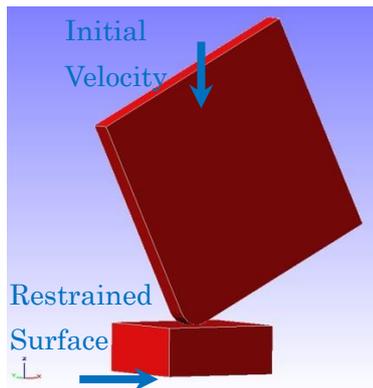


Figure 4.14.1: Shape of Floor and Square Plate

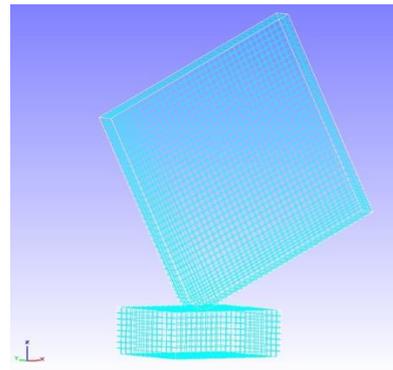


Figure 4.14.2: Mesh Data of Floor and Square Plate

4.14.2 Analysis Content

An initial velocity of 4,427 mm/s is set for the square plate of the analysis object, to implement the contact dynamic analysis. The analysis control data is shown in the following.

```
!! Control File for FISTR
!VERSION
3
!WRITE, LOG, FREQUENCY=20
!WRITE, RESULT, FREQUENCY=20
!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
```

4.14.3 Analysis Results

The contour figure of the Mises stress at the time of the drop impact is shown in Figure 4.14.3. A portion of the energy output file (dyna_energy.txt) of the monitoring node is shown in the following as numeric data of the analysis results.

