

FrontISTR Analysis Flow Manual

FrontISTR Commons

August 19, 2019

Contents

1	FrontISTR Analysis Flow Manual	2
1.1	Manuals	2
1.2	List of description on this manual	2
1.3	Analysis Flow and Input/Output File	3
1.3.1	Analysis Flow	3
1.3.2	Overall Control Data	4
1.3.3	Mesh Data	4
1.3.4	Analysis Control Data	4
1.3.5	Output File	5
1.3.6	Execution Procedure	5
1.4	Element Library and Material Data	6
1.4.1	Element Library	6
1.4.2	Material Data	10
1.4.3	Nonlinear Static Analysis	14
1.5	Step control	15
1.5.1	About time on analysis	15
1.5.2	Control for static analysis	15
1.5.3	Control for dynamic analysis	17
1.6	Overall Control Data	17
1.6.1	Outline of Overall Control Data	17
1.6.2	Input Rules	18
1.6.3	Header List	18
1.7	Single Domain Mesh Data	22
1.7.1	Outline of Single Mesh Data	22
1.7.2	Input Rules	22
1.7.3	Header List of Single Domain Mesh Data	23
1.8	Analysis Control Data	41
1.8.1	Outline of Analysis Control Data	41
1.8.2	Input Rules	42
1.8.3	Analysis Control Data	43
1.8.4	Solver Control Data	54
1.8.5	Post Process (Visualization) Control Data	55
1.8.6	Details of Analysis Control Data Parameters	65
1.9	User Subroutines	103
1.9.1	Input of User Defined Material	104
1.9.2	Subroutine regarding Elastoplasticity Deformation (<i>uyield.f90</i>)	104
1.9.3	Subroutine regarding Elastic Deformation (<i>uelastic.f90</i>)	104
1.9.4	Subroutine regarding User Defined Materials (<i>umat.f</i>)	105
1.9.5	Process Subroutine of User Defined External Load (<i>uload.f</i>)	106

1 FrontISTR Analysis Flow Manual

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	5.0
License	MIT License
Corresponding Clerks	FrontISTR Commons2-11-16 Yayoi, Bunkyo-ku, Tokyo/o Institute of Engineering Innovation, School of EngineeringE-mailsupport@frontistr.com

1.1 Manuals

- Introduction
- How to install
- Theory
- User’s manual
- Tutorial

This manual describes the analysis method by the finite element method (FEM) used in FrontISTR.

Regarding the stress analysis method of solids, the infinitesimal deformation linear elasticity static analysis method is described first, and geometric nonlinear analysis method and elastoplasticity analysis method which are required when handling finite deformation problems are described next. Furthermore, a summarized evaluation method of the fracture mechanics parameters which can be acquired using the results of the stress analysis by FEM is described. Finally, the eigenvalue analysis and heat conduction analysis method is described.

1.2 List of description on this manual

- PDF
- Analysis Flow
- Input File
 - Overall Control Data
 - Mesh Partitioning Data
 - Mesh Data
 - Analysis Control Data
 - Visualization Control Data
- Output File
 - Log Data
 - Restart Data
 - Visualization Data

- User Subroutines

1.3 Analysis Flow and Input/Output File

1.3.1 Analysis Flow

The input and output file of the structural analysis code FrontISTR is shown in Figure 3.1.1.

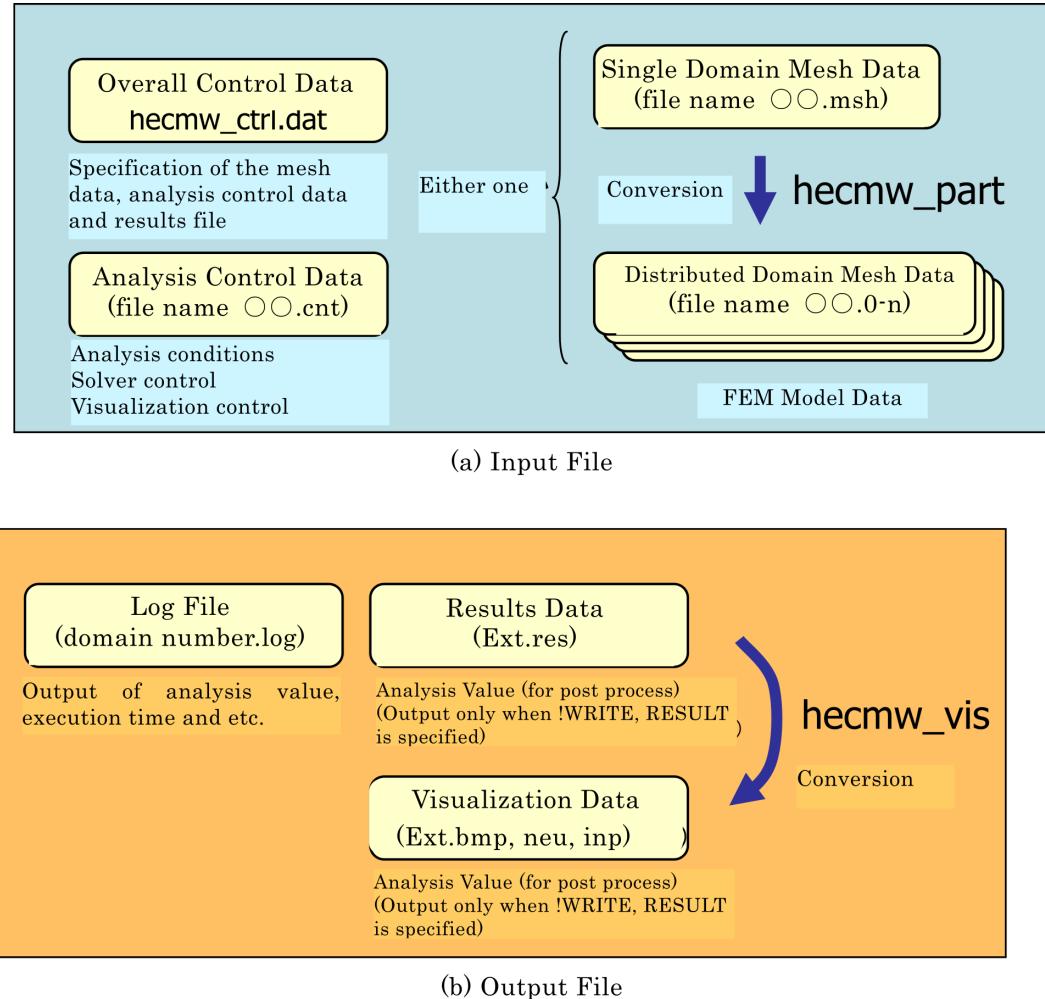


Figure 1: FrontISTR Input/Output File

Figure 3.1.1:FrontISTR Input/Output File

FrontISTR requires three files, such as the overall control data, mesh data and analysis control data as input files.

When analyzing the overall model with a single CPU, the single domain mesh file is used. When performing parallel execution with multiple CPUs, the distributed domain mesh data as a result of performing domain partitioning of the single domain mesh data in advance by the hecmw_part program which is the partitioner of *HEC-MW* is used.

The overall control data, analysis control data and single domain mesh data are text data. The user can create and edit using the proper editor, according to the description in this manual. The user can also create files using neu2fstr which converts a neutral file (*.neu) supported by Femap which is a pre-post processor sold commercially as an attachment tool for FrontISTR, into FrontISTR input data.

Executing FrontISTR will output the log data file, results data file and visualization data. The existence and content of these outputs, depends on the description and analysis content in the analysis control file.

After FrontISTR is executed, the visualization data can also be created from the created results file by the hecmw_vis program which is a tool included in *HEC-MW*.

1.3.2 Overall Control Data

This file specifies the input file and results output file of the mesh data and analysis control data.

The details of the overall control data are described in Chapter 5.

(Example)

```
!MESH, NAME=fstrMSH ,TYPE=HECMW-DIST
```

Definition of header of the distributed mesh data file (mandatory in the domain distribution model) : Foo_P16

```
!MESH, NAME=fstrMSH ,TYPE=HECMW-ENTIRE
```

Definition of mesh data file name (mandatory in the single domain model) : Foo.msh

```
!CONTROL,NAME=fstrCNT
```

Definition of analysis control data file name (mandatory) : Foo.cnt

```
!RESULT,NAME=fstrRES ,IO=OUT
```

Definition of analysis results data file name (arbitrary) : Foo.res

```
!RESULT,NAME=vis_out ,IO=OUT
```

Definition of visualization file name (arbitrary) : Foo.vis

1.3.3 Mesh Data

This file defines the finite element mesh, and defines the material data and section data. This file also defines the group data used in analysis control data.

The details of the mesh data are described in Chapter 6

(Example)

```
!HEADER                                ——— Setting of mesh title
  TEST MODEL A361
!NODE                                 ——— Definition of node coordinates
  0.0 ,0.0 ,0.0
!ELEMENT,TYPE=361                      ——— Definition of element connectivity
  1001 ,1 ,2 ,3 ,4 ,5 ,6 ,7 ,8
!SECTION,TYPE=SOLID,EGRP=ALL,MATERIAL=M1 ——— Definition of section data
!MATERIAL,NAME=M1,ITEM=1                 ——— Definition of material data
!ITEM=1,SUBITEM=2
  4000.,      0.3
!NGROUP,NGRP=FIX,GENERATE              ——— Definition of node group
  1001 ,1201 ,50
!EGROUP,EGRP=TOP,GENERATE              ——— Definition of element group
  1001 ,1201 ,1
!END
```

1.3.4 Analysis Control Data

This file defines analysis control data, such as the type of analysis, displacement boundary conditions, concentrated load and etc. Control of solver and the control data of the visualizer are also included in the analysis control data.

The details of the analysis control data described in Chapter 7

(EXAMPLE)

```

!! Analysis Type
!SOLUTION,TYPE=STATIC           ----- Specification of analysis type
!! Analysis control data
!BOUNDARY                         ----- Definition of displacement boundary conditions
    FIX,1,3,0.0
!CLOAD                            ----- Definition of concentrated load conditions
    CL1,1,-1.0
!DLOAD                            ----- Definition of distributed load conditions
    ALL,BX,1.0
!REFTEMP                           ----- Definition of reference temperature
    20.0
!TEMPERATURE                      ----- Definition of heat load (temperature) coditions
    ALL,100.0
!! Solver Control Data
!SOLVER,METHOD=CG,PRECOND=1,TIMELOG=YES,ITERLOG=YES ----- Control of Solver
    10000,2
    1.0e-8,1.0,0.0
!! Post Control Data
!WRITE,RESULT                     ----- Analysis results data output
!WRITE,VISUAL                     ----- Visualizer control
!! Visualizer
!visual                           ----- Hereinafter , the control data of the visualizer
!surface_num=1
!surface_style=1
!END

```

1.3.5 Output File

When the execution is completed, the log file (Ext.log) will be output. The analysis results file (Ext.res) for visualization will also be output by specifying the output.

The contents of the log files shown in the following will be output.

- Displacement, strain, Max/Min values of stress component
- Eigenvalues
- Engenvector values

1.3.6 Execution Procedure

1.3.6.1 1. Preparation of FrontISTR

Save the main body of FrontISTR (Linux ver.: fistr1 , Windows ver. : fistr1 .exe) in the path directory, or the current direct directory at the time of execution.

1.3.6.2 2. Preparation of Input Files

Prepare three types of input files hecmw_ctrl.dat, analysis control and (single or distributed domain) mesh data, and enter the file name (pathname) of the analysis control data and mesh data in hecmw_ctrl.dat. If necessary, also specify the analysis results data file and the visualization data file.

1.3.6.3 3. Execution of Single Domain Analysis

Start the Linux terminal or the command prompt of Windows, move the current directory to the directory with the input file, and execute the analysis as follows (however, > refers to the prompt).

Example : In the case of Linux

> ./fistr1

Example : In the case of Windows

```
> fistr1
```

1.3.6.4 4. Parallel Execution on Linux

In the Linux ver., the MPI must be installed in advance, and compiled for parallel execution. For details of the compiling method, refer to the Installation Manual. The execution depends on the settings of the execution environment of MPI. An example of execution int four domains is shows in the following.

```
> mpirun -np 4 ./fistr1
```

1.3.6.5 5. Parallel Execution on Windows

In the Windows ver., it is necessary to download the library of MPICH2 from the following URL and install. For the method of parallel execution, refer to MPICH2 Manual.

- <http://www-unix.mcs.anl.gov/mpi/mpich/>

1.3.6.6 6. Execution of Parallel Domain Contact Analysis

FrontISTR requires hecmw_ctrl.dat, analysis control data and distributed domain mesh data for parallel execution in general. Parallel domain contact analysis does not requires distributed domain mesh as it partitions the domain after reading the single domain mesh data.

```
!MESH,NAME=fstrMSH ,TYPE=HECMW-ENTIRE
```

The all other processes of execution procedure are the same as the other analysis.

1.4 Element Library and Material Data

1.4.1 Element Library

In FrontISTR, the element groups shows in Table 4.1.1 can be used for analysis. Since HEC-MW is used to input the mesh data into FrontISTR, the following descriptions of the element library is based on the description of HEC-MW. The element library is shown in Figure 4.1.1, and element connectivity and the definitions of the surface number are shown in Figure 4.1.2.

Table 4.1.1: Element Library List

Element Types	Element No.	Description
Line element	111	Two node link element
	112	Three node link element
Plane element	231	Three node triangular element
	232	Six node triangular quadratic element
	241	Four node quadrilateral element
	242	Eight node quadrilateral quadratic elment
Solid element	301	Two node truss element
	341	Four node tetrahedral element
	342	Ten node tetrahedral quadratic element
	351	Six node pentahedral element
	352	Fifteen node pentahedral quadratic element
	361	Eight node hexahedral element
Interface element	362	Twenty node hexahedral quadratic element
	541	Quadrilateral cross section linear element
	542	Quadrilateral cross section quadratic element
Beam element	611	Two node beam element
	641	Two node beam element (with four 3-dof nodes)

Element Types	Element No.	Description
Shell element	731	Three node three-dimensional linear element
	732	Six node three-dimensional quadratic element
	741	Four node three-dimensional linear element
	743	Nine node three-dimensional quadratic element
	761	Three node three-dimensional linear element (with six 3-dof nodes)
	781	Four node three-dimensional linear element (with eight 3-dof nodes)

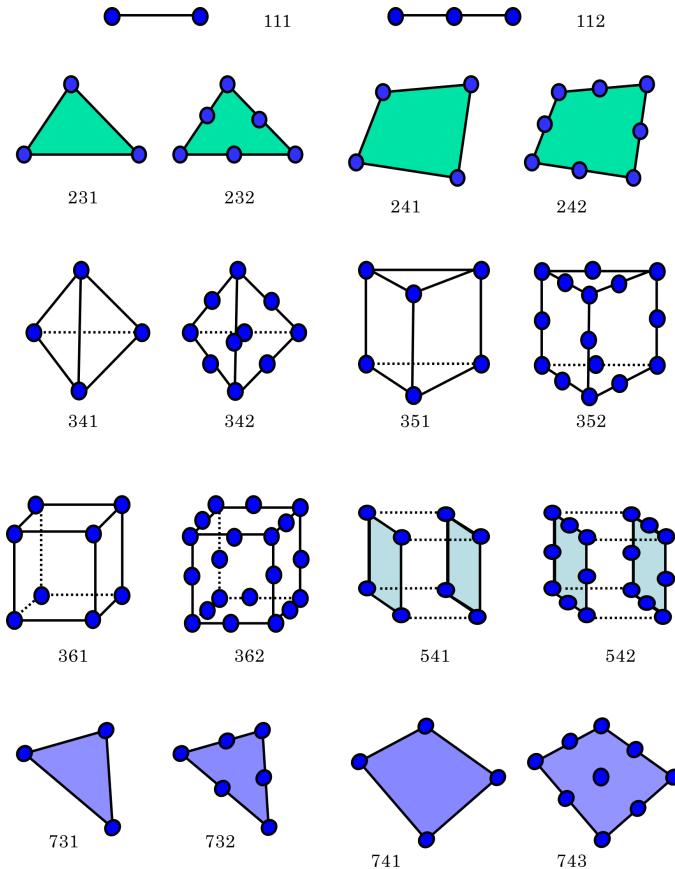


Figure 2: Element Library

Figure 4.1.1: Element Library

1.4.1.1 (Line Element)



Figure 3: Line Element

1.4.1.2 (Triangular Plane Element)

Surface No.	Linear	Quadratic
1	1 - 2	1 - 6 - 2
2	2 - 3	2 - 4 - 3
3	3 - 1	3 - 5 - 1

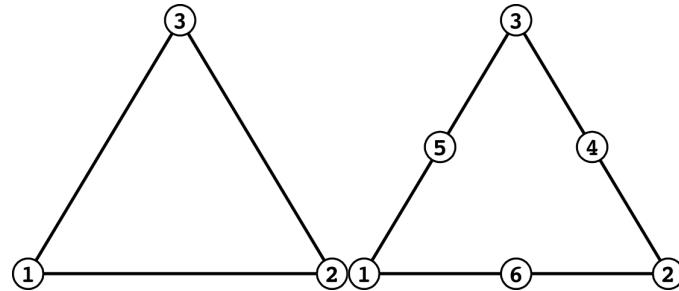


Figure 4: Triangular Plane Element

1.4.1.3 (Quadrilateral Plane Element)

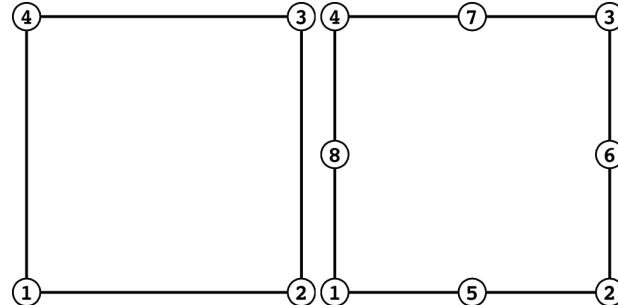


Figure 5: Quadrilateral Plane Element

Surface No.	Linear	Quadratic
1	1 - 2	1 - 5 - 2
2	2 - 3	2 - 6 - 3
3	3 - 4	3 - 7 - 4
4	4 - 1	4 - 8 - 1

1.4.1.4 (Tetrahedral Element)

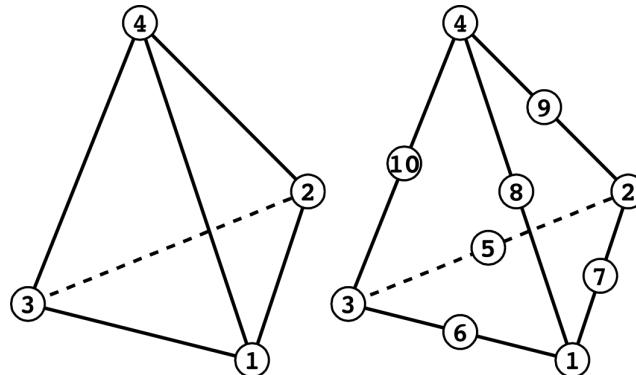


Figure 6: Tetrahedral Element

Surface No.	Linear	Quadratic
1	1 - 2 - 3	1 - 7 - 2 - 5 - 3 - 6
2	1 - 2 - 4	1 - 7 - 2 - 9 - 4 - 8
3	2 - 3 - 4	2 - 5 - 3 - 10 - 4 - 9
4	3 - 1 - 4	3 - 6 - 1 - 10 - 4 - 8