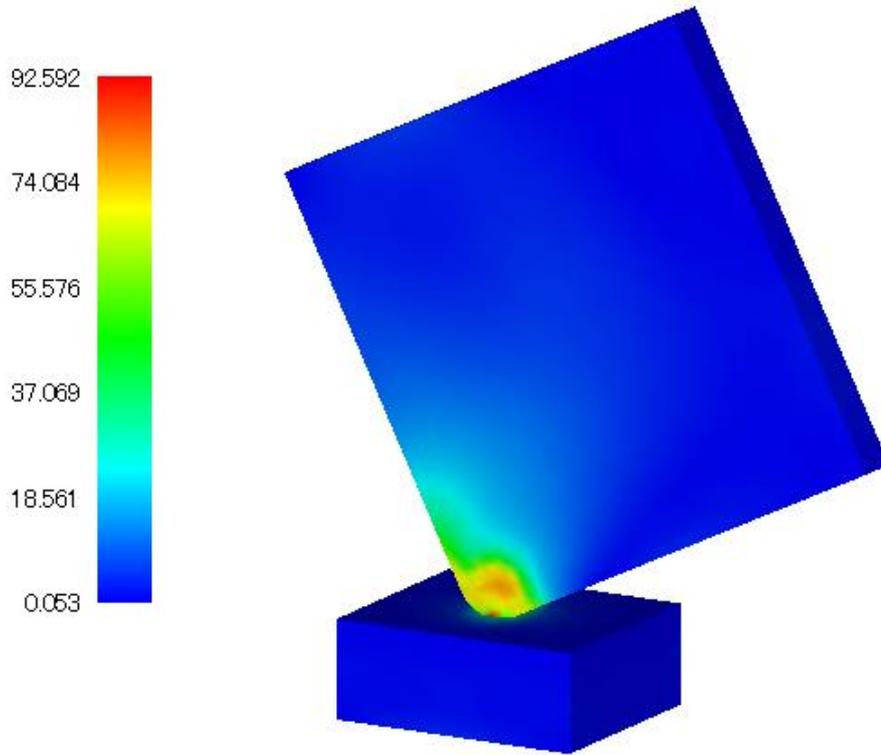


Figure 4.14.3: Mises Stress at time of Drop Impact

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	4.9520E-006	9.7806E-003
2	2.0000E-008	9.7653E-003	1.4640E-005	9.7800E-003
3	3.0000E-008	9.7535E-003	2.5204E-005	9.7787E-003
4	4.0000E-008	9.7408E-003	3.7426E-005	9.7782E-003
5	5.0000E-008	9.7278E-003	5.0061E-005	9.7779E-003
6	6.0000E-008	9.7147E-003	6.2937E-005	9.7776E-003
7	7.0000E-008	9.7015E-003	7.5913E-005	9.7774E-003
8	8.0000E-008	9.6883E-003	8.8933E-005	9.7772E-003
9	9.0000E-008	9.6751E-003	1.0199E-004	9.7771E-003
10	1.0000E-007	9.6619E-003	1.1508E-004	9.7769E-003
11	1.1000E-007	9.6486E-003	1.2823E-004	9.7768E-003
12	1.2000E-007	9.6353E-003	1.4139E-004	9.7767E-003

4.15 Eigenvalue Analysis

Data of tutorial/ 15_eigen_spring/ is used to implement this analysis.

4.15.1 Analysis Object

The same spring as in the static analysis (hyperelasticity part 2) in Section 4.4 is the object of the analysis.

4.15.2 Analysis Content

The displacement of the restrained surface shown in Figure 4.4.1 is restrained, and an eigenvalue analysis is implemented up to the 5th mode. The analysis control data is shown in the following.

```
# 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
```

4.15.3 Analysis Results

Analysis results data file spring.res.0.3 is used, and a 3rd oscillation mode (compression extension of the spring in y direction) was created by REVOCAP_PrePost, and is shown in Figure 4.15.1. The deformation magnification is set to 1,000. The character frequency list output to the analysis results log file is shown in the following as numeric data of the analysis results.