1.4.1.5 (Pentahedral Element)

Surface No.	Linear	Quadratic
1	1 - 2 - 3	1 - 9 - 2 - 7 - 3 - 8
2	4 - 5 - 6	4 - 12 - 5 - 10 - 6 - 11
3	5 - 6 - 7	5 - 11 - 6 - 10 - 7 - 12

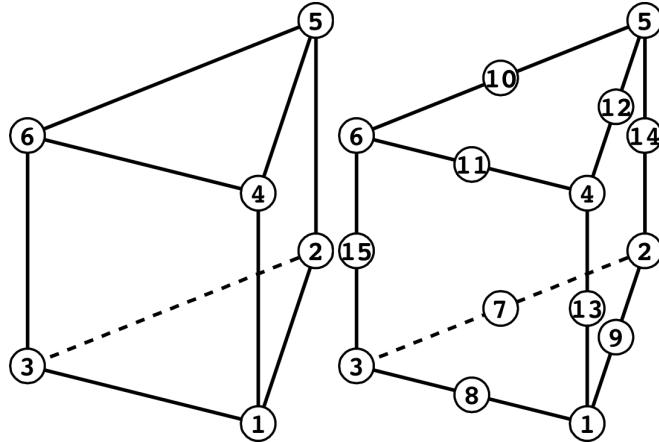


Figure 7: Pentahedral Element

1.4.1.6 (Hexahedral Element)

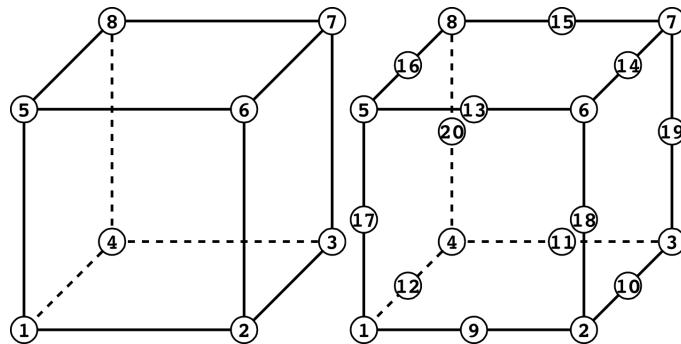


Figure 8: Hexahedral Element

Surface No.	Linear	Quadratic
1	1 - 2 - 3 - 4	1 - 9 - 2 - 10 - 3 - 11 - 4 - 12
2	5 - 6 - 7 - 8	5 - 13 - 6 - 14 - 7 - 15 - 8 - 16
3	1 - 2 - 6 - 5	1 - 9 - 2 - 18 - 6 - 13 - 5 - 17
4	2 - 3 - 7 - 6	2 - 10 - 3 - 19 - 7 - 14 - 6 - 18
5	3 - 4 - 8 - 7	3 - 11 - 4 - 20 - 8 - 15 - 7 - 19
6	4 - 1 - 5 - 8	4 - 12 - 1 - 17 - 5 - 16 - 8 - 20

1.4.1.7 (Beam Element)



Figure 9: Beam Element

1.4.1.8 (Beam Element with 3-dof nodes)

Nodes 1 and 2 for translational DOF, nodes 3 and 4 for rotational DOF.

1.4.1.9 (Triangular Shell Element)

Surface No.	Linear	Quadratic
1	1 - 2 - 3 [front]	1 - 6 - 2 - 4 - 3 - 5 [front]
2	3 - 2 - 1 [back]	3 - 4 - 2 - 6 - 1 - 5 [back]

1.4.1.10 (Triangular Shell Element with 3-dof nodes)

Node 1, 2 and 3 for translational DOF, nodes 4, 5 and 6 for rotational DOF.

Surface No.	Linear	Quadratic
1	1 - 2 - 3 - 4 [front]	1 - 5 - 2 - 6 - 3 - 7 - 4 - 8 [front]
2	4 - 3 - 2 - 1 [back]	4 - 7 - 3 - 6 - 2 - 5 - 1 - 8 [back]

1.4.1.12 (Quadrilateral shell Element with 3-dof nodes)

Nodes 1, 2, 3 and 4 for translational DOF, nodes 5, 6, 7 and 8 for rotational DOF.

Surface No.	Linear
1	1 - 2 - 3 - 4 [front]
2	4 - 3 - 2 - 1 [back]

Figure 4.1.2: Connectivity and Surface Number

1.4.2 Material Data

1.4.2.1 Elastic Static Analysis, Linear Dynamic Analysis and Eigenvalue Analysis

In the elastic static analysis and eigenvalue analysis of FrontISTR, it is necessary to use isotropic elastic material and define the Young's modulus, Poisson's ratio density and the coefficient of linear expansion for each element. The property value of these materials is defined in the header !SECTION and !MATERIAL of the data. An example of the definition is in the following.

(Example)

```
!! Definition of SECTION
!SECTION,TYPE=SOLID,EGRP=ALL,MATERIAL=M1
```

In the above !SECTION, the material data name of the element belonging to the group name=ALL in the solid type element is defined as M1.

Next, the definition method of the material data is shown in the following.

(Example)

```
!! Intention of defining three types of property values in the material name M1
!MATERIAL,NAME=M1,ITEM=3
```

```
!! Young's modulus and Poisson's ratio are defined in !ITEM=1 (mandatory)
!ITEM=1,SUBITEM=2
4000.0,0.3
```

```
!! The mass density must be defined in !ITEM=2 (mandatory in the case of ITEM=3)
!ITEM=2
8.0102E-10
```

```
!! The coefficient of linear expansion must be defined in !ITEM=3
!ITEM=3
1.0E-5
```

As long as each item number and the type of physical property are compatible, and the ITEM number is correct, it can be defined in any order. However, ITEM=1 must be defined in the order of the Young's modulus and Poisson's ratio.

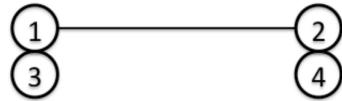


Figure 10: Beam Element with 3-dof nodes

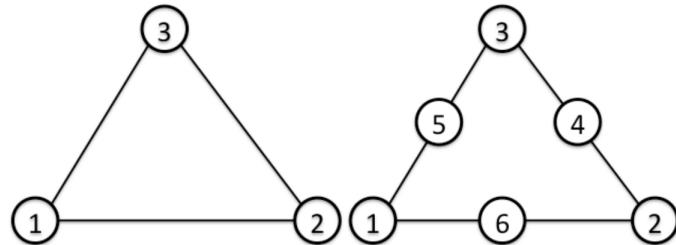


Figure 11: Triangular Shell Element

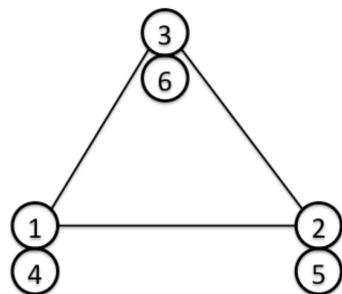


Figure 12: Triangular Shell Element with 3-dof nodes

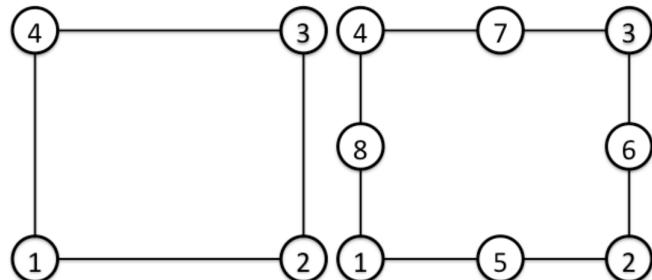


Figure 13: Quadrilateral Shell Element

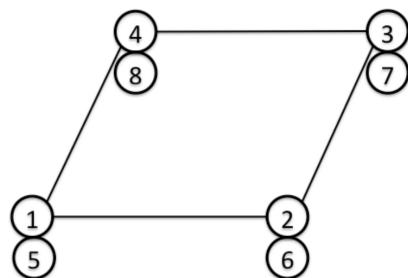


Figure 14: Quadrilateral shell Element with 3-dof nodes

1.4.2.2 Heat Conduction Analysis

In the heat conduction analysis of FrontISTR, isotropic material taking into consideration the temperature dependency can be used. Regarding the property values to be defined, the density, specific heat and thermal conductivity are defined for the link, plane surface, solid and shell element, and the gap heat transfer coefficient and the gap radiation factor are defined in the interface element. An example of the definition method of these property values is shown in the following.

1.4.2.2.1 (1) In the case of link, plane surface and solid element

Defined in the !SECTION and !MATERIAL header.

(Example)

```
!! Definition of section
!SECTION,TYPE=SOLID,EGRP=ALL,MATERIAL=M1
```

In the above !SECTION, the material data name of the element belonging to the group name=ALL in the solid type element is defined as M1. The following shows the definition method of the material data.

(Example)

```
!! Intension of defining three types of property values
!! in the material of material name M1
!MATERIAL,NAME=M1,ITEM=3
```

```
!! The density and temperature are defined in !ITEM=1 (mandatory)
!ITEM=1,SUBITEM=1
7850., 300.
7790., 500.
7700., 800.
```

```
!! The specific heat and temperature are defined in !ITEM=2 (mandatory)
!ITEM=2
0.465, 300.
0.528, 500.
0.622, 800.
```

```
!! The thermal conductivity and temperature are defined in !ITEM=3 (mandatory)
!ITEM=3
43., 300.
38.6, 500.
27.7, 800.
```

Since the type of physical properties are identified by each item number in FrontISTR, it can be defined in any order as long as both are consistent.

1.4.2.2.2 (2) In the case of an interface element

Defined in the !SECTION header. (Material data is not required)

(Example)

```
!! Definition of section
!SECTION,TYPE=INTERFACE,EGRP=GAP
1.0, 20.15, 8.99835E-9, 8.99835E-9
```

In the above !SECTION, the gap parameter of the element belonging to the group name=GAP in the interface element is defined.

- 1st parameter : Gap width
- 2nd parameter : Gap heat transfer coefficient

- 3rd parameter : Gap radiation factor 1
- 4th parameter : Gap radiation factor 2

1.4.2.2.3 (3) In the case of a shell element

Defined in the !SECTION and MATERIAL header.

(Example)

```
!! Definition of section
!SECTION, TYPE=SHELL, EGRP=SH, MATERIAL=M2
 10.0, 5
```

In the above !SECTION, the shell characteristics of the element belonging to the group name=SH in the shell type element is defined.

- 1st parameter : Shell thickness
- 2nd parameter : Integration points in the thickness direction

This also means that the material data name of the material physical properties of the element which belong to this group is defined as M2.

For single layered isotropic shell, the definition method of the material physical properties is the same as in the case of the solid element. Refer to the description of the solid element.

In addition, another definition method stated below is available for multi-layered and/or orthotropic shell.

(Example) Isotropic, single-layered shell

```
!MATERIAL, NAME=M1, ITEM=1
!ITEM=1,SUBITEM=4
 0, 200000, 0.3, 2.0
```

- 1st parameter : 0=isotropic
- 2nd parameter : Young's modulus
- 3rd parameter : Poisson's ratio
- 4th parameter : Shell thickness

(Example) Isotropic, double-layered shell

```
!MATERIAL, NAME=M1, ITEM=1
!ITEM=1,SUBITEM=7
 0, 200000, 0.3, 2.0, 200000, 0.3, 2.0
```

- 1st parameter : 0=isotropic
- 2nd parameter : Young's modulus (1st layer)
- 3rd parameter : Poisson's ratio (1st layer)
- 4th parameter : Shell thickness (1st layer)
- 5th parameter : Young's modulus (2nd layer)
- 6th parameter : Poisson's ratio (2nd layer)
- 7th parameter : Shell thickness (2nd layer)

By specifying as many material constants as needed for the number of layers, the material is automatically treated as multi-layered shell. The total thickness of the shell is the sum of the thickness of all the layers. The layers are ordered from the top surface of the shell with respect to the normal direction.

(Example) Orthotropic, single-layered shell

```
!MATERIAL, NAME=M1, ITEM=1
!ITEM=1, SUBITEM=9
 1, 28600., 0.15, 32.3, 28600., 12434., 12434., 12434., 0.0
```

- 1st parameter : 1=orthotropic
- 2nd parameter : Young's modulus E1
- 3rd parameter : Poisson's ratio n12

- 4th parameter : Shell thickness
- 5th parameter : : Young's modulus E2
- 6th parameter : Shear modulus G12
- 7th parameter : Shear modulus G23
- 8th parameter : Shear modulus G13
- 9th parameter : Orthotropy angle

Orthotropy angle is specified in [degree].

(Example) Orthotropic, double-layered shell

```
!MATERIAL, NAME=M1, ITEM=1
!ITEM=1, SUBITEM=17
1, 28600., 28600., 0.15, 32.3, 12434., 12434., 28600., 12434., 12434., 12434., 12434., 0.0
```

- 1st parameter : 1=orthotropic
- 2nd parameter : Young's modulus E1 (1st layer)
- 3rd parameter : Poisson's ratio n12 (1st layer)
- 4th parameter : Shell thickness (1st layer)
- 5th parameter : : Young's modulus E2 (1st layer)
- 6th parameter : Shear modulus G12 (1st layer)
- 7th parameter : Shear modulus G23 (1st layer)
- 8th parameter : Shear modulus G13 (1st layer)
- 9th parameter : Orthotropy angle q (1st layer)
- 10th parameter : Young's modulus E1 (2nd layer)
- 11th parameter : Poisson's ratio n12 (2nd layer)
- 12th parameter : Shell thickness (2nd layer)
- 13th parameter : Young's modulus E2 (2nd layer)
- 14th parameter : Shear modulus G12 (2nd layer)
- 15th parameter : Shear modulus G23 (2nd layer)
- 16th parameter : Shear modulus G13 (2nd layer)
- 17th parameter : Orthotropy angle (2nd layer)

1.4.3 Nonlinear Static Analysis

In the nonlinear static analysis of FrontISTR, in addition to the method to define the !SECTION and !MATERIAL shown in item 4.2.1, !ELASTIC, !HYPERELASTIC, !PLASTIC and etc. in the analysis control data can also be defined. An example of the definition is shown in the following.

(Example) Definition of hyperelastic material

```
!MATERIAL
!! Definition of the Neo Hooke hyperelastic material
!HYPERELASTIC, TYPE=NEOHOOKE
!! C_10 and D are defined (mandatory)
1000.0, 0.00005
```

(Example) Definition of the elastoplastic material

```
!MATERIAL
!! Definition of the isotropic elastic material
!ELASTIC, TYPE=ISOTROPIC
!! The Young's modulus and the Poisson's ratio are defined (mandatory)
21000.0, 0.3
!! Definition of the Drucker-Prager plastic material
!PLASTIC, TYPE=DRUCKER-PRAGER
```

```
!! Viscosity , angle of friction and hardening factor are defined (mandatory)
500.0 , 4.0 , 10.0
```

1.5 Step control

1.5.1 About time on analysis

In the analysis of FrontISTR, following definitions of “time” are used:

- Current time : Total time from start of analysis.
- Step time : Time from start in each of step.
- Time span : Duration of time for each step.
- Relative time : Ratio of “Step time” to “Time span”.
- Time increment : Increment from “Current time” to “time at which the equilibrium is attained”.

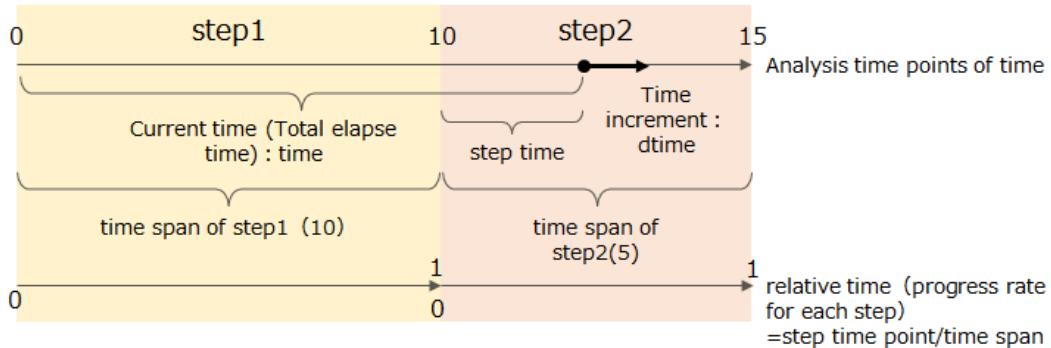


Figure 15: Definition of terms for analysis time

Figure 4.7.1 Definition of terms for analysis time

1.5.2 Control for static analysis

In static analysis of FrontISTR holds one analysis step or multiple continuous analysis steps. In each step, analysis gives a pair of basic boundary condition that is incremental method analysis.

Below, iteration of analysis is called “step loop”, and incremental method is called “sub-step loop”.

Incremental control of static analysis could be choose from the following solution.

- Analyze with fixed time increments. If convergence fails, the calculation will be stoped immediately.
- Automatic time increment and cutback. Depending on the convergence, change the increment width. If the calculation fails, recalculate after decreasing the increment width.

1.5.2.1 Automatic time increment and cut-back

Flow is automatic time increment and cut-back is as follows(Figure 4.7.2).

Figure 4.7.2 Flow of automatic time increment and cut-back

Overview of program flows is as follows,

1. Loop from step 1 to final step(N_step):
2. The time increment base dtime_base is determined from the current dtime_base and the convergence status in the previous sub-step. First time uses initdt.
3. The actual dtime is determined by “step end or the remaining time until the nearest specified output time and smaller of dtime_base.
4. Try calculation of equilibrium of forces at time+dtime.
5. If converge, the time is forward by dtime, when it fails, restore time and return to 2.

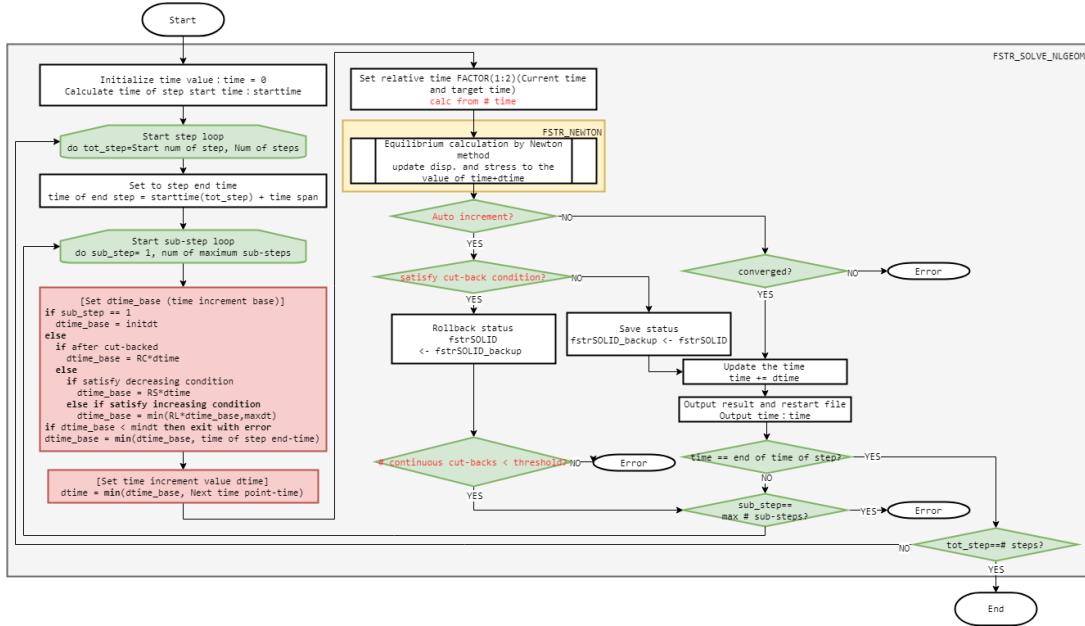


Figure 16: Flow of automatic time increment and cut-back

6. Step ends when time reaches step end time.

If any of the following cases occurs during the analysis, the nonlinear static analysis is considered to have failed and an error is terminated.

- When the number of sub-steps reaches the upper limit before time reaches the step end time.
- When the time increment dtime_base falls below the time increment lower limit mindt.
- When does not converge continuously for the specified number (N_C) of times.

1.5.2.2 Adjust time increment baseline dtime_base

The first dtime_base of the step is set to the value of the specified initial time increment initdt. Otherwise, it is set as follows according to the convergence status of the previous sub-step.

1. When failed to converge (a.k.a. cut-back) the value dtime_base multiplied by R_C (reduction ratio of cut-back).
2. When be a successful to converge
 1. Corresponds to the decrease condition : the value of dtime_base multiplied by R_S (decrease ratio).
 2. In the case of an increase condition, not a decrease condition : the smaller value of 'dtime_base' multiplied by 'R_L' (increase ratio) and the time increment upper limit maxdt.
 3. If neither : dtime_base dose not change.

1.5.2.3 Increase / decrease conditions

In the automatic time incremental adjustment function uses the following variables to determine the increment / decrement conditions:

- N_max : Maximum number of Newton iterations of previous substep.
- N_sum : Total number of Newton iterations in the previous substep(Same as N_max, when there is no repeated contact).
- N_cont : Number of contact iterations in previous substep.

Decrease condition is when both of the following are met:

- One of N_max, N_sum or N_cont exceeds the respective threshold NS_max, NS_sum or NS_cont.
- When the above conditions are satisfied by a substep that continues N_S times or more.

Increase condition is when both of the following are met:

- N_max, N_sum and N_cont are all within their respective threshold NL_max, NL_sum and NL_cont.

- The above condition is met by substep that continues more than N_L times.

1.5.2.4 Specify calculation time and output time

With automatic time increments, the increments change with the convergence state, so it is not possible to determine in advance when to run the equilibrium calculation and result output. If this is inconvenient, you can specify a list of output times to perform the equilibrium calculation and result output at the desired time.

For steps that include a list of output times, dtime is adjusted so that the calculation is always performed at specified time.

1.5.2.5 Usage of function of automatic time increments and cut-back

Settings related to this function are made in the analysis control file.

The function of automatic time increment and cut-back are enabled specified by TYPE_INC=AUTO on the !STEP card. For parameters that adjust the time increment, define !AUTOINC_PARAM and specify it in each step using the !STEP, AUTOINCPARAM'. If not specified, the default value of!AUTOINC_PARAM' is used.

The output time is specified for each step with the !STEP, TIMEPOINTS parameter after defining the time list with !TIME_POINTS.

1.5.2.5.1 Example

Enable automatic incremental adjustment and set initial time increment 0.01, step duration 2.5, time increment lower limit 1E-5, time increment upper limit 0.3, maximum substep number to 200. Specify AP1 for the auto-increment parameter set and 1.5, 2.7 and 3.9 for the output time.

```

!AUTOINC_PARAM, NAME=AP1
0.25, 10, 50, 10, 1
1.25, 1, 1, 1, 2
0.25, 5
!TIME_POINTS, TIME=TOTAL, TIME=,NAME=TP1
1.5
2.7
3.9
!STEP, INC_TYPE=AUTO, SUBSTEPS=200, AUTOINCPARAM=AP1, TIMEPOINTS=TP1
0.01, 2.5, 1E-5, 0.3

```

1.5.3 Control for dynamic analysis

In this development code, dynamic analysis direct time integration method consists of one analysis step.

Incremental control for dynamic analysis is only fixed increments and automatic time increments are not available.

1.6 Overall Control Data

1.6.1 Outline of Overall Control Data

The overall control data is for defining the file name of the input/output files for FrontISTR. The features of the overall control data file are as follows.

- This is an ASCII format file based on a free format.
- This file consists of a header which starts with “!” and the data following this.
- The order of description of the header is basically free.
- A “,” is used as a punctuation mark of the data.

1.6.2 Input Rules

The overall control data file consists of a header line, data line and a comment line. One header is always included in the header line.

Header The meaning of the data and data block is specified in the overall control data file.

When the head of the term starts with a “!”, it is considered to be a header.

Header Line The header and the parameter accompanying this are described in this line.

The header line must start with a header.

When a parameter is required, a “,” must be used to continue after that.

When the parameter takes on a value, use an “=” after the parameter and describe the value after that.

Data Line The data line starts from the next line of the header line, and the necessary data is described.

The data lines may be in multiple lines; however, this is determined according to the rules of the data description defined by each header.

There are cases where data lines are not required.

Punctuation A comma “,” is used as a punctuation of the data.

Handling of Blanks Blanks are disregarded.

Name Regarding the characters which can be used for the name, there is the underscore “_”, hyphen “-”, and alphanumeric characters “a – z, A – Z, 0 – 9”; however, the first letter of the name must start with “_”, or an alphabetic character “a – z, A – Z”.

There is no distinction between uppercase and lowercase letters, and all letters are internally handled as uppercase letters.

The maximum length of the name is 63 characters.

File Name Regarding the characters which can be used for the file name, there are the underscore “_”, hyphen “-”, period “.”, slash “/”, and the alphanumeric characters “a – z, A – Z, 0 – 9”.

As long as there is no specific description, a path can be included in the file name. Both the relative path and the absolute path can be specified.

The maximum length of the file name is 1,023 characters.

Floating Point Data Exponents are optional. An “E” or “e” character must be added before the exponent.

The selection of “E” or “e” is optional. “D” or “d” can not be used.

!!, # Comment Line Lines starting with “!!” or “#” are considered to be comment lines, and are disregarded. A comment line can be inserted in any position in the file, and there are no restrictions on the number of lines.

1.6.3 Header List

The overall control data consists of the following headers.

Header Name	Contents
!CONTROL	Analysis control data definition
!MESH	Mesh data definition
!RESTART	Restart data definition
!RESULT	Analysis results data definition
!SUBDIR	Dividing to sub directories

In each header, there are data items which are compatible to the parameter and each header. Each of the above headers is described in the following with examples of data creation.

1.6.3.1 (1) !CONTROL

Specifies the analysis control data file.

!CONTROL,NAME=<name>
file

Example of Use

```
!CONTROL, NAME=fstrCNT  
myctrl.dat
```

Parameter	
NAME	Identifier (mandatory)

Parameter Name	Parameter Value	Contents
NAME	fstrCNT	Analysis control data

Parameter Name	Contents
file	Analysis control data file name (both the relative path and the absolute path can be specified. When the relative path is specified, it becomes the path from the current directory.)

1.6.3.2 (2) !MESH

Specifies the mesh data file.

```
!MESH, NAME=<name>, TYPE=<type> [ , optional parameter ]
fileheader
```

```
#Example of Use
!MESH, NAME=fstrMSH, TYPE=HECMW-DIST, REFINE=1
Mesh.in
```

Parameter	
NAME	Identifier (mandatory)
TYPE	Mesh type (mandatory)
IO	Input/output specification (omissible)
REFINE	Mesh subdivision specification (arbitrary)

Parameter Name	Parameter Value	Contents
NAME	fstrMSH	Solver input data
	part_in	Partitioner input data
	part_out	Partitioner output data
TYPE	HECMW-DIST	HEC-MW distribution mesh data
	HECMW-ENTIRE	HEC-MW single domain mesh data
IO	IN	For input (default)
	OUT	For output
REFINE	<integer>	Number of mesh subdivisions

Parameter Name	Contents
fileheader	Header of the mesh data file name (both the relative path and the absolute path can be specified. When the relative path is specified, it becomes the path from the current directory.)

Note:

The existence of IO parameters, or parameter values will have no affect on others. When the TYPE is HECMW-DIST, the end of the file name “.<rank>” is excluded for the file header specified in the data line.

1.6.3.3 (3) !RESTART

Specifies the restart data file.

```
!RESTART, NAME=<name>, IO=<io>
fileheader
```

```
# Example of Use
!RESTART, NAME=restart-in, IO=IN
restart.in
```

Parameter	
NAME	Identifier (mandatory)
IO	Input/output specification (mandatory)

Parameter Name	Parameter Value	Contents
NAME	<name>	Identifier
IO	IN	For input
	OUT	For output
	INOUT	Common to input/output

Parameter Name	Contents
fileheader	Header of the restart data file name (both the relative path and the absolute path can be specified. When the relative path is specified, it becomes the path from the current directory.)

Note:

The file name created by this definition is the file header+.<rank>.

1.6.3.4 (4) !RESULT

Specifies the analysis results data file.

```
!RESULT, NAME=<name> [ , optional parameter]
fileheader
```

```
# Example of Use
!RESULT, NAME=fstrRES, IO=OUT, TYPE=BINARY
result.out
```

Parameter	
NAME	Identifier (mandatory)
IO	Input/output specification (mandatory)
TYPE	Output format (omissible)

Parameter Name	Parameter Value	Contents
NAME	fstrRES	Solver output data, Visualizer input data
	fstrTEMP	Temperature input data (Result of heat analysis)
	vis_out	Visualizer output data
IO	IN	For input
	OUT	For output
TYPE	TEXT	Text format (default)
	BINARY	Binary format

Parameter Name	Contents
fileheader	Header of the analysis results data file name (both the relative path and the absolute path can be specified. When the relative path is specified, it becomes the path from the current directory.)

Note:

The file name created by this definition is the file header+.<rank>.

1.6.3.5 (5) !SUBDIR

Specifies storing input/output files into subdirectories

!SUBDIR, ON [,optional parameter]

```
# Example of Ud
!SUBDIR, ON, LIMIT=8000
```

Parameter	
ON	Enable (mandatory)
LIMIT	Number of files (omissible)

Parameter Name	Parameter Value	Contents
ON	N/A	

Parameter Name	Parameter Value	Contents
LIMIT	<integer>	Maximum number of files per directory (default:5000)

Note:

The input/output data which are comprised of plural files are automatically stored into each subdirectory by this definition. Furthermore, if the number of ranks exceeds “LIMIT”, those files are divided to subdirectories such as TRUNK0, TRUNK1.

1.7 Single Domain Mesh Data

1.7.1 Outline of Single Mesh Data

In order to acquire the mesh data of FrontISTR, there are two methods, such as the method to input the single domain mesh data file, and the method to input the distributed mesh data file to perform parallel processing. Single domain mesh data is described in this section.

The features of single domain mesh data are as follows.

- This is an ASCII format file based on a free format.
- This file consists of a header which starts with “!” and the data following this.
- The order of description of the header is basically free.
- A “,” is used as a punctuation mark of the data.

1.7.2 Input Rules

The single domain mesh data file consists of a header line, data line and a comment line. One header is always included in the header line.

Header The meaning of the data and data block is specified in the single domain mesh data file.

When the head of the term starts with a “!”, it is considered to be a header.

Header Line The contents of the header and the parameter accompanying this are described in this line.

The header line must start with a header. When a parameter is required, a “,” must be used to continue after that.

When the parameter takes on a value, use an “=” after the parameter and describe the value after that. The header line can not be described in more than two lines.

Data Line The data line starts after the header line, and the necessary data is described.

The data lines may be in multiple lines; however, this is determined according to the rules of the data description defined by each header.

There are cases where data lines are not required.

Punctuation A comma “,” is used as a punctuation of the data.

Handling of Blanks Blanks are disregarded.

Name Regarding the characters which can be used for the name, there is the underscore “_”, hyphen “-”, and alphanumeric characters “a – z, A – Z, 0 – 9”; however, the first letter of the name must start with “_”, or an alphabetic character “a – z, A – Z”. There is no distinction between uppercase and lowercase letters, and all letters are internally handled as uppercase letters.

The maximum length of the name is 63 characters.

File Name Regarding the characters which can be used for the file name, there are the underscore “_”, hyphen “-”, period “.”, slash “/”, and the alphanumeric characters “a – z, A – Z, 0 – 9”.

As long as there is no specific description, a path can be included in the file name. Both the relative path and the absolute path can be specified.

The maximum length of the file name is 1,023 characters.

Floating Point Data Exponents are optional. An “E” or “e” character must be added before the exponent.

The selection of “E” or “e” is optional. “D” or “d” can not be used.

!!, # Comment Line Lines starting with “!!” or “#” are considered to be comment lines, and are disregarded.

A comment line can be inserted in any position in the file, and there are no restrictions on the number of lines.

1.7.3 Header List of Single Domain Mesh Data

The single domain mesh data consists of the following headers.

Header Name	Contents
!CONTROL	Specify of analysis control file
!MESH	Specify of mesh file
!RESTART	Specify of restart file
!RESULT	Specify of analysis results file
!SUBDIR	Specify of making sub-directories

Header Name	Contents	Description No.
!AMPLITUDE	Unsteady load	M1-1
!EGROUP	Element group	M1-2
!ELEMENT	Element information	M1-3
!EQUATION	Restricted point data	M1-4
!HEADER	Title of mesh data	M1-5
!MATERIAL	Material information	M1-6
!NGROUP	Node group	M1-7
!NODE	Node information	M1-8
!SECTION	Section information	M1-9
!SGROUP	Surface group	M1-10
!ZERO	Absolute zero-point	M1-11
!CONTACT PAIR	Contact surface pair	M1-12
!END	Read end	M1-13

In each header, there are data items which are compatible to the parameter and each header.

Each of the above headers is briefly described in the following with examples of data creation. The number indicated on the right end of the data creation is the description number of the above Table.

1.7.3.1 Example of Mesh Data

```
!HEADER                                     M1-5
  TEST MODEL CTLR10

!NODE                                         M1-8
  1, 0.00000E+00, 0.00000E+00, 0.00000E+00
  2, 0.50000E+01, 0.00000E+00, 0.00000E+00
  3, 0.10000E+02, 0.00000E+00, 0.00000E+00
  ...   ...   ...

!ELEMENT,TYPE=351                           M1-3
  1, 1, 2, 4, 34, 35, 37
  2, 2, 5, 4, 35, 38, 37
  3, 2, 3, 5, 35, 36, 38

!SECTION, TYPE=SOLID, EGRP=ALL, MATERIAL=M1    M1-9
  1.0

!MATERIAL, NAME=M1, ITEM=2                   M1-6
!ITEM=1, SUBITEM=2
  2.1E5, 0.3
!ITEM=2, SUBITEM=1
  7.8e-6
```

```

!NGROUP, NGRP=FIX, GENERATE                               M1-7
  2 , 2 , 1
  3 , 3 , 1
  1 , 1 , 1
  69 , 69 , 1
  67 , 67 , 1

!NGROUP, NGRP=CL1                                         M1-8
  50

!END                                                       M1-13

```

1.7.3.2 (1) !AMPLITUDE (M1-1)

Specifies the changes of time of the variables which provide the load conditions in the step.

```

!AMPLITUDE, NAME=<name> [ , optional parameter]
VAL1, T1, VAL2, T2, VAL3, T3 ...

```

(up to four items in one line)

Parameter	
NAME	Name (mandatory)
DEFINITION	Type (omissible)
TIME	Type of time (omissible)
VALUE	Type of value (omissible)
INPUT	External file name (omissible)

Parameter Name	Parameter Value	Contents
NAME	<name>	AMPLITUDE Name
DEFINITION	TABULAR	Default (default only in current version)
TIME	STEP TIME	Default (default only in current version)
VALUE	RELATIVE	Relative value (default)
	ABSOLUTE	Absolute value
INPUT	<filename>	External file name (omissible), can also be use together with the 2nd line or later

Parameter Name	Attributions	Contents
VAL1	R	Value at time T1
T1	R	Time T1
VAL2	R	Value at time T2
T2	R	Time T2
VAL3	R	Value at time T3
T3	R	Time T3

1.7.3.3 (2) !EGROUP (M1-2)

Definition of element group

```

!EGROUP, EGRP=<egrp> [ , optional parameter ]

```

Parameter	
EGRP	Element group name (mandatory)

Parameter

GENERATE Automatic generation of nodes belonging to the element group (omissible)
INPUT External file name (omissible)

Parameter Name	Parameter Value	Contents
EGRP	<egrp>	Element group name
GENERATE	N/A	Automatic generation of nodes belonging to the element group
INPUT	<filename>	External file name (omissible), can also be use together with the 2nd line or later

2nd Line or later (when GENERATE is not used)

(2nd Line) elem1 , elem2 , elem3 ...
(Hereinafter the same)

Parameter Name	Attributions	Contents
elemX	I	Element number belonging to the element group

2nd Line or later (when GENERATE used)

(2nd Line) elem1 , elem2 , elem3
(Hereinafter the same)

Parameter Name	Attributions	Contents
elem1	I	First element number in the element group
elem2	I	Last element number in the element group
elem3	I	Element number increment (omissible, number becomes elem3=1 when omitted)

Note:

- Any number of elements can be inserted in one line. Any number of lines can be inserted until the next option starts.
- It is necessary to define the element to be specified before “!EGROUP”.
- The element not defined in the “!ELEMENT” option will be excluded, and a warning message will be displayed.
- When the specified element exists in the same group, it will be ignored and a warning message will be displayed. All the elements belong to the element group named “ALL” (generated automatically). One group can be defined by dividing into multiple groups.