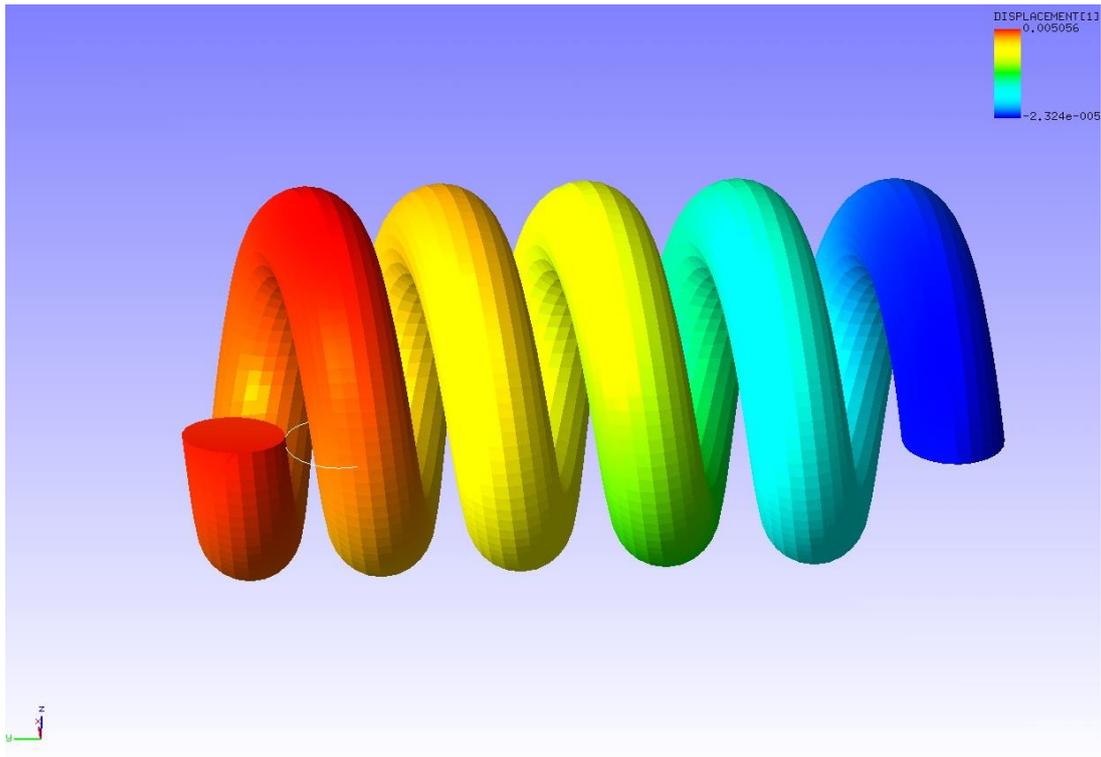


Figure 4.15.1: 3rd Oscillation Mode of Spring

```

*****
*RESULT OF EIGEN VALUE ANALYSIS*
*****
NUMBER OF ITERATIONS =          26

NO.    EIGENVALUE    ANGL. FREQUENCY    FREQUENCY (HZ)
-----
1      0.783085E+07    0.279837E+04      0.445374E+03
2      0.787176E+07    0.280567E+04      0.446536E+03
3      0.326006E+08    0.570969E+04      0.908726E+03
4      0.383712E+08    0.619445E+04      0.985877E+03
5      0.129322E+09    0.113720E+05      0.180991E+04

```

4.16 Heat Conduction Analysis

Data of tutorial/ 16_heat_block/ is used to implement this analysis.

4.16.1 Analysis Object

A block with a hole is the object of the analysis. The shape is shown in Figure 4.16.1, and the mesh data is shown in Figure 4.16.2. Hexahedral linear elements are used for the mesh, and the scale of the mesh consists of 32,160 elements and 37,386 nodes.

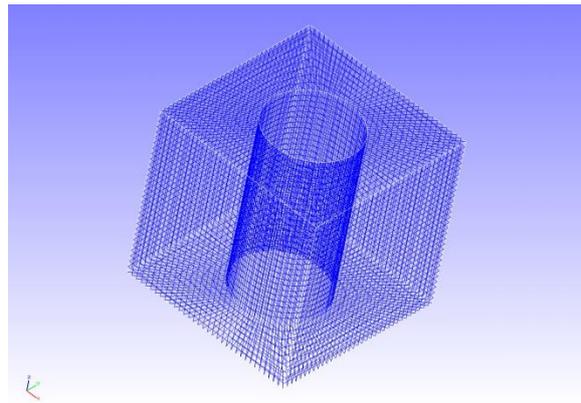
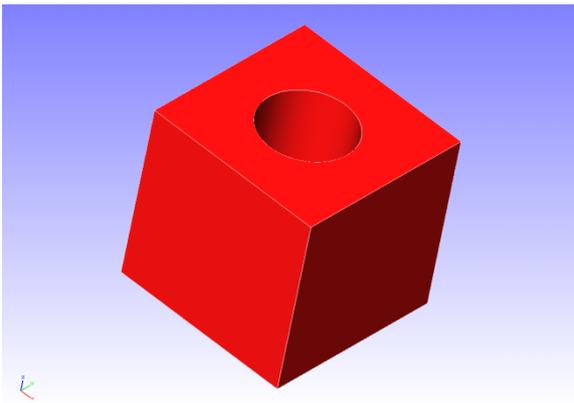


Figure 4.16.1: Shape of Block with Hole

Figure 4.16.2: Mesh Data of Block with Hole

4.16.2 Analysis Content

A steady heat conduction analysis is implemented, where a heat source is applied to the cylindrical inner surface of the analysis object. The analysis control data is shown in the following.

```
# Control File for FISTR
## Analysis Control
!VERSION
  3
!SOLUTION, TYPE=HEAT
!HEAT
  0.0
!WRITE, RESULT
!WRITE, VISUAL
## Solver Control
### Boundary Conditon
!FIXTEMP
  FTMPC, 100.0
  FTMPS1, 20.0
  FTMPS2, 20.0
  FTMPS3, 20.0
  FTMPS4, 20.0
### Solver Setting
!SOLVER, METHOD=CG, PRECOND=2, ITERLOG=YES, TIMELOG=YES
  100, 2
  1.0e-8, 1.0, 0.0
```

4.16.3 Analysis Results

A temperature contour figure was created by REVOCAP_PrePost, and is shown in Figure 4.16.3. Moreover, a portion of the analysis results log file is shown in the following as numeric data of the analysis results.