Example of Use

```
!EGROUP, EGRP=EA01
 1, 2, 3, 4, 5, 6
 101, 102
 205
!EGROUP, EGRP=EA02
 101, 102
!EGROUP, EGRP=EA01      "501, 505" are added to group "EA01".
 501, 505
!EGROUP, EGRP=EA04, GENERATE    "301, 303, 305, 307, 309, 311, 312, 313"
 301, 309, 2              are added to group "NA04".
```

1.7.3.4 !ELEMENT (M1-3)

Definition of elements

1st Line

!ELEMENT, TYPE=<type> [, optional parameter]

Parameter	
TYPE	Element type (mandatory)
EGRP	Element group name (omissible)
MATITEM	Number of physical property items when defining the material physical properties for each element (not used when defining physical properties for each section)
INPUT	External file name (omissible)

Parameter Name	Parameter Value	Contents
TYPE	111	Rod, link element (Linear)
	231	Triangular element (Linear)
	232	Triangular element (Quadratic)
	241	Quadrilateral element (Linear)
	242	Quadrilateral element (Quadratic)
	301	Truss element (Linear)
	341	Tetrahedral element (Linear)
	342	Tetrahedral element (Quadratic)
	351	Triangular prism element (Linear)
	352	Triangular prism element (Quadratic)
	361	Hexahedral element (Linear)
	362	Hexahedral element (Quadratic)
	541	Interface element (Quadrilateral cross section, Linear)
	611	Beam element (Linear)
	641	Beam element (Linear, with 3-dof nodes)
	731	Triangular shell element (Linear)
	741	Quadrilateral shell element (Linear)
	743	Quadrilateral shell element (Quadratic)
	761	Triangular shell element (Linear, with 3-dof nodes)
	781	Quadrilateral shell element (Linear, with 3-dof nodes)
EGRP	<egrp>	Element group name (omissible)
INPUT	<filename>	External file name (omissible), can also be used together with the 2nd line or later

2nd Line or later

(2nd Line) ELEM_ID, nod1, nod2, nod3, ..., MAT1, MAT2, ...
(Hereinafter the same)

Parameter Name	Attributions	Contents
ELEM_ID	I	Element number
nodX	I	Connectivity
MATy	R	Physical Property value for each element

Note:

- For details of the element types and connectivity, refer to “Chapter 4 Element Library”.
- The node specified by the connectivity must be defined before “!ELEMENT”.
- The element numbers do not have to be continued.
- The “!ELEMENT” option can be defined any number of times.
- The element number must be a natural number. This can not be omitted.
- When the same element number is used repeatedly, the value input last will be used. In this case, a warning message will be output.
- Undefined nodes can not be used for connectivity.
- The definition of one element can be described in multiple lines.

Example of Use

```
!ELEMENT, TYPE=231
1, 1, 2, 3
2, 4, 8, 5
4, 6, 7, 8
!ELEMENT, TYPE=361, EGRP=A
101, 101, 102, 122, 121, 201, 202, 222, 221
102, 102, 103, 123, 122, 202, 203, 223, 222
103, 103, 104, 124, 123, 203, 204, 224, 223
```

1.7.3.5 (4) !EQUATION (1-4)

Definition of restricted node group

1st Line

```
!EQUATION [ , optional parameter]
```

Parameter	
INPUT	External file name (omissible)

Parameter Name	Parameter Value	Contents
INPUT	<filename>	External file name (omissible), can also be used together with the 2nd line or later

2nd Line or later

```
(2nd Line) NEQ, CONST
```

```
(3rd Line or later) nod1, DOF1, A1, nod2, DOF2, A2 ... (up to seven terms for one line)
(Hereinafter repeated)
```

Parameter Name	Attributions	Contents
NEQ	I	Number of equation terms
CONST	R	Constant term of equation (right value)
nod1	I/C	1st node or node group
DOF1	I	Restricted degree of freedom of 1st node or node group
A1	R	Factor of 1st node or node group
nod2	I/C	2nd node or node group
DOF2	I	Restricted degree of freedom of 2nd node or node group
A2	R	Factor of 2nd node or node group

Note:

- When a node or a node group not defined by “!NODE” is specified, it will be ignored and a warning message will be displayed.
- In the case of “nod1=nod2”, it will be ignored and a warning message will be displayed.
- When a node group is specified, if the number of nodes is not consistent an error will occur.
- The degree of freedom number differs by the type of analysis and elements. An inconsistent degree of freedom will be ignored, and a warning message will be displayed.

Example of Use

```
!EQUATION
3
101, 1, 1.0, 102, 1, -1.0, 103, 1, -1.0
2
NG1, 2, 1.0, NG5, 2, -1.0
```

1.7.3.6 (3) !HEADER (M1-5)

Title of mesh data

1st Line

```
!HEADER
```

Parameter
N/A

2nd Line or later

```
(2nd Line) TITLE
```

Parameter Name	Attributions	Contents
TITLE	C	Header title

Example of Use

```
!HEADER
```

Mesh for CFD Analysis

Note:

- Omissible
- Although the header can use multiple lines, it can be recognized as a header up to the 127th column of the first line.
- When “!HEADER” is defined multiple times, the contents will be updated and a warning message will be displayed.

1.7.3.7 (6) !MATERIAL (M1-6)

Definition of material physical properties.

When the physical properties depend on the temperature, table input can be performed for each compatible temperature.

The table input can also be performed for the relationship of the stress strain in a stress analysis.

1st Line

```
!MATERIAL, NAME=<name> [, optional parameter]
```

Parameter

NAME	Material name (mandatory)
ITEM	Number of physical property items (omissible, becomes “1” when omitted)
INPUT	External file name (omissible)

Parameter Name	Parameter Value	Contents
NAME	<name>	Material name
ITEM	<ITEMnum>	Number of physical property items by user definition
INPUT	<filename>	External file name (omissible), can also be used together with the 2nd line or later

2nd Line or later

(2nd Line) !ITEM=1, SUBITEM=<k>

(3rd Line) VAL1-1-1, VAL1-1-2, ... VAL1-1-k, TEMP1-1

(4th Line) VAL1-2-1,VAL1-2-2, ... VAL1-2-k, TEMP1-2

(L+2nd Line) VAL1-L-1,VAL1-L-2, ... VAL1-L-k, TEMP1-L

Hereinafter , the definition is repeated until ”!ITEM=<ITEMnum>”.

Subparameter (for “!ITEM”)

SUBITEM	Number of sub physical property items defined in each of the physical property items(omissible, becomes “1” when omitted, used to define anisotropy and etc.)
---------	---

Subparameter

Name	Parameter Value	Contents
SUBITEM	<subITEMnum>	Number of sub physical property items by user definition

[When the m-th physical property depends on the temperature]

When the number of items of the temperature dependent table is N, input as follows:

!ITEM=m, SUBITEM=k

VALm1-1, ... , VALm1-k, TEMPm1

VALm2-1, ... , VALm2-k, TEMPm2

...

VALmN-1, ... , VALmN-k, TEMPm-N

Parameter Name	Attributions	Contents
VALmn-k	R	Physical property value (Temperature dependent)
TEMPmn	R	Compatible temperature

Must be input as TEMPm1 < TEMPm2 < ... < TEMPmN.

VALm1 is used when the temperature is TEMPm1 or below, and VALmN is used when the temperature is TEMPmN or more.

[When the m-th physical property does not depend on the temperature]

!ITEM=m, SUBITEM=k

VALm1-1, ... , VALm1-k

VALm2-1, ..., VALm2-k

...

VALmN-1, ..., VALmN-k

Parameter Name	Attributions	Contents
VALmn-k	R	Physical property value (not temperature dependent)

Note:

- When the material name is duplicated, an error will occur.
- When the MATERIAL referred to in the “!SECTION” option is not defined, an error will occur.
- The value used to input the physical property for each element using the parameter “MATITEM” in the “!ELEMENT” option, is used preferentially. In this case, the physical property value input using the “!MATERIAL” option will not be used.
- When the number of “!ITEM=m” suboptions and the number of parameters “ITEM” is not consistent, or when there is an undefined suboption, an error will occur.
- The “!ITEM=m” suboption does not have to be in order from the smaller m.
- When using the “!SUBITEM=k” suboption and the temperature dependency, the omitted value will become “0.0”.
- When using the temperature dependency, it must be defined in order from the lower temperature.
- When using the temperature dependency, if the same temperature is used twice or more, an error will occur.

Example of Use

```
!MATERIAL, NAME= STEEL, ITEM= 2
!ITEM=1                                ----- No temperature dependency
  35.0
!ITEM=2
  40.0, 0.0
  45.0, 100.0
  50.0, 200.0
!MATERIAL, NAME= CUPPER                ----- Number of items = 1 (Default value)
!ITEM=1                                ----- No temperature dependency
  80.0
```

Incorrect Example of Use

- Example 1: [Number of parameter “ITEM” and “!ITEM=m” suboptions are not consistent -1]

```
!MATERIAL, NAME= STEEL, ITEM= 2
!ITEM=3
  20.0
!ITEM=1
  35.0
!ITEM= 2
  40.0
```

- Example 2: [Number of parameter “ITEM” and “!ITEM=m” suboptions are not consistent -2]

```
!MATERIAL, NAME= STEEL, ITEM= 3
!ITEM=3
  20.0
!ITEM= 2
  40.0
!MATERIAL, NAME= CUPPER
...
```

Elastic Static Analysis and Eigenvalue Analysis

```

!MATERIAL, NAME=<name>, ITEM=<ITEMnum>
!ITEM=1, SUBITEM=2
<Yang_modulus>, <Poisson_ratio>
!ITEM=2
<Density>
!ITEM=3
<Expansion_coeff>

```

Parameter Name	Parameter Value	Contents
NAME	<name>	Compatible to material name, and MATERIAL of !SECTION
ITEM	<ITEMnum>	Number of physical property items by user definition (1 or more) <Yang_modulus> ... Young's modulus (mandatory) <Poisson_ratio> ... Poisson's ratio (mandatory) <Density> ... Mass density (mandatory when ITEMnum=3) <Expansion_coeff> ... Coefficient of linear expansion (when ITEMnum=3)

Example

```

!! Intention of defining three types of property values
!! in the material of material name M1
!MATERIAL, NAME=M1, ITEM=3

!! The Young's modulus and Poisson's ratio is defined in !ITEM=1 (mandatory)
!ITEM=1, SUBITEM=2
4000., 0.3

!! The mass density must be defined in !ITEM=2 (mandatory in the case of ITEM=3)
!ITEM=2
8.0102-E10

!! The coefficient of linear expansion must be defined in !ITEM=3
!ITEM=3
1.0-E5

```

Heat Conduction Analysis

In the case of link, plane surface, solid and three-dimensional plate elements

```

!MATERIAL, NAME=<name>, ITEM=3
!ITEM=1, SUBITEM=2
<Density>, <Temperature>
!ITEM=2, SUBITEM=2
<Specific_heat>, <Temperature>
!ITEM=3, SUBITEM=2
<Conductivity>, <Temperature>

```

Parameter Name	Parameter Value	Contents
NAME	<name>	Compatible to material name, and MATERIAL of !SECTION
ITEM	<ITEMnum>	Number of physical property items by user definition (always 3)<Density> ... Density<Specific_heat> ... Specific heat<Conductivity> ... Thermal conductivity<Temperature> ... Temperature

Example

```

!! Intention of defining three types of property
!! values in the material of material name M1

```

```

!MATERIAL, NAME=M1, ITEM=3

!! The density and temperature are defined in !ITEM=1 (mandatory)
!ITEM=1, SUBITEM=1
 7850., 300.
 7790., 500.
 7700., 800.

!! The specific heat and temperature are defined in !ITEM=2 (mandatory)
!ITEM=2, SUBITEM=1
 0.465, 300.
 0.528, 500.
 0.622, 800.

!! The thermal conductivity and temperature are defined in !ITEM=3 (mandatory)
!ITEM=3
 43., 300.
 38.6, 500.
 27.7, 800.

```

In the case of interface element

Defined in the !SECTION header. (Material data is not required)

Example

```

!! Definition of section
!SECTION, TYPE=INTERFACE, EGRP=GAP
 1.0, 20.15, 8.99835E-9, 8.99835E-9

```

In the above !SECTION, the gap parameter of the element belonging to the “group name = GAP” in the interface element is defined.

- 1st parameter : Gap width
- 2nd parameter : Gap heat transfer coefficient
- 3rd parameter : Gap radiation factor 1
- 4th parameter : Gap radiation factor 2

Reference

```

program TEST
use hecmw
implicit REAL*8 (A-H,O-Z)
type (hecmwT_local_mesh) :: hecMESH

!C
!C !MATERIAL, NAME=SUS304, ITEM=3
!C !ITEM=1, SUBITEM= 3
!C   100.0, 200.0, 300.0, 0.00
!C   101.0, 210.0, 301.0, 1.00
!C   102.0, 220.0, 302.0, 2.00
!C   103.0, 230.0, 303.0, 3.00
!C !ITEM=3, SUBITEM= 2
!C   1000.0, , 0.00
!C   1001.0, 1., 1.00
!C   1002.0, 2., 2.00
!C   1003.0, 3., 3.00
!C !ITEM=2
!C   5000.0
!C
!C !MATERIAL, NAME=FEC, ITEM=2
!C !ITEM=1, SUBITEM= 3

```

```

!C      2100.0, 2200.0, 2300.0, 0.00
!C      2101.0, 2210.0, 2301.0, 1.00
!C      2102.0, 2220.0, 2302.0, 2.00
!C      2103.0, 2230.0, 2303.0, 3.00
!C      3103.0, 3230.0, 2304.0, 4.00
!C      !ITEM=2
!C      6000.0, 10.0
!C      6500.0, 30.0
!C

hecMESH%material%n_mat = 2

nn= hecMESH%material%n_mat
allocate (hecMESH%material%mat_name(nn))

hecMESH%material%mat_name(1)= 'SUS304'
hecMESH%material%mat_name(2)= 'FEC'

nn= hecMESH%material%n_mat
allocate (hecMESH%material%mat_ITEM_index(0:nn))
hecMESH%material%mat_ITEM_index(0)= 0
hecMESH%material%mat_ITEM_index(1)= 3
hecMESH%material%mat_ITEM_index(2)= hecMESH%material%mat_ITEM_index(1) + 2

hecMESH%material%n_mat_ITEM= hecMESH%material%mat_ITEM_index(hecMESH%material%n_mat)

nn= hecMESH%material%n_mat_ITEM
allocate (hecMESH%material%mat_subITEM_index(0:nn))

hecMESH%material%mat_subITEM_index(0)= 0
hecMESH%material%mat_subITEM_index(1)= 3
hecMESH%material%mat_subITEM_index(2)= hecMESH%material%mat_subITEM_index(1) + 1
hecMESH%material%mat_subITEM_index(3)= hecMESH%material%mat_subITEM_index(2) + 2
hecMESH%material%mat_subITEM_index(4)= hecMESH%material%mat_subITEM_index(3) + 3
hecMESH%material%mat_subITEM_index(5)= hecMESH%material%mat_subITEM_index(4) + 1

hecMESH%material%n_mat_subITEM=
&      hecMESH%material%mat_subITEM_index(hecMESH%material%n_mat_ITEM)

nn= hecMESH%material%n_mat_subITEM
allocate (hecMESH%material%mat_TABLE_index(0:nn))
hecMESH%material%mat_TABLE_index(0)= 0
hecMESH%material%mat_TABLE_index(1)= 4
hecMESH%material%mat_TABLE_index(2)= hecMESH%material%mat_TABLE_index(1) + 4
hecMESH%material%mat_TABLE_index(3)= hecMESH%material%mat_TABLE_index(2) + 4
hecMESH%material%mat_TABLE_index(4)= hecMESH%material%mat_TABLE_index(3) + 1
hecMESH%material%mat_TABLE_index(5)= hecMESH%material%mat_TABLE_index(4) + 4
hecMESH%material%mat_TABLE_index(6)= hecMESH%material%mat_TABLE_index(5) + 4
hecMESH%material%mat_TABLE_index(7)= hecMESH%material%mat_TABLE_index(6) + 5
hecMESH%material%mat_TABLE_index(8)= hecMESH%material%mat_TABLE_index(7) + 5
hecMESH%material%mat_TABLE_index(9)= hecMESH%material%mat_TABLE_index(8) + 5
hecMESH%material%mat_TABLE_index(10)= hecMESH%material%mat_TABLE_index(9) + 2

hecMESH%material%n_mat_TABLE=
&      hecMESH%material%mat_TABLE_index(hecMESH%material%n_mat_subITEM)

nn= hecMESH%material%n_mat_TABLE
allocate (hecMESH%material%mat_VAL(nn))
allocate (hecMESH%material%mat_TEMP(nn))

```

```

hecMESH%material%mat_VAL = 0.d0
hecMESH%material%mat_TEMP= 0.d0

hecMESH%material%mat_VAL ( 1)= 100.0d0
hecMESH%material%mat_TEMP( 1)= 0.0d0
hecMESH%material%mat_VAL ( 2)= 101.0d0
hecMESH%material%mat_TEMP( 2)= 1.0d0
hecMESH%material%mat_VAL ( 3)= 102.0d0
hecMESH%material%mat_TEMP( 3)= 2.0d0
hecMESH%material%mat_VAL ( 4)= 103.0d0
hecMESH%material%mat_TEMP( 4)= 3.0d0

hecMESH%material%mat_VAL ( 5)= 200.0d0
hecMESH%material%mat_TEMP( 5)= 0.0d0

hecMESH%material%mat_VAL (13)= 5000.0d0

hecMESH%material%mat_VAL (14)= 1000.0d0
hecMESH%material%mat_TEMP (14)= 0.0d0
hecMESH%material%mat_VAL (15)= 1001.0d0
hecMESH%material%mat_TEMP (15)= 1.0d0
hecMESH%material%mat_VAL (16)= 1002.0d0
hecMESH%material%mat_TEMP (16)= 2.0d0
hecMESH%material%mat_VAL (17)= 1003.0d0
hecMESH%material%mat_TEMP (17)= 3.0d0

hecMESH%material%mat_VAL (18)= 0.0d0
hecMESH%material%mat_TEMP (18)= 0.0d0
hecMESH%material%mat_VAL (19)= 1.0d0
hecMESH%material%mat_TEMP (19)= 1.0d0
hecMESH%material%mat_VAL (20)= 2.0d0
hecMESH%material%mat_TEMP (20)= 2.0d0
hecMESH%material%mat_VAL (21)= 3.0d0
hecMESH%material%mat_TEMP (21)= 3.0d0

hecMESH%material%mat_VAL (22)= 2100.0d0
hecMESH%material%mat_TEMP (22)= 0.0d0
hecMESH%material%mat_VAL (23)= 2101.0d0
hecMESH%material%mat_TEMP (23)= 1.0d0
hecMESH%material%mat_VAL (24)= 2102.0d0
hecMESH%material%mat_TEMP (24)= 2.0d0
hecMESH%material%mat_VAL (25)= 2103.0d0
hecMESH%material%mat_TEMP (25)= 3.0d0
hecMESH%material%mat_VAL (26)= 3103.0d0
hecMESH%material%mat_TEMP (26)= 4.0d0

write (* ,'(a,i10)') '%n_mat_ITEM ', hecMESH%material%n_mat_ITEM
write (* ,'(a,i10)') '%n_mat_subITEM', hecMESH%material%n_mat_subITEM
write (* ,'(a,i10)') '%n_mat_TABLE ', hecMESH%material%n_mat_TABLE

```

end program TEST

1.7.3.8 (7) !NGROUP (M1-7)

Definition of node group

1st Line

!NGROUP, NGRP=<ngrp> [, optional parameter]

Parameter

NGRP	Node group name (mandatory)
GENERATE	Automatic generation of nodes belonging to the node group (omissible)
INPUT	External file name (omissible)

Parameter Name	Parameter Value	Contents
NGRP	<ngrp>	Node group name GENERATE N/A Automatic generation of nodes belonging to the node group
INPUT	<filename>	External file name (omissible), can also be used together with the 2nd line or later

2nd Line or later (when GENERATE is not used)

(2nd Line) nod1, nod2, nod3
(Hereinafter the same)

Parameter Name	Attributions	Contents
nodX	I	Node number belonging to the node group

2nd Line or later (when GENERATE is used)

(2nd Line) nod1, nod2, nod3
(Hereinafter the same)

Parameter Name	Attributions	Contents
nod1	I	First node number in the node group
nod2	I	Last node number in the node group
nod3	I	Node number increment (omissible, number becomes nod3=1 when ommited)

Note:

- Any number of nodes can be inserted in one line. Any number of lines can be inserted until the next option starts.
- It is necessary to define the nodes to be specified before “!NGROUP”.
- The node not defined in the “!NODE” option will be excluded, and a warning message will be displayed.
- When the specified node exists in the same group, it will be ignored and a warning message will be displayed.
- All the nodes belong to the node group named “ALL” (generated automatically).
- One group can be defined by dividing into multiple groups.

Example of Use

!NGROUP, NGRP=NA01

1, 2, 3, 4, 5, 6
101, 102

!NGROUP, NGRP=NA02

101, 102

!NGROUP, NGRP=NA01

501, 505

———— “501 and 505” are added to group “NA01”.

!NGROUP, NGRP=NA02 501, 505	—— "501 and 505" are added to group "NA02".
!NGROUP, NGRP=NA04,GENERATE 301, 309, 2 311, 313	—— "301, 303, 305, 307, 309, 311, 312, 313" are added to group "NA04".

1.7.3.9 (8) !NODE (M1-8)

Definition of node coordinates

1st Line

!NODE [, optional parameter]

Parameter	
SYSTEM	Coordinate system (omissible)
NGRP	Node group name (omissible)
INPUT	External file name (omissible)

Parameter Name	Parameter Value	Contents
SYSTEM	R	Cartesian coordinate system (Default value)
	C	Cylindrical coordinate system
NGRP	<ngrp>	Node group name (omissible)
INPUT	<filename>	External file name (omissible), can also be used together with the 2nd line or later

2nd Line or later

(2nd Line) NODE_ID, Xcoord, Ycoord, Zcoord
(Hereinafter the same)

Parameter Name	Attributions	Contents
NODE_ID	I	Node number
Xcoord	R	X coordinate
Ycoord	R	Y coordinate
Zcoord	R	Z coordinate

Note:

- When node coordinates including the punctuation mark is omitted, the value will become "0.0".
- When an already defined node is redefined, the contents will be updated and a warning message will be displayed.
- The node which is not referred to in "!ELEMENT" will be excluded.
- The node defined in "!ELEMENT" must be defined before "!ELEMENT".

Example of Use

!NODE, NGRP=TEST
1, 0.0, 0.0, 0.5
2, 0.0, 0.0, 1.0
3, 0.0, ,1.5
4,

—— Y coordinate is "0.0"
—— X, Y and Z coordinates are "0.0"

1.7.3.10 (9) !SECTION (M1-9)

Definition of section

1st Line

```
!SECTION, TYPE=<type>, EGRP=<egrp> [ , optional parameter ]
```

Parameter

TYPE	Section type (mandatory)
EGRP	Element group name (mandatory)
MATERIAL	User defined material name (mandatory)
SECOPT	Auxiliary parameter for the element type (omissible, becomes =0 when omitted)
INPUT	External file name (omissible)

Parameter Name	Parameter Value	Contents
TYPE	SOLID	Rod, triangular, quadrilateral, tetrahedral, pentahedral, hexahedral elements
	SHELL	Shell element
	BEAM	Beam element
	INTERFACE	Interface element
EGRP	<egrp>	Element group name
	<material>	Material name defined by user
MATERIAL	= 0:	Not specified, plane stress
	= 1:	Plane strain
SECOPT	= 2:	Axial symmetry
	= 10:	0 + reduced integration
	= 11:	1 + reduced integration
	= 12:	2 + reduced integration
	<filename>	External file name (omissible), can also be used together with the 2nd line or later

2nd Line or later

In the case of [TYPE=SOLID]

(2nd Line) THICKNESS

Parameter Name	Attributions	Contents
THICKNESS	R	Element thickness, cross-sectional area

In the case of “TYPE=SOLID”, the “THICKNESS” can be omitted, and default value (1.0) is inserted.

In the case of [TYPE=SHELL]

(2nd Line) THICKNESS, INTEGPOINTS

Parameter Name	Attributions	Contents
THICKNESS	R	Shell cross section thickness
INTEGPOINTS	I	Integral point in shell cross sectional direction

In the case of [TYPE=BEAM]

(2nd Line) vx, vy, vz, area, Iyy, Iz, Jx

Parameter Name	Attributions	Contents
vx,vy,vz	R	Direction cosine of reference axis
area	R	Area of cross section
Iyy, Izz	R	Second moment of cross section
Jx	R	Torsion constant of cross section

In the case of [TYPE=INTERFACE]

(2nd Line) THICKNESS, GAPCON, GAPRAD1, GAPRAD2

Parameter Name	Attributions	Contents
THICKNESS	R	Cross-sectional thickness
GAPCON	R	Gap heat transfer coefficient (0 when omitted)
GAPRAD1	R	Gap radiant heat transfer factor-1 (0 when omitted)
GAPRAD2	R	Gap radiant heat transfer factor-2 (0 when omitted)

Note:

- When the parameter “TYPE” is not consistent with the element type, an error will occur.
- When there is an element without SECTION information, an error will occur.
- When the section name is duplicated, an error will occur.

Example of Use

```
!SECTION, EGRP=SOLID1, TYPE=SOLID, MATERIAL=STEEL
!SECTION, EGRP=SHELL2, TYPE=SHELL, MATERIAL=STEEL
1.0, 5
```

1.7.3.11 (10) !SGROUP (M1-10)

Definition of surface group

1st Line

!SGROUP, SGRP=<sgrp> [, optional parameter]

Parameter	
SGRP	Surface group name (mandatory)
INPUT	External file name (omissible)

Parameter Name	Parameter Value	Contents
SGRP	<sgrp>	Surface group name
INPUT	<filename>	External file name (omissible), can also be used together with the 2nd line or later

2nd Line or later

(2nd Line) elem1, lsuf1, elem2, lsuf2, elem3, lsuf3, ...
(Hereinafter the same)

Parameter Name	Attributions	Contents
elemX	I	Element number belonging to the surface group

Parameter Name	Attributions	Contents
lsufX	I	Local surface number of the element belonging to the surface group

Note:

- For the element type and surface number, refer to “Chapter 4 Element Library”.
- The surface consists of a combination of (elements and local surface numbers). Any number of surfaces can be inserted in one line. Any number of lines can be inserted until the next option starts. The combination of (elements and local surface numbers) must be in the same line.
- It is necessary to define the element to be specified before “!SGROUP”.
- The element not defined in “!ELEMENT” option will be excluded, and a warning message will be displayed.
- The surface which includes the element not defined in “!ELEMENT” option will be excluded, and a warning message will be displayed.
- The surface where the element type and the surface number are not consistent will be excluded, and a warning message will be displayed.
- One group can be defined by dividing into multiple groups.

Example of Use

```
!SGROUP, SGRP= SUF01
101, 1, 102, 1, 103, 2, 104, 2
201, 1, 202, 1
501, 1
!SGROUP, SGRP= SUF02
101, 2, 102, 2
!SGROUP, SGRP= EA01      "(601,1) and (602 2)" are added to group "SUF01".
601, 1
602, 2
```

Incorrect Example of Use

- Example 1: [When (elements, and local surface numbers) group exists in multiple lines]

```
!SGROUP, SGRP= SUF01
101, 1, 102, 1, 103
1, 104, 1
```

- Example 2: [Local surface numbers and element type are not consistent]

```
!ELEMENT, TYPE= 211, SECTION= A
101, 1, 2, 3
102, 2, 3, 4
...
!SGROUP, SGRP= SUF01
101, 1
101, 2
101, 4      Since a 4th surface does not exist in a triangular element,
               this combination will be disregarded.
```

1.7.3.12 (11) !ZERO (M1-11)

Absolute zero-point

1st Line

!ZERO

Parameter
N/A

2nd Line or later

(2nd Line) ZERO

Parameter Name	Attributions	Contents
ZERO	R	Absolute zero-point

Note:

- Omissible. Becomes “absolute zero-point = 0” when omitted.
- When “!ZERO” is defined multiple times, the contents will be updated and a warning message will be displayed.

Example of Use

```
!ZERO  
-273.16
```

1.7.3.13 (12) !CONTACT PAIR (M1-12)

Definition of contact surface pair used for contact analysis

1st Line

!CONTACT PAIR, NAME=<name> [, optional parameter]

Parameter	
NAME	Contact pair name (mandatory)
TYPE	Type (omissible)

Parameter Name	Parameter Value	Contents
TYPE	NODE-SURF SURF-SURF	Slave surface is the node groupMaster surface is the surface group (default) Both the slave surface and master surface are the surface group

2nd Line or later

(2nd Line or later) SLAVE_GRP, MASTER_GRP
(Hereinafter the same)

Parameter Name	Attributions	Contents
SLAVE_GRP	C	Slave surface node / surface group name
MASTER_GRP	C	Surface group name of the master surface

1.7.3.14 (13) !END (M1-13)

End of mesh data

When this header is displayed, the reading of the mesh data is completed.

1.7.3.15 1st Line

!END

Parameter
N/A

1.7.3.16 2nd Line or later

Parameter
N/A

1.8 Analysis Control Data

1.8.1 Outline of Analysis Control Data

In FrontISTR, an analysis control data file is input to acquire the computing control data, solver control data and post process (visualization) control data as shown in the following figure, in order to implement the analytical calculations.

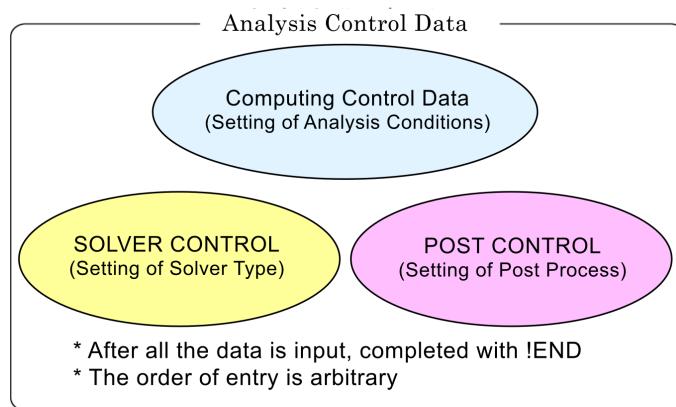


Figure 17: Analysis Control Data

The features of the analysis control data file are as follows.

- This is an ASCII format file based on a free format.
- This file consists of a header which starts with “!” and the data following this.
- The order of description of the header is basically free.
- A “,” is used as a punctuation mark of the data.
- The inside of the file is briefly divided into three zones.
- “!END” is input at the end of the file for completion.

Example of Analysis Control Data

```

#####
##### (1) Computing control data portion #####
#####
### Control File for HEAT solver
!SOLUTION,TYPE=HEAT
!FIXTEMP
XMIN, 0.0
XMAX, 500.0

#####
##### (2) Solver control data portion #####
#####
### Solver Control
!SOLVER,METHOD=1,PRECOND=2,ITERLOG=NO,TIMELOG=NO
100,
  
```

```

2
1.0e-8,1.0,0.0

#####
##### (3) Post control (visualization) data portion #####
#####

### Post Control
!WRITE,RESULT
!WRITE,VISUAL
!VISUAL, method=PSR
!surface_num = 1
!surface 1
!surface_style = 1
!display_method 1
!color_comp_name = TEMPERATURE
!color_subcomp = 1
!output_type = BMP
!x_resolution = 500
!y_resolution = 500
!num_of_lights = 1
!position_of_lights =
-20.0, 5.8, 80.0
!viewpoint = -20.0 10.0 8.0
!up_direction = 0.0 0.0 1.0
!ambient_coef= 0.3
!diffuse_coef= 0.7
!specular_coef= 0.5
!color_mapping_style= 1
!!interval_mapping= -0.01, 0.02
!color_mapping_bar_on = 1
!scale_marking_on = 1
!num_of_scale = 5
!font_size = 1.5
!font_color = 1.0 1.0 1.0
!END

```

1.8.2 Input Rules

The analysis control data consists of a header line, data line and a comment line.

One header is always included in the header line.

Header The header specifies the meaning of the data and the data block in the analysis control data. When the head of the term starts with a “!”, it is considered to be a header.

Header Line The header and the parameter accompanying this are described in this line.

The header line must start with a header. When a parameter is required, a “,” must be used to continue after that. When the parameter takes on a value, use an “=” after the parameter and describe the value after that.
The header line can not be described in more than two lines.

Data Line The data line starts after the header line, and the necessary data is described.

The data lines may be in multiple lines; however, this is determined according to the rules of the data description defined by each header.

There are cases where data lines are not required.

Punctuation A comma “,” is used as a punctuation of the data.

Handling of Blanks Blanks are disregarded.

Name Regarding the characters which can be used for the name, there is the underscore “_”, hyphen “-”, and alphanumeric characters “a – z, A – Z, 0 – 9”; however, the first letter of the name must start with “_”, or an alphabetic character “a – z, A – Z”. There is no distinction between uppercase and lowercase letters, and all letters are internally handled as uppercase letters.

The maximum length of the name is 63 characters.

File Name Regarding the characters which can be used for the file name, there are the underscore “_”, hyphen “-”, period “.”, slash “/”, and the alphanumeric characters “a – z, A – Z, 0 – 9”.

As long as there is no specific description, a path can be included in the file name. Both the relative path and the absolute path can be specified.

The maximum length of the file name is 1,023 characters.

Floating Point Data Exponents are optional. An “E” or “e” character must be added before the exponent. The selection of “E” or “e” is optional.

!!, #, Comment Line Lines starting with “!!” or “#” are considered to be comment lines, and are disregarded. A comment line can be inserted in any position in the file, and there are no restrictions on the number of lines.

!END End of mesh data

When this header is displayed, the reading of the mesh data is completed.

1.8.3 Analysis Control Data

1.8.3.1 Header List of Computing Control Data

In FrontISTR, the following items can be mentioned as the boundary conditions which can be used for the computing control data.

- Distributed load conditions (body force, pressure loading, gravity, centrifugal force)
- Concentrated load conditions
- Heat load
- Single point restriction conditions (SPC conditions)
- Spring boundary conditions
- Contact
- Concentrated heat flux
- Distributed heat flux
- Convective heat transfer boundary
- Radiant heat transfer boundary
- Specified temperature boundary

The same as the mesh data, the !HEADER format is used as the definition method of the above boundary conditions.

The header list of the common control data is shown in the following Table 7.3.1, and the header list for each analysis type is shown in Table 7.3.2.

Table 7.3.1: Control Data Common to All Analysis

Header	Meaning	Remarks	Description No.
!VERSION	Solver version number		1-1
!SOLUTION	Specification of analysis type	Mandatory	1-2
!WRITE,VISUAL	Specification of visualization output		1-3
!WRITE,RESULT	Specification of results output		1-4
!WRITE,LOG	Specification of results output		1-5
!OUTPUT_VIS	Control of visualization output items		1-6
!OUTPUT_RES	Control of results output items		1-7
!RESTART	Control of restarting		1-8
!ECHO	Echo output		1-9
!ORIENTATION	Definition of local coordinate system		1-10
!SECTION	Definition of local coordinate system the section correspondent to		1-11
!INITIAL_CONDITION	Definition of initial condition		1-12
!END	Ending specification of control data		1-13

Table 7.3.2: Control Data for Static Analysis

Header	Meaning	Remarks	Description No.
!STATIC	Static analysis control		2-1
!MATERIAL	Material name		2-2
!ELASTIC	Elastic material physical properties		2-2-1
!PLASTIC	Plastic material physical properties		2-2-2
!HYPERELASTIC	Hyperelastic material physical properties		2-2-3
!VISCOELASTIC	Viscoelastic material physical properties		2-2-4
!CREEP	Creep material physical properties		2-2-5
!DENSITY	Mass density		2-2-6
!EXPANSION_COEFF	Coefficient of linear expansion		2-2-7
!TRS	Tempearture dependent behaviour of viscoelastic material		2-2-8
!FLUID	Flow Condition		2-2-9
!USE_MATERIAL	User defined material		2-2-10
!BOUNDARY	Displacement boundary conditions		2-3
!SPRING	Spring boundary conditions		2-3-1
!CLOAD	Concentrated load		2-4
!DLOAD	Distributed load		2-5
!ULOAD	User defined external load		2-6
!CONTACT_ALGO	Contact analytic algorithm		2-7
!CONTACT	Contact		2-8
!TEMPERATURE	Nodal temperature in thermal stress analysis		2-9
!REFTEMP	Reference temperature in thermal stress analysis		2-10
!STEP	Analysis step control		2-11

Table 7.3.3: Control Data for Eigenvalue Analysis

Header	Meaning	Remarks	Description No.
!EIGEN	Eigenvalue analysis control	Mandatory in eigenvalue analysis	3-1

Table 7.3.4: Control Data for Heat Conduction Analysis

Header	Meaning	Remarks	Description No.
!HEAT	Heat conduction analysis control	Mandatory in heat conduction analysis	4-1
!FIXTEMP	Nodal temperature		4-2
!CFLUX	Concentrated heat flux given to node		4-3
!DFLUX	Distributed heat flux / internal heat generation given to element surface		4-4
!SFLUX	Distributed heat flux by surface group		4-5
!FILM	Heat transfer coefficient given to boundary plain		4-6
!SFILM	Heat transfer coefficient by surface group		4-7
!RADIATE	Radiation factor given to boundary plane		4-8
!SRADIATE	Radiation factor by surface group		4-9
!WELD_LINE	Weld line		4-10

Table 7.3.5: Control Data for Dynamic Analysis

Header	Meaning	Remarks	Description No.
!DYNAMIC	Dynamic analysis control	Mandatory in dynamic analysis	5-1
!VELOCITY	Velocity boundary conditions		5-2
!ACCELERATION	Acceleration boundary conditions		5-3

Header	Meaning	Remarks	Description No.
!COUPLE	Coupled surface definition Required in coupled analysis		5-4
!EIGENREAD	Specification of eigenvalues and eigenvectors	Mandatory in frequency response analysis	5-5
!FLOAD	Definition of concentrated load for frequency response analysis		5-6

In each header, there are data items which comply with the parameter and each header.