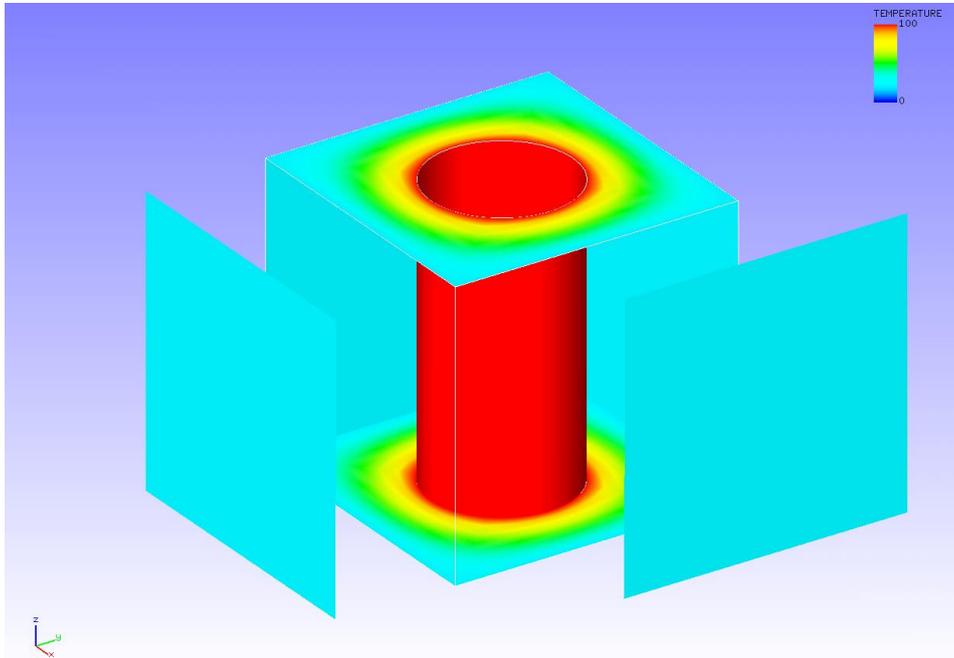


Figure 4.16.3: Temperature Analysis Results

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

4.17 Frequency Response Analysis

Use the files in the directory, tutorial/ 17_freq_beam/, in order to reproduce the test. The analysis consists of two steps; 1st Eigenvalue analysis, 2nd Frequency response analysis. For the 1st step, the following name change is required.

```
hecmw_ctrl_eigen.dat -> hecmw_ctrl.dat
```

After changing the name of file, eigenvalues analysis should be executed. You will get 0.log as the result of eigenvalue analysis. The file name should be changed as follows.

```
0.log -> eigen_0.log
```

Then start frequency response analysis.

4.17.1 Analysis Object

The analysis model is shown in Fig. 4.17.1 and the discretized mesh is shown in Fig.4.17.2. The model is mesh with Element Type 341 (Number of Elements: 126, Number of Nodes: 55).