Each of the above headers is described in the following with examples of data creation for each analysis type. The description number in the above Table is the number indicated on the right end of the example of the data creation.

1.8.3.2 (1) Control data common to all analyses

1.8.3.2.1 Example of Analysis Control Data

```
### Control File for FISTR
!VERSION                                1-1
 3
!SOLUTION, TYPE=STATIC                  1-2
!WRITE, VISUAL                           1-3
!WRITE, RESULT                           1-4
!ECHO                                    1-9
!BOUNDARY
  FIX, 1, 3, 0.0
!CLOAD
  CL1, 3, -1.0
!END                                     1-12
```

1.8.3.2.2 Description of Header

1-1 !VERSION

Refer to the solver version.

1-2 !SOLUTION, TYPE=STATIC

TYPE=analysis type

1-3 !WRITE, VISUAL

Output of data by visualizer via memory

Outputs the file just by entering

1-4 !WRITE, RESULT

Output of analysis results file

Outputs the file just by entering

1-6 !ECHO

Output of node data, element data and material data to log file

Outputs to the file just by entering

1-8 !END

Indicates the end of control data

1.8.3.3 (2) Static analysis control data

1.8.3.3.1 Example of Static Analysis Control data

```
### Control File for FISTR
!SOLUTION, TYPE=STATIC          1-2
!WRITE, VISUAL                  1-3
!WRITE, RESULT                  1-4
!ECHO                           1-9
!MATERIAL, NAME=M1              2-2
!ELASTIC, TYPE=ISOTROPIC        2-2-1
    210000.0, 0.3
!BOUNDARY
    FIX, 1, 3, 0.0
!SPRING 2-3-1
    200, 1, 0.03
!CLOAD 2-4
    CL1, 3, -1.0
!DLOAD 2-5
    1, P1, 1.0
!TEMPERATURE                     2-9
    1, 10.0
!REFTEMP                         2-10
!STEP, CONVERG=1.E-5, MAXITER=30 2-11
!END                            1-12
```

1.8.3.3.2 Description of Header

- Red figures are the values indicated in the example.
- Alphabetic characters in the 2nd line of the table express the parameter name.

2-1 !STATIC

Setting of static analysis method

2-2 !MATERIAL

Definition of material physical properties

NAME = name of material physical properties

2-2-1 !ELASTIC, TYPE=ISOTROPIC

Definition of elastic substance

TYPE = elastic type

Young's Modulus	Poisson's Ratio
YOUNG_MODULUS	POISSON_RATIO
210000.0	0.3

2-3 !BOUNDARY

Definition of displacement boundary conditions

Node ID or Node Group Name	Start No. of Restricted Freedom	End No. of Restricted Freedom	Restricted Value
NODE_ID	DOF_idS	DOF_idE	Value
FIX,	1,	3,	0.0

2-3-1 !SPRING

Definition of spring boundary conditions

Node ID or Node Group Name	Restricted Degree of Freedom	Spring Constant
NODE_ID	DOF_id	Value
200,	1,	0.03

2-4 !CLOAD

Definition of concentrated load

Node ID or Node Group Name	Degree of Freedom No.	Load Value
NODE_ID	DOF_id	Value
CL1,	3,	-1.0

2-5 !DLOAD

Definition of distributed load

Element ID or Element Group Name	Load Type No.	Load Parameter
ELEMENT_ID	LOAD_type	param
1,	P1,	1.0

2-9 !TEMPERATURE

Specification of nodal temperature used for thermal stress analysis

Node ID or Node Group Name	Temperature
NODE_ID	Temp_Value
1,	10

2-10 !REFTEMP

Definition of reference temperature in thermal stress analysis

2-11 !STEP

Control of nonlinear static analysis (Omissible in the case of linear analysis)

Conver-	No. of Sub	Time	
gence Value	Steps (When AMP exists,	Func-	Name
Judgment	AMP has priority)	tion	(Spec-
Threshold			ified
CONVERG	SUBSTEPS	MAXITER	!AMPLITUDE)
1.0E-05	10	30	AMP

1.8.3.4 (3) Eigenvalue analysis control data

1.8.3.4.1 Example of Eigenvalue Analysis Control Data

```

### Control File for FISTR
!SOLUTION, TYPE=EIGEN           1-2
!WRITE, VISUAL                  1-3
!WRITE, RESULT                  1-4
!ECHO                           1-9
!EIGEN                          3-1
   3, 1.0E-8, 60
!BOUNDARY                        2-3
  FIX, 1, 2, 0.0
!END

```

1.8.3.4.2 Description of Header

Red figures are the values indicated in the example.

3-1 !EIGEN

Parameter settings of eigenvalue analysis

No. of Eigenvalue	Allowance	Max No. of Iterations
NSET	LCZTOL	LCZMAX
3,	1.0E-8,	60

2-3 !BOUNDARY (Same items as in Static Analysis)

Definition of displacement boundary conditions

Node ID or Node Group Name	Start No. of Restricted Degree of Freedom	End No. of Restricted Degree of Freedom	Restricted Value
NODE_ID	DOF_idS	DOF_idE	Value
FIX,	1,	3,	0.0

1.8.3.5 (4) Heat conduction analysis control data

1.8.3.5.1 Example of Heat Conduction Analysis Control Data

```

### Control File for FISTR
!SOLUTION, TYPE=HEAT           1-2
!WRITE, VISUAL                 1-3
!WRITE, RESULT                 1-4
!ECHO                           1-9
!HEAT                           4-1
!FIXTEMP                         4-2
  XMIN, 0.0
  XMAX, 500.0
!CFLUX                           4-3
ALL, 1.0E-3
!DFLUX                           4-4
ALL, S1, 1.0
!SFLUX                           4-5
SURF, 1.0
!FILM                            4-6
  FSURF, F1, 1.0, 800
!SFILM                           4-7
SFSURF, 1.0, 800.0
!RADIATE                         4-8

```

RSURF, R1, 1.0E-9, 800.0
!SRADIATE
RSURF, R1, 1.0E-9, 800.0
!END

4-9

1-12

1.8.3.5.2 Description of Header

Red figures are the values indicated in the example.

4-1 !HEAT

Definition of control data for calculation

!HEAT	
(No data)	—— Steady calculation
!HEAT	
0.0	—— Steady calculation
!HEAT	
10.0, 3600.0	—— Fixed time increment unsteady calculation
!HEAT	
10.0, 3600.0, 1.0	—— Automatic time increment unsteady calculation
!HEAT	
10.0, 3600.0, 1.0, 20.0	—— Automatic time increment unsteady calculation

4-2 !FIXTEMP

Node group name, or node ID and fixed temperature

4-3 !CFLUX

Definition of concentrated heat flux given to node

Node Group Name or Node ID	Heat Flux Value
NODE_GRP_NAME	Value
ALL,	1.0E-3

4-4 !DFLUX

Definition of distributed heat flux and internal heat generation given to surface of element

Element Group Name or Element ID	Load Type No.	Heat Flux Value
ALL,	S1,	1.0

Load Parameter

Load Type No.	Applied Surface	Parameter
BF	Element Overall	Calorific value
S1	Surface No. 1	Heat flux value
S2	Surface No. 2	Heat flux value
S3	Surface No. 3	Heat flux value
S4	Surface No. 4	Heat flux value
S5	Surface No. 5	Heat flux value
S6	Surface No. 6	Heat flux value
S0	Shell surface	Heat flux value

4-5 !SFLUX

Definition of distributed heat flux by surface group

Surface Group Name	Heat Flux Value
SURFACE_GRP_NAME	Value
SURF,	1.0

4-6 !FILM

Definition of heat transfer coefficient given to boundary plane

Element Group Name or Element ID	Load Type No.	Heat Transfer Coefficient	Ambient Temperature
ELEMENT_GRP_NAME FSURF,	LOAD_type F1,	Value 1.0,	Sink 800.0

Load Parameter

Load Type No.	Applied Surface	Parameter
F1	Surface No. 1	Heat transfer coefficient and ambient temperature
F2	Surface No. 2	Heat transfer coefficient and ambient temperature
F3	Surface No. 3	Heat transfer coefficient and ambient temperature
F4	Surface No. 4	Heat transfer coefficient and ambient temperature
F5	Surface No. 5	Heat transfer coefficient and ambient temperature
F6	Surface No. 6	Heat transfer coefficient and ambient temperature
F0	Shell surface	Heat transfer coefficient and ambient temperature

4-7 !SFILM

Definition of heat transfer coefficient by surface group

Surface Group Name	Heat Transfer Rate	Ambient Temperature
SURFACE_GRP_NAME SFSURF,	Value 1.0,	Sink 800.0

4-8 !RADIATE

Definition of radiation factor given to boundary plane

Element Group Name or Element ID	Load Type No.	Radiation Factor	Ambient Temperature
ELEMENT_GRP_NAME RSURF,	LOAD_type R1,	Value 1.0E-9,	Sink 800.0

Load Parameter

Load Type No.	Applied Surface	Parameter
R1	Surface No. 1	Radiation factor and ambient temperature
R2	Surface No. 2	Radiation factor and ambient temperature
R3	Surface No. 3	Radiation factor and ambient temperature
R4	Surface No. 4	Radiation factor and ambient temperature
R5	Surface No. 5	Radiation factor and ambient temperature
R6	Surface No. 6	Radiation factor and ambient temperature
R0	Shell surface	Radiation factor and ambient temperature

4-9 !SRADIATE

Definition of radiation factor by surface group

Surface Group Name	Radiation Factor	Ambient Temperature
SURFACE_GRP_NAME SRSURF,	Value 1.0E-9,	Sink 800.0

1.8.3.6 (5) Dynamic analysis control data

1.8.3.6.1 Example of Dynamic Analysis Control Data

```
### Control File for FISTR
!SOLUTION, TYPE=DYNAMIC                                1-2
!DYNAMIC, TYPE=NONLINEAR                               5-1
  1 , 1
  0.0 , 1.0 , 500 , 1.0000e-5
  0.5 , 0.25
  1 , 1 , 0.0 , 0.0
  100 , 5 , 1
  0 , 0 , 0 , 0 , 0 , 0
!BOUNDARY, AMP=AMP1                                     2-3
  FIX , 1 , 3 , 0.0
!CLOAD, AMP=AMP1                                       2-4
  CL1 , 3 , -1.0
!COUPLE, TYPE=1                                         5-4
  SCOUPLE
!STEP, CONVERG=1.E-6, ITMAX=20                         2-11
!END                                              1-12
```

1.8.3.6.2 Description Header

- Red figures are the values indicated in the example.
- Alphabetic characters in the 2nd line of the table express the parameter name.

5-1 !DYNAMIC

Controlling the linear dynamic analysis

Solution of Equation of Motion	Analysis Types
idx_eqa 11	idx_resp 1

Analysis Start Time	Analysis End Time	Overall No. of STEPS	Time Increment
t_start 0.0	t_end 1.0	n_step 500	t_delta 1.0000e-5

Parameter γ of Newmark- β Method	Parameter β of Newmark- β Method
gamma 0.5	beta 0.25

Type of Mass Matrix	Type of Damping	Parameter R_m of Rayleigh Damping	Parameter of R_k of Rayleigh Damping
idx_mass 1	idx_dmp 1	ray_m 0.0	ray_k 0.0

Resules Output Interval	Monitoring Node ID or Node Group Name	Results Output Interval of Displacement Monitoring
nout 100	node_monit_1 55	nout_monit nout_monit

Output Control Displacement	Output Control Velocity	Output Control Acceleration	Output Control Reaction Force	Output Control Strain	Output Control Stress
iout_list(1) 0	iout_list(2) 0	iout_list(3) 0	iout_list(4) 0	iout_list(5) 0	iout_list(6) 0

2-3 !BOUNDARY (Same items as in Static Analysis)

Definition of displacement boundary conditions

Node ID or Node Group Name	Start No. of Restricted Degree of Freedom	End No. of Restricted Degree of Freedom	Restricted Value
NODE_ID FIX,	DOF_idS 1,	DOF_idE 3,	Value 0.0

2-4 !CLOAD (Same items as in Static Analysis)

Definition of concentrated load

Node ID or Node Group Name	Degree of Freedom No.	Load Value
CL1,	3,	-1.0

5-4 !COUPLE, TYPE=1

Definition of coupled surface

Coupling Surface Group Name
SCOUPLE

2-11 !STEP, CONVERG=1.E-10, ITMAX=20

Control of nonlinear static analysis

(Omissible in the case of linear analysis, and unnecessary for explicit method)

Convergence Value Judgment Threshold (Default: 1.0E-06)	No. of Sub Steps (When AMP exists, AMP has priority)	Max No. of Iterative Calculations
CONVERG 1.0E-10	SUBSTEPS	ITMAX 20

1.8.3.7 (6) Dynamic analysis (Frequency Response Analysis) Control Data

1.8.3.7.1 Example of Dynamic analysis (Frequency Response Analysis)

```

!SOLUTION, TYPE=DYNAMIC          1-2
!DYNAMIC                          5-1
 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                         5-5
 eigen0.log
 1, 5
!FLOAD, LOAD CASE=2               5-6
 _PickedSet5 , 2, 1.0
!FLOAD, LOAD CASE=2
 _PickedSet6 , 2, 1.0

```

1.8.3.7.2 Description of Header

- Red figures are the values indicated in the example.
- Alphabetic characters in the 2nd line of the table express the parameter name.

5-1 !DYNAMIC

Controlling the frequency response analysis

Solution of Equation of Motion	Analysis Types
idx_eqa	idx_resp
11	2

Minimum Frequency	Maximum Frequency	Number of divisions for the frequency range	Frequency to obtain displacement
f_start 14000	f_end 16000	n_freq 20	f_disp 15000.0

Analysis Start Time	Analysis End Time
0.0	6.6e-5

Type of Mass Matrix	Type of Damping	Parameter R_m of Rayleigh Damping	Parameter R_k of Rayleigh Damping
idx_mass 1	idx_dmp 1	ray_m 0.0	ray_k 7.2E-7

Results Output Interval in Time Domain	Visualization Type(1-Mode shapes,2-Time history result at f_disp)	Monitoring Node ID in Frequency Domain
nout 10	vistype 2	nodeout 1

Output ControlDisplacement	Output ControlVelocity	Output ControlAcceleration	Output Controlignored	Output Controlignored	Output Controlignored
iout_list(1)	iout_list(2)	iout_list(3)	iout_list(4)	iout_list(5)	iout_list(6)
1	1	1	1	1	1

5-5 !EIGENREAD

Controlling the input file for frequency response analysis

The name of eigenvalue analysis log
eigenlog_filename
eigen0.log

lowest mode to be used in frequency response analysis	highest mode to be used frequency response analysis
start_mode	end_mode
1	5

5-6 !FLOAD

Defining external forces applied in frequency response analysis

Node ID, Node Group Name or Surface Group	Degree of Freedom No.	Load Value
_PickedSet5	2	1.0

1.8.4 Solver Control Data

1.8.4.1 Example of Solver Control Data

```
### SOLVER CONTROL
!SOLVER, METHOD=CG, PRECOND=1, ITERLOG=YES, TIMELOG=YES      6-1
   10000, 2
   1.0e-8, 1.0, 0.0                                         6-2
```

1.8.4.2 Description of Header

- Red figures are the values indicated in the example.

1.8.4.2.1 6-1 !SOLVER

```
METHOD      = method
            (DIRECT is the direct method,
             in addition there are CG, BiCGSTAB, GMRES, GPBiCG, etc.)
DUMPTYPE    = type of matrix dumping
DUMPEXIT    = whether program exits right after dumping matrix
```

The following parameters will be disregarded when a direct solver is selected in the method.

```
PRECOND     = preconditioner
ITERLOG     = whether solver convergence history is output
TIMELOG     = whether solver computation time is output
SCALING     = whether matrix is scaled so that each diagonal element becomes 1
USEJAD      = whether matrix ordering optimized for vector processors is performed
```

```

MPCMETHOD = method for multipoint constraints
  (1: Penalty method,
   2: MPC-CG method,
   3: Explicit master-slave elimination)
ESTCOND  = frequency for estimating condition number
  (Estimation performed at every specified number of iterations and
   at the last iteration. No estimation when 0 is specified.)

```

6-2

No. of Iterations	Iteration Count of Preconditioning	No. of Krylov Subspaces	No. of Colors for Multi-Color ordering
NIER 10000	iterPREMAX 10000	NREST	NCOLOR_IN

6-3

Truncation Error	Scale Factor for Diagonal Elements when computing Preconditioning Matrix	Not Used
1.0e-8,	1.0,	0.0

1.8.5 Post Process (Visualization) Control Data

An example of the post process (visualization) control data and the contents are shown in the following.

1.8.5.1 Example of Visualization Control Data

Each description number (P1-0, P1-1, etc.) is linked to the number of the detailed descriptions in the following.

- P1- expresses the common data, and P2- expresses the parameter for the purpose of the rendering. In addition, the rendering will become valid only when the output_type=BMP.
- When the surface_style is !surface_style = 2 (isosurface) !surface_style = 3 (user specified curved surface), a separate setting is required. The data is indicated collectively after the common data. (P3- is a description of the isosurface in !surface_style = 2. P4- is a description of the user specified curved surface in !surface_style = 3.)
- The items indicated with two ! like “!!”, will be recognized as a comment and will not affect the analysis.

## Post Control	Description No.
!VISUAL, method=PSR	P1-0
!surface_num = 1	P1-1
!surface 1	P1-2
!surface_style = 1	P1-3
!display_method = 1	P1-4
!color_comp_name = STRESS	P1-5
!colorsubcomp_name	P1-6
!color_comp 7	P1-7
!! color_subcomp = 1	P1-8
!iso_number	P1-9
!specified_color	P1-10
!deform_display_on = 1	P1-11
!deform_comp_name	P1-12
!deform_comp	P1-13
!deform_scale = 9.9e-1	P1-14
!initial_style = 1	P1-15
!deform_style = 3	P1-16
!initial_line_color	P1-17
!deform_line_color	P1-18

!output_type = BMP	P1-19
!x_resolution = 500	P2-1
!y_resolution = 500	P2-2
!num_of_lights = 1	P2-3
!position_of_lights = -20.0, 5.8, 80.0	P2-4
!viewpoint = -20.0 -10.0 5.0	P2-5
!look_at_point	P2-6
!up_direction = 0.0 0.0 1.0	P2-7
!ambient_coef= 0.3	P2-8
!diffuse_coef= 0.7	P2-9
!specular_coef= 0.5	P2-10
!color_mapping_style= 1	P2-11
!!interval_mapping_num	P2-12
!interval_mapping= -0.01, 0.02	P2-13
!rotate_style = 2	P2-14
!rotate_num_of_frames	P2-15
!color_mapping_bar_on = 1	P2-16
!scale_marking_on = 1	P2-17
!num_of_scale = 5	P2-18
!font_size = 1.5	P2-19
!font_color = 1.0 1.0 1.0	P2-20
!background_color	P2-21
!isoline_color	P2-22
!boundary_line_on	P2-23
!color_system_type	P2-24
!fixed_range_on = 1	P2-25
!range_value = -1.E-2, 1.E-2	P2-26

1.8.5.1.1 Common Data List<P1-1 - P1-19>

No.	Keywords	Types	Contents
P1-0	!VISUAL		Specification of the visualization method
P1-1	surface_num		No. of surfaces in one surface rendering
P1-2	surface		Setting of the contents of surface
P1-3	surface_style	integer	Specification of the surface type (Default: 1): Boundary surface2: Isosurface3: Curved surface defined by user based on the equation

No.	Keywords	Types	Contents
P1-4	display_method	integer	Display method (Default: 1) 1. Color code display2. Boundary line display3. Color code and boundary line display4. Display of 1 specified color5. Isopleth line display by classification of color
P1-5	color_comp_name	character(100)	Compatible with parameter name and colormap(Default: 1st parameter name)
P1-6	color_subcomp_name	character(4)	When the parameter is a vector, specifies the component to be displayed. (Default: x)norm: Norm of the vectorx: x componenty: y componentz: z component
P1-7	color_comp	integer	Provides an ID number to the parameter name(Default: 0)
P1-8	color_subcomp	integer	When the degree of freedom of the parameter is 1 or more, specifies the degree of freedom number to be displayed.0: Norm(Default: 1)

No.	Keywords	Types	Contents
P1-9	iso_number	integer	Specifies the number of isopleth lines.(Default:5)
P1-10	specified_color	real	Specified the color when the display_method = 40.0 < specified_color < 1.0
P1-11	!deform_display_on	integer	Specifies the existence of deformation.1: On, 0: Off (Default: 0)
P1-12	!deform_comp_name	character(100)	Specifies the attribution to be adopted when specifying deformation.(Default: Parameter called DISPLACEMENT)
P1-13	!deform_comp	integer	ID number of the parameter when specifying deformation.(Default: 0)
P1-14	!deform_scale	real	Specifies the displacement scale when displaying deformation.Default:Autostandard_scale = 0.1 * sqrt(x_range2 + y_range2 + z_range2) / max_deformuser_defined: real_scale = standard_scale * deform_scale

No.	Keywords	Types	Contents
P1-15	<code>! initial_style</code>	integer	Specifies the type of deformation display.(Default: 1)0: Not specified1: Solid line mesh(Displayed in blue if not specified)2: Gray filled pattern3: Shading (Let the physical attributions respond to the color)4: Dotted line mesh (Displayed in blue if not specified)
P1-16	<code>!deform_style</code>	integer	Specifies the shape display style after the initial deformation.(Default: 4)0: Not specified1: Solid line mesh (Displayed in blue if not specified)2: Gray filled pattern3: Shading (Let the physical attributions respond to the color)4: Dotted line mesh (Displayed in blue if not specified)

No.	Keywords	Types	Contents
P1-17	! initial_line_color	real (3)	Specifies the color when displaying the initial mesh. This includes both the solid lines and dotted lines.(Default: Blue(0.0, 0.0, 1.0))
P1-18	!deform_line_color	real (3)	Specifies the color when displaying the deformed mesh. This includes both the solid lines and dotted lines.(Yellow(1.0, 1.0, 0.0))
P1-19	output_type	character(3)	Specifies the type of output file. (Default: AVS)AVS: UCD Data for AVS(only on object surface)BMP: Image data (BMP format)COMPLETE_AVIS: UCD data for AVSCOMPLETE_REORDER_AVIS: Rearranges the node and element IDSEPARATE_COMPLETE_AVIS: For each decomposed do- mainCOMPLETE_MICROAVS: Outputs the physical value scalarFSTR_FEMAP_NEUTRAL: Neutral file for FEMAP

1.8.5.1.2 Rendering Data List <P2-1 - P2-26>

(Valid only when the output_type=BMP)

	Keywords	Types	Contents
P2-1	x_resolution	integer	Specifies the width of final figure. (Default: 512)
P2-2	y_resolution	integer	Specifies the height of final figure. (Default: 512)
P2-3	num_of_lights	integer	Specifies the number of lights. (Default: 1)
P2-4	position_of_lights	real(:)	Specifies the position of the lights by coordinates.(Default: Directly above front)Specification method !position_of_lights= x, y, z, x, y, z , !position_of_lights=100.0, 200.0, 0.0
P2-5	viewpoint	real(3)	Specifies the viewpoint position by coordinates.(Default: x = (xmin + xmax)/2.0) y = ymin + 1.5 *(ymax - ymin)z = zmin + 1.5 *(zmax - zmin))
P2-6	look_at_point	real(3)	Specifies the look at point position.(Default: Center of data)
P2-7	up_direction	real(3)	Defines the view frame at Viewpoint, look_at_point and up_direction(Default: 0.0, 0.0, 1.0)
P2-8	ambient_coef	real	Specifies the peripheral brightness. (Default: 0.3)
P2-9	diffuse_coef	real	Specifies the intensity of the diffused reflection light by coefficient.(Default: 0.7)

	Keywords	Types	Contents
P2-10	specular_coef	real	Specifies the intensity of specular reflection by coefficient.(Default: 0.6)
P2-11	color_mapping_style	integer	Specifies the color mapping style.(Default: 1)1: Complete linear mapping (Maps overall color in RGB linear)2: Clip linear mapping (Maps from mincolor to maxcolor in the RGB color space)3: Nonlinear color mapping (Partitions all domains into multiple sections, and performs linear mapping for each section)4: Optimum auto adjustment (Performs a statistical process of the data distribution to determine the color mapping)
P2-12	interval_mapping_num	integer	Specifies the number of sections when the color_mapping_style = 3
P2-13	interval_mapping	real(:)	Specifies the section position and color number when the color_mapping_style = 2 or 3.if the color_mapping_style=2;!interval_map the color_mapping_style=3;!interval_map Must be describe in one line.

	Keywords	Types	Contents
P2-14	rotate_style	integer	Specifies the rotating axis of animation.1: Rotates at x-axis.2: Rotates at y-axis.3: Rotates at z axis.4: Particularly, specifies the viewpoint to perform animation. (8 frames)
P2-15	rotate_num_of_frames	integer	Specifies the cycle of animation.(rotate_style = 1, 2, 3)(Default: 8)
P2-16	color_mapping_bar_on	integer	Specifies the existence of color mapping bar.0: off; 1: on; Default: 0
P2-17	scale_marking_on	integer	Specifies whether to display the value on the color mapping bar.0: off; 1: on; Default: 0
P2-18	num_of_scale	integer	Specifies the number of memories of the color bar. (Default: 3)
P2-19	font_size	real	Specifies the font size when displaying the value of the color mapping bar. Range: 1.0-4.0 (Default: 1.0)
P2-20	font_color	real(3)	Specifies the display color when displaying the value of the color mapping bar.(Default: 1.0, 1.0, 1.0 (White))

	Keywords	Types	Contents
P2-21	background_color	real(3)	Specifies the background color. (Default: 0.0, 0.0, 0.0 (Black))
P2-22	isoline_color	real(3)	Specifies the color of the isopleth line. (Default: Same color as the value)
P2-23	boundary_line_on	integer	Specifies whether to display the zone of the data.0: off; 1: on; Default: 0
P2-24	color_system_type	integer	Specifies the color mapping style.(Default: 1)1: (Blue - Red)(in ascending order)2: Rainbow mapping (Ascending order from red to purple)3: (Black - White)(in ascending order)
P2-25	fixed_range_on	integer	Specifies whether to maintain the color mapping style for other time steps.0: off; 1: on; (Default: 0)
P2-26	range_value	real(2)	Specifies the section.

1.8.5.1.3 Data List by Setting Values of `surface_style`

In the case of isosurface (`surface_style=2`)

	Keywords	Types	Contents
P3-1	data_comp_name	character(100)	Provides the name to the attribution of the isosurface.
P3-2	data_subcomp_name	character(4)	When the parameter is a vector, specifies the component to be displayed. (Default: x)norm: Norm of the vectorx: x componenty: y componentz: z component
P3-3	data_comp	integer	Provides an ID number to the parameter name(Default: 0)
P3-4	data_subcomp	integer	When the degree of freedom of the parameter is 1 or more, specifies the degree of freedom number to be displayed.0: Norm(Default: 1)

	Keywords	Types	Contents
P3-5	iso_value	real	Specifies the value of the isosurface.

In the case of a curved surface (surface_syle = 3) specified by the equation of the user

	Keywords	Types	Contents
P4-1	method	integer	Specifies the attribution of the curved surface. (Default: 5)1: Spherical surface2: Ellipse curved surface3: Hyperboloid4: Paraboloid5: General quadric surface
P4-2	point	real(3)	Specifies the coordinates of the center when method = 1, 2, 3, or 4(Default: 0.0, 0.0, 0.0)
P4-3	radius	real	Specifies the radius when method = 1(Default: 1.0)
P4-4	length	real	Specifies the length of the diameter when method = 2, 3, or 4.Note: The length of one diameter in the case the ellipse curved surface is 1.0.
P4-5	coef	real	Specifies the coefficient of a quadric surface when method=5.coef[1]x2 + coef[2]y2 + coef[3]z2 + coef[4]xy + coef[5]xz + coef[6]yz + coef[7]x + coef[8]y + coef[9]z + coef[10]=0Ex: coef=0.0, 0.0, 0.0, 0.0, 0.0, 0.0, 1.0, 0.0, -10.0This means the plain surface of y=10.0

1.8.6 Details of Analysis Control Data Parameters

The details of each parameter explained in item 7.3 are described in the following.

The analysis control data is classified as follow.

1. Common control data
2. Control data for static analysis
3. Control data for eigenvalue analysis
4. Control data for heat conduction analysis
5. Control data for dynamic analysis
6. Solver control data
7. Post process (visualization) control data

1.8.6.1 Common Control Data

1.8.6.1.1 (1) !VERSION (1-1)

Specifies the solver version number. The current version number is 3.

Example of Use

```
!VERSION
 3
```

1.8.6.1.2 (2) !SOLUTION (1-2)

Specifies the type of analysis.

Parameter

```
TYPE=
  STATIC      : Linear static analysis
  NLSTATIC    : Nonlinear static analysis
```

```
HEAT      : Heat conduction analysis
EIGEN     : Eigenvalue analysis
DYNAMIC   : Dynamic analysis
STATICEIGEN: Nonlinear static analysis &rarr; Eigenvalue analysis
ELEMCHECK : Element shape check
```

Example of Use

```
!SOLUTION, TYPE=STATIC
```

1.8.6.1.3 (3) !WRITE, VISUAL (1-3)

Specifies the output data by the visualizer via memory.

Parameter

```
FREQUENCY = step interval of output (Default: 1)
```

Example of Use

```
!WRITE, VISUAL, FREQUENCY=2
```

1.8.6.1.4 (4) !WRITE, RESULT (1-4)

Specifies the output of the analysis results file.

Parameter

```
FREQUENCY = step interval of output (Default:1)
```

Example of Use

```
!WRITE, RESULT, FREQUENCY=2
```

1.8.6.1.5 (5) !WRITE, LOG (1-5)

Specifies the step interval for output to the log file.

Parameter

```
FREQUENCY = step interval of output (Default:1)
```

Example of Use

```
!WRITE, LOG, FREQUENCY=2
```

1.8.6.1.6 (6) !OUTPUT_VIS (1-6)

Output item control of the visualization

!WRITE, VISUAL must be specified

Parameter

N/A

2nd Line or later

```
(2nd line or later) Parameter name, ON/OFF
```

Parameter Names	Physical Values
-----------------	-----------------

The following parameter names can be specified.

Parameter Names	Physical Values
DISP	Displacement (Default output)
ROT	Rotation (Only for 781,761 shell)
REACTION	Reaction force of nodes
NSTRAIN	Strain of nodes
NSTRESS	Stress of nodes (Default output)
NMISES	Mises stress of nodes (Default output)
TH_NSTRAIN	Thermal strain of nodes (Not included)
VEL	Velocity
ACC	Acceleration

Example of Use

```
!OUTPUT_VIS
  NSTRAIN, ON
  NSTRESS, OFF
```

1.8.6.1.7 (7) !OUTPUT_RES (1-7)

Output item control of the result

!WRITE, RESULT must be specified

Parameter

N/A

2nd Line or later

```
(2nd line or later) Parameter name, ON/OFF
```

The following parameter names can be specified.

Parameter Names	Physical Values
DISP	Displacement (Default output)
ROT	Rotation (Only for 781,761 shell)
REACTION	Reaction force of nodes
NSTRAIN	Strain of nodes
NSTRESS	Stress of nodes (Default output)
NMISES	Mises stress of nodes (Default output)
ESTRAIN	Strain of elements
ESTRESS	Stress of elements (Default output)
EMISES	Mises stress of elements (Default output)
ISTRAIN	Strain of integration points
ISTRESS	Stress of integration points
PL_ISTRAIN	Plastic strain of integration points
TH_NSTRAIN	Thermal strain of nodes (Not included)
TH_ESTRAIN	Thermal strain of elements (Not included)
TH_ISTRAIN	Thermal strain of integration points (Not included)
VEL	Velocity
ACC	Acceleration
TEMP	Temperature
PRINC_NSTRESS	Nodal principal stress(Scalar value)
PRINCV_NSTRESS	Nodal principal stress(Vector value)

Parameter Names	Physical Values
PRINC_NSTRAIN	Nodal principal strain(Scalar value)
PRINCV_NSTRAIN	Nodal principal strain(Vector value)
PRINC_ESTRESS	Elemental principal stress(Scalar value)
PRINCV_ESTRESS	Elemental principal stress(Vector value)
PRINC_ESTRAIN	Elemental strain(Scalar value)
PRINCV_ESTRAIN	Elemental strain(Scalar value)
SHELL_LAYER	Output per layer of layered shell element
SHELL_SURFACE	Output of surface information of shell element

Example of Use

```
! OUTPUT_RES
ESTRESS, OFF
ISTRESS, ON
```

1.8.6.1.8 (8) !RESTART (1-8)

Controls the writing of the restart file. When not specified, the restart file can not be written.

Parameter

```
FREQUENCY = n      : step interval of output (Default: 0)
      n > 0    : Output for each n step
      n < 0    : First, reads the restart file, then outputs for each n step
```

Example of Use

```
!RESTART, FREQUENCY=-2
```

1.8.6.1.9 (9) !ECHO (1-9)

Outputs the node data, element data and material data to the log file.

Parameter

N/A

1.8.6.1.10 (10) !ORIENTATION (1-10)

Definition of local coordinate system

Parameter

```
NAME = Name of local coordinate system
DEFINITION = COORDINATES (Default)/NODES
```

1.8.6.1.11 2nd Line of later

- In case of DEFINITION=COORDINATES

```
(2nd line or later) a1, a2, a3, b1, b2, b3, c1, c2, c3
```

- In case of DEFINITION=NODES

```
(2nd line or later) a, b, c
```

Parameter Name	Attributions	Contents
Parameter Name	Attributions	Contents
a1, a2, a3	R	coordinate of point a
b1, b2, b3	R	coordinate of point b
c1, c2, c3	R	coordinate of point c
a,b,c	I	Node ID of a,b,c, respectively

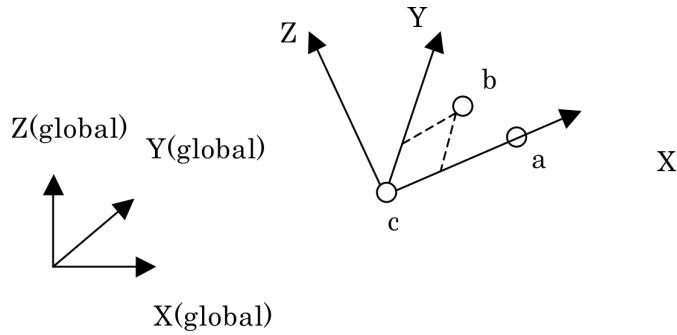


Figure 18: Analysis Control Data

1.8.6.1.12 (11) !SECTION (1-11)

Definition of local coordinate system the section correspondent to.

Parameter

SECNUM = Index of section defined in M1–10 in chapter 6.3.

ORIENTATION = Name of local coordinate system defined in (1–10) above.

2nd Line or later

N/A

1.8.6.1.13 (12) !INITIAL_CONDITION (1-12)

Definition of initial condition

Parameter

TYPE = TEMPERATURE/VELOCITY/ACCELERATION

In case of TYPE = TEMPERATURE

(2nd line) ng1, t1

(3rd line or later) ng2, t2

...

Parameter Name	Attributions	Contents
ng1,ng2, ...	C/I	name of node group/index of node
t1, t2, ...	R	temperature

In case of TYPE= VELOCITY/ACCELERATION

(2nd line) ng1, dof1, v1

(3rd line or later) ng2, dof2, v2

...

Parameter Name	Attributions	Contents
ng1,ng2, ...	C/I	name of node group/index of node
dof1, dof2, ...	I	dof number(1~6)
v1, v2, ...	R	velocity/acceleration

1.8.6.1.14 (13) !END (1-13)

Displays the end of the control data.

Parameter

N/A

1.8.6.2 Control Data for Static Analysis

1.8.6.2.1 (1) !STATIC (2-1)

Performs the static analysis. (Default: omissible)

Parameter

N/A

1.8.6.2.2 (2) !MATERIAL (2-2)

Definition of material physical properties

The definition of the material physical properties is used in a set with the !MATERIAL and the !ELASTICITY, !PLASTICITY and etc. entered next. The !ELASTICITY, !PLASTICITY and etc. entered before !MATERIAL will be disregarded.

Note: When the !MATERIAL is defined in the analysis control data, the !MATERIAL definition in the mesh data will be disregarded. When the !MATERIAL is not defined in the analysis control data, the !MATERIAL definition in the mesh data is used.