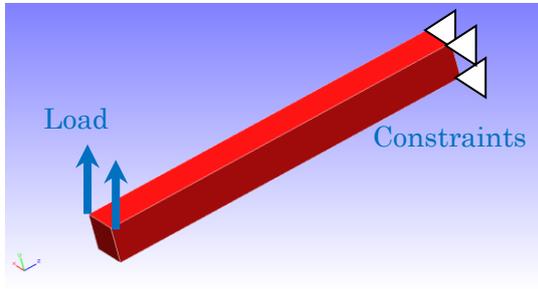


Fig. 4.17.1 The analysis model

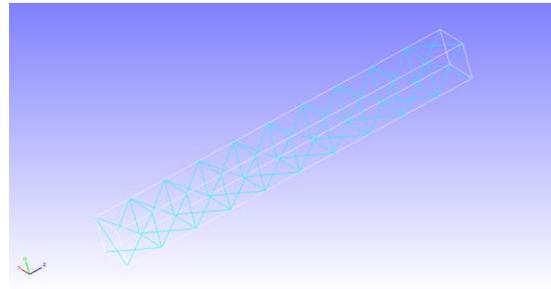


Fig. 4.17.2 The mesh

4.17.2 Analysis Content

One of the cantilever beam end was fixed and the other is applied load as nodal force on two nodes. The eigenvalues up to 10th mode are computed and the resulting eigenvalues and eigenvectors up to 5th mode are used for frequency response analysis. The analysis control data shown below.

```
# 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, 2
1.0e-8, 1.0, 0.0
```

4.17.3 Analysis Results

The frequency dependency of amplitude of displacement at a monitoring node (Node ID 1) specified in the analysis control data is shown in Fig. 4.17.3. A portion of a log file is shown below to show the numerical data obtained by the frequency response analysis.

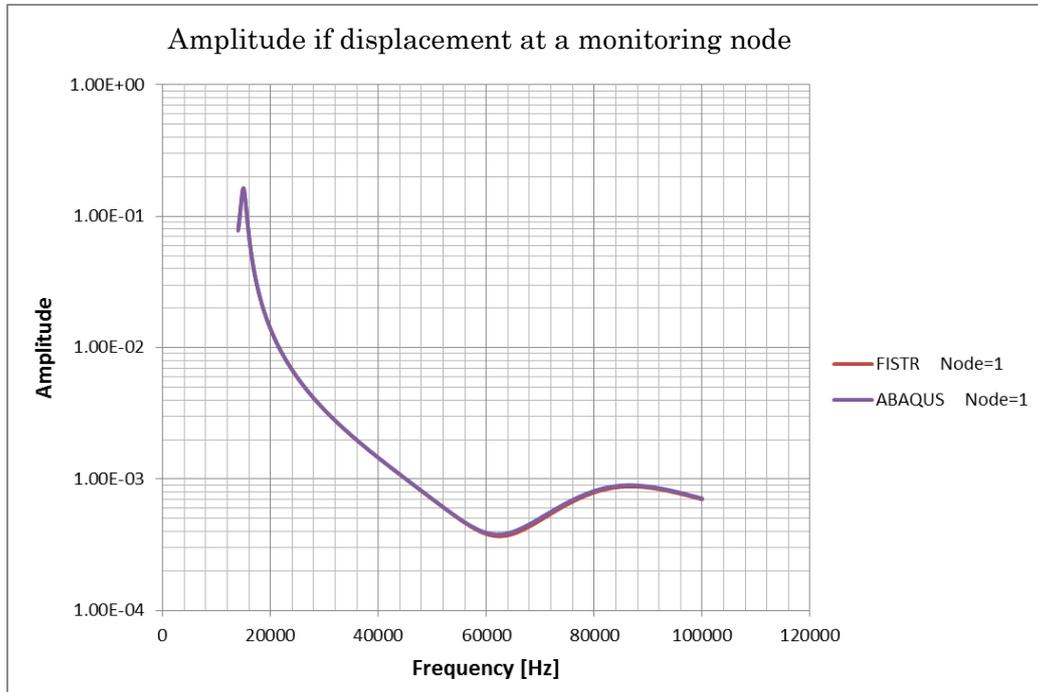


Fig.4.17.3 Frequency dependency of amplitude of displacement at a monitoring node

```

Rayleigh alpha: 0.000000000000000E+000
Rayleigh beta: 7.200000000000000E-007
read from=eigen_0.log
start mode=      1
end mode=       5
start frequency: 14000.0000000000
end frequency:  16000.0000000000
number of the sampling points      20
monitor nodeid=      1
14100.0000000000    [Hz] : 8.395286141741409E-002
14100.0000000000    [Hz] : 1 .res
14200.0000000000    [Hz] : 9.123156781733653E-002
14200.0000000000    [Hz] : 2 .res
14300.0000000000    [Hz] : 9.960390920903195E-002
14300.0000000000    [Hz] : 3 .res

```

End