Parameter

NAME = Material name

1.8.6.2.3 (3) !ELASTIC (2-2-1)

Definition of elastic material

Parameter

TYPE = ISOTROPIC (Default)/ ORTHOTROPIC / USER
DEPENDENCIES = 0 (Default)/1

2nd Line or later

- In the case of TYPE = ISOTROPIC

(2nd Line) YOUNGS, POISSON, Temperature

Parameter Name	Attributions	Contents
YOUNGS	R	Young's Modulus
POISSON	R	Poisson's Ratio
Temperature	R	Temperature (required when DEPENDENCIES = 1)

- In case of TYPE=ORTHOTROPIC

(2nd Line) E1, E2, E3, 12, 13, 23, G12, G13, G23, Temperature

$$\begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ 2\varepsilon_{12} \\ 2\varepsilon_{23} \\ 2\varepsilon_{31} \end{bmatrix} = \begin{bmatrix} 1/E_1 & -\nu_{12}/E_1 & -\nu_{13}/E_1 & 0 & 0 & 0 \\ & 1/E_2 & -\nu_{23}/E_2 & 0 & 0 & 0 \\ & & 1/E_3 & 0 & 0 & 0 \\ & & & 1/G_{12} & 0 & 0 \\ & & & & 1/G_{23} & 0 \\ & & & & & 1/G_{31} \end{bmatrix} \begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{23} \\ \sigma_{31} \end{bmatrix} \quad (1)$$

Symmetric

- In the case of TYPE=USER

(2nd line – 10th line) v1, v2, v3, v4, v5, v6, v7, v8, v9, v10

1.8.6.2.4 (4) !PLASTIC (2-2-2)

Definition of plastic material

Parameter

YIELD = MISES (Default), Mohr–Coulomb, DRUCKER-PRAGER, USER
 HARDEN = BILINEAR (Default), MULTILINEAR, SWIFT, RAMBERG-OSGOOD,
 KINEMATIC, COMBINED
 DEPENDENCIES = 0 (Default)/1

** 2nd line or later **

In case of YIELD = MISES (Default)

** In case of HARDEN = BILINEAR (Default) **

(2nd line) YIELD0, H

** In case of HARDEN = MULTILINEAR **

(2nd line) YIELD, PSTRAIN, Temperature
 (3rd line) YIELD, PSTRAIN, Temperature

... continues

** In case of HARDEN = SWIFT **

(2nd line) 0, K, n

** In case of HARDEN = RAMBERG-OSGOOD **

(2nd line) 0, D, n

** In case of HARDEN = KINEMATIC **

(2nd line) YIELD0, C

** In case of HARDEN = COMBINED **

(2nd line) YIELD0, H, C

In case of YIELD = Mohr-Coulomb or Drucker-Prager

** In case of HARDEN = BILINEAR(Default) **

(2nd line) c, FAI, H

** In case of HARDEN = MULTILINEAR **

```
(2nd line) FAI
(3rd line) PSTRAIN, c
(4th line) PSTRAIN, c
... continues
```

HARDEN =others will be disregarded, becomes the default (BILINEAR).

Parameter Name	Attribu-tions	Contents
YIELD0	R	Initial yield stress
H	R	Hardening factor
PSTRAIN	R	Plastic strain
YIELD	R	Yield stress
ε_0, K, n	R	$\bar{\sigma} = k (\varepsilon_0 + \bar{\varepsilon})^n$
ε_0, D, n	R	$\varepsilon = \frac{\sigma}{E} + \varepsilon_0 \left(\frac{\sigma}{D}\right)^n$
FAI	R	Internal frictional angle
c	R	Viscosity
C	R	Linear motion hardening factor
Tempearture	R	Temperature (required when DEPENDENCIES = 1)
v1, v2...v10	R	Material constant

** In the case of YIELD= USER **

```
(2nd Line or later) v1, v2, v3, v4, v5, v6, v7, v8, v9, v10
```

Example of Use

```
!PLASTIC, YIELD=MISES, HARDEN=MULTILINEAR, DEPENDENCIES=1
```

```
276.0, 0.0, 20.
296.0, 0.0018, 20.
299.0, 0.0053, 20.
303.0, 0.008, 20.
338.0, 0.0173, 20.
372.0, 0.0271, 20.
400.0, 0.037, 20.
419.0, 0.0471, 20.
437.0, 0.0571, 20.
450.0, 0.0669, 20.
460.0, 0.0767, 20.
469.0, 0.0867, 20.
477.0, 0.0967, 20.
276.0, 0.0, 100.
276.0, 0.0018, 100.
282.0, 0.0053, 100.
295.0, 0.008, 100.
330.0, 0.0173, 100.
370.0, 0.0271, 100.
392.0, 0.037, 100.
410.0, 0.0471, 100.
425.0, 0.0571, 100.
445.0, 0.0669, 100.
450.0, 0.0767, 100.
460.0, 0.0867, 100.
471.0, 0.0967, 100.
128.0, 0.0, 400.
208.0, 0.0018, 400.
243.0, 0.0053, 400.
259.0, 0.008, 400.
```

```

309.0, 0.0173, 400.
340.0, 0.0271, 400.
366.0, 0.037, 400.
382.0, 0.0471, 400.
396.0, 0.0571, 400.
409.0, 0.0669, 400.
417.0, 0.0767, 400.
423.0, 0.0867, 400.
429.0, 0.0967, 400.

```

The work hardening coefficient will be calculated by inserting the data from the above inputdata, regarding the specified temperature or plastic strain. It is necessary to input the same PSTRUAN array for each temperature.

1.8.6.2.5 (5) !HYPERELASTIC (2-2-3)

Definition of hyperelastic material

Parameter

```

TYPE = NEOHOOKE ( Default )
    MOONEY-RIVLIN
    ARRUDA-BOYCE
    USER

```

2nd Line or later

```
TYPE = NEOHOOKE
```

(2nd line) C10, D

Parameter Name	Attributions	Contents
C10	R	Material constant
D	R	Material constant

In case of TYPE = MOONEY-RIVLIN

(2nd line) C10, C01, D

Parameter Name	Attributions	Contents
C10	R	Material constant
C01	R	Material constant
D	R	Material constant

In case of TYPE = ARRUDA-BOYCE

(2nd line) mu, lambda_m, D

Parameter Name	Attributions	Contents
mu	R	Material constant
lambda_m	R	Material constant
D	R	Material constant

In case of TYPE = USER

(2nd line -10th line) v1, v2, v3, v4, v5, v6, v7, v8, v9, v10

1.8.6.2.6 (6) !VISCOELASTIC (2-2-4)

Definition of viscoelastic material

Parameter

DEPENDENCIES = the number of parameters depended upon (Not included)

** 2nd Line or later **

(2nd line) g, t

Parameter Name	Attributions	Contents
g	R	Shear relaxation modulus
t	R	Relaxation time

1.8.6.2.7 (7) !CREEP (2-2-5)

Definition of creep material

Parameter

TYPE = NORTON (Default)

DEPENDENCIES = 0 (Default) / 1

** 2nd Line or later **

(2nd line) A, n, m, Tempearature

Parameter Name	Attributions	Contents
A	R	Material modulus
n	R	Material modulus
m	R	Material modulus
Tempearture	R	Temperature(required when DEPENDENCIES=1)

1.8.6.2.8 (8) !DENSITY (2-2-6)

Definition of mass density

Parameter

DEPENDENCIES = the number of parameters depended upon (Not included)

** 2nd Line or later **

(2nd line) density

Parameter Name	Attributions	Contents
density	R	Mass density

1.8.6.2.9 (9) !EXPANSION_COEFF (2-2-7)

Definition of coefficient of linear expansion

Parameter

TYPE = ISOTROPIC(Default) / ORTHOTROPIC

DEPENDENCIES = 0(Default) / 1

** 2nd Line or later **

In case of TYPE=ISOTROPIC

(2nd line) expansion , Temperature

In case of TYPE=ORTHOTROPIC

(2nd line) 11 , 22 , 33 , Temperature

Parameter Name	Attributions	Contents
expansion	R	Coefficient of thermo expansion
11, 22, 33	R	Coefficient of thermo expansion
Tempearture	R	Temperature (required when DEPENDENCIES = 1)

1.8.6.2.10 (10) !TRS (2-2-8)

Thermorheological Simplicity description on temperature behavior of viscoelastic materials

Parameter

DEFINITION = WLF(Default) /ARRHENUS

** 2nd line or later **

(2nd line) 0, C1, C2

Parameter Name	Attributions	Contents
0	R	Reference temperature
C1, C2	R	Material constants

1.8.6.2.11 (11) !FLUID (2-2-9)

Definition of flow condition

Parameter

TYPE = INCOMP_NEUTONIAN (Default)

** 2nd Line or later **

(2nd line) mu

Parameter Name	Attributions	Contents
mu	R	Viscosity

1.8.6.2.12 (12) !USER/MATERIAL (2-2-10)

Input of user defined material

Parameter

NSTATUS = Specifies the number of state variables of material (Default: 1)

** 2nd line or later **

(2nd line -10th line) v1 , v2 , v3 , v4 , v5 , v6 , v7 , v8 , v9 , v10

1.8.6.2.13 (13) !BOUNDARY (2-3)

Definition of displacement boundary conditions

Parameter

GRPID = Group ID
AMP = Time function name (Specified in !AMPLITUDE, valid in dynamic analysis)
ROT_CENTER = Node number of rotational constraint or node group name.
When specified it, this `!BOUNDARY is recognized as rotational constraint.

** 2nd line or later **

(2nd line) NODE_ID, DOF_idS, DOF_idE, Value

Parameter Name	Attributions	Contents
NODE_ID	I/C	Node ID or node group name
DOF_idS	I	Start No. of restricted degree of freedom
DOF_idE	I	End No. of restricted degree of freedom
Value	R	Restricted value (Default: 0)

Example of Use

!BOUNDARY, GRPID=1
1, 1, 3, 0.0
ALL, 3, 3,

Note: Restricted value is 0.0

1.8.6.2.14 (14) !SPRING (2-3-1)

Definition of spring boundary conditions

Parameter

GRPID = Group ID

** 2nd line or later **

(2nd line) NODE_ID, DOF_id, Value

Parameter Name	Attributions	Contents
NODE_ID	I/C	Node ID or node group name
DOF_id	I	Restricted degree of freedom
Value	R	Spring constant

Example of Use

!SPRING, GRPID=1
1, 1, 0.5

1.8.6.2.15 (15) !CLOAD (2-4)

Definition of concentrated load

Parameter

GRPID = Group ID
AMP = Time function name (Specified in !AMPLITUDE, valid in dynamic analysis)

ROT_CENTER = Node number of rotational constraint or node group name.
When specified it , this `!CLOAD` is recognized as load of torque.

** 2nd line or later **

(2nd line) NODE_ID, DOF_id, Value

Parameters	Attributions	Contents
NODE_ID	I/C	Node ID or node group name
DOF_id	I	Degree of freedom No.
Value	R	Load value

Example of Use

```
!CLOAD, GRPID=1
  1, 1, 1.0e3
  ALL, 3, 10.0
!CLOAD, ROT_CENTER=7, GRPID=1
  TORQUE_NODES, 1, 3
  TORQUE_NODES, 3, -4
```

1.8.6.2.16 (16) !DLOAD (2-5)

Definition of distributed load

Parameter

GRPID = Group ID

AMP = Time Function Name (Specified in !AMPLITUDE, valid in dynamic analysis)

FOLLOW = YES(Default) / NO
(whether pressure load follow deformation , valid in finite displacement analysis)

** 2nd Line or later **

(2nd line) ID_NAME, LOAD_type, param1, param2,...

Parameter Name	Attributions	Contents
ID_NAME	I/C	Surface group name, element group name, or element ID
LOAD_type	C	Load type No.
param*	R	Load parameter (refer to following)

Load Parameters

Load Type No.	Types	No. of Parameters	Parameter Array & Meaning
S	Applies pressure to surface specified in the surface group	1	Pressure value
P0	Applies pressure to shell element	1	Pressure value
PX	Pressure to shell element along X direction	1	Pressure value
PY	Pressure to shell element along Y direction	1	Pressure value
PZ	Pressure to shell element along Z direction	1	Pressure value

Load Type No.	Types	No. of Parameters	Parameter Array & Meaning
P1	Applies pressure to 1st surface	1	Pressure value
P2	Applies pressure to 2nd surface	1	Pressure value
P3	Applies pressure to 3rd surface	1	Pressure value
P4	Applies pressure to 4th surface	1	Pressure value
P5	Applies pressure to 5th surface	1	Pressure value
P6	Applies pressure to 6th surface	1	Pressure value
BX	Body force in X direction	1	Body force value
BY	Body force in Y direction	1	Body force value
BZ	Body force in Z direction	1	Body force value
GRAV	Gravity	4	Gravitaional acceleration, gravity direction cosine
CENT	Centrifugal force	7	Angular velocity, position vector at a point on the rotation axis, vector in the rotating axis direction

Example of Use

```
!DLOAD, GRPID=1
  1, P1, 1.0
  ALL, BX, 1.0
  ALL, GRAV, 9.8, 0.0, 0.0, -1.0
  ALL, CENT, 188.495, 0.0, 0.0, 0.0, 0.0, 1.0
```

1.8.6.2.17 (17) !ULOAD (2-6)

Input of user definition load

Parameter

FILE = file name (Mandatory)

1.8.6.2.18 (18) !CONTACT_ALGO (2-7)

Specification of the contact analysis algorithm

Parameter

TYPE = SLAGRANGE (Lagrange multiplier method)
 ALAGRANGE (Extended Lagrange multiplier method)

1.8.6.2.19 (19) !CONTACT (2-8)

Definition of contact conditions

Parameter

GRPID = Boundary conditions group ID
 INTERACTION = SSLID(Default) / FSLID
 NTOL = Contact normal direction convergence threshold (Default: 1.e-5)
 TTOL = Contact tangential direction convergence threshold (Default: 1.e-3)
 NPENALTY = Contact normal direction Penalty (Default: stiffness matrix 1.e3)
 TPENALTY = Contact tangential direction Penalty (Default: 1.e3)

2nd line or later

(2nd line) PAIR_NAME, fcoef , factor

Parameter Name	Attributions	Contacts
PAIR_NAME	C	Contact pair name (Defined in !CONTACT_PAIR)
fcoef	R	Friction coefficient (Default: 0.0)
factor	R	Friction penalty stiffness

Example of Use

```
!CONTACT_ALGO, TYPE=SLAGRANGE  
!CONTACT, GRPID=1, INTERACTION=FSLID  
    CP1, 0.1, 1.0e+5
```

1.8.6.2.20 (20) !TEMPERATURE (2-9)

Specification of nodal temperature used for thermal stress analysis

Parameter

READRESULT = Number of result steps of heat conduction analysis.
When specified, the temperature is sequentially input from
the results file of the heat conduction analysis,
and the 2nd line and later will be disregarded.
SSTEP = First step number that performs the reading
of the heat conduction analysis results (Default: 1)
INTERVAL = Step interval that performs the reading
of the heat conduction analysis results (Default: 1)

** 2nd line or later **

(2nd line) NODE_ID, Temp_Value

Parameter Name	Attributions	Contents
NODE_ID	I/C	Node ID or node group name
Temp_Value	R	Temperature (Default: 0)

Example of Use

```
!TEMPERATURE  
    1, 10.0  
    2, 120.0  
    3, 330.0  
!TEMPERATURE  
    ALL, 20.0  
!TEMPERATURE, READRESULT=1, SSTEP=1
```

1.8.6.2.21 (21) !REFTEMP (2-10)

Definition of reference temperature in thermal stress analysis

Parameter

N/A

** 2nd line or later **

(2nd line) Value

Parameter Name	Attributions	Contents
Value	R	Reference temperature (Default: 0)

1.8.6.2.22 (22) !STEP (2-11)

Setting of analysis steps

Setting is mandatory in the nonlinear static analysis and nonlinear dynamic analysis.

When this definition is omitted in analyses other than the above, all the boundary conditions will become valid and is calculated in 1 step.

When the material characteristics have viscoelasticity and creep, specify TYPE=VISCO and set the computation time conditions.

Parameter

```
TYPE      = STATIC ( default ) / VISCO
            ( semi - static analysis )
SUBSTEPS = Number of substeps of the boundary conditions
            ( Default: 1 )
CONVERG   = Convergence judgment threshold
            ( Default: 1.0e-6 )
MAXITER  = Maximum number of iterative calculations in nonlinear analysis
            ( Default: 50 )
AMP       = Time function name
            ( specified in !AMPLITUDE )
```

** 2nd line or later **

(2nd line) DTIME, ETIME (specified when TYPE=VISCO)

Parameter Name	Attribution	Contents
DTIME	R	Time increment value (Default: 1)
ETIME	R	End value of time increment in this step (Default: 1)

** 3rd line or later **

```
BOUNDARY, id           GRPID defined in id = !BOUNDARY
LOAD, id               GRPID defined in id = !CLOAD, !DLOAD, !SPRING, !TEMPERATURE
CONTACT, id            GRPID defined in id = !CONTACT
```

Parameter

```
!STEP, CONVERG=1.E-8
    0.1, 1.0
    BOUNDARY, 1
    LOAD, 1
    CONTACT, 1
```

1.8.6.3 Control Data for Eigenvalue Analysis

1.8.6.3.1 (1) !EIGEN (3-1)

Parameter settings of eigenvalue analysis

Parameter

N/A

** 2nd line or later **

(2nd line) NGET, LCZTOL, LCZMAX

Parameter Name	Attributions	Contents
NSET	I	No. of eigenvalue
LCZTOL	R	Allowance (Default: 1.0e-8)
LCZMAX	I	Max No. of iterations (Default: 60)

Example of Use

```
!EIGEN
 3, 1.0e-10, 40
```

1.8.6.4 Control Data for Heat Conduction Analysis

1.8.6.4.1 (1) !HEAT (4-1)

Definition of control data regarding calculation

Parameter

N/A

** 2nd line or later **

(2nd line) DT, ETIME, DTMIN, DELTMX, ITMAX, ESP

Parameter Name	Attributions	Contents
DT	R	Initial time increment 0: Steady calculation > 0: Unsteady calculation
ETIME	R	Unsteady calculation time (mandatory for unsteady calculation)
DTMIN	R	Minimum time increment 0: Fixed time increment > 0: Auto time increment
DELTMX	R	Allowable change in temperature
ITMAX	I	Maximum number of iterations of nonlinear calculation (Default: 20)
EPS	R	Convergence judgment value (Default: 1.0e-6)

Example of Use

```
!HEAT
  (No data)           ----- Steady calculation
!HEAT
  0.0                ----- Steady calculation
!HEAT
  10.0, 3600.0       ----- Fixed time increment unsteady calculation
!HEAT
  10.0, 3600.0, 1.0  ----- Auto time increment unsteady calculation
!HEAT
  10.0, 3600.0, 1.0, 20.0 ----- Auto time increment unsteady calculation
```

1.8.6.4.2 (2) !FIXTEMP (4-2)

Definition of fixed temperature

Parameter

AMP = Flux history table name (specified in !AMPLITUDE)

** 2nd line or later **

(2nd line) NODE_GRP_NAME, Value

Parameter Name	Attributions	Contents
NODE_GRP_NAME	C/I	Node group name or node ID
Value	R	Temperature (Default: 0)

Example of Use

```
!FIXTEMP
  ALL, 20.0
!FIXTEMP, AMP=FTEMP
  ALL, 1.0
```

1.8.6.4.3 (3) !CFLUX (4-3)

Definition of centralized heat flux given to the node

Parameter

AMP = Flux history table name (specified in !AMPLITUDE)

** 2nd line or later **

(2nd line) NODE_GRP_NAME, Value

Parameter Name	Attributions	Contents
NODE_GRP_NAME	C/I	Node group name or node ID
Value	R	Heat flux value

Parameter

```
!CFLUX
  ALL, 1.0E-3
!CFLUX, AMP=FUX1
  ALL, 1.0
```

1.8.6.4.4 (4) !DFLUX (4-4)

Definition of distributed heat flux and internal heat generation given to surface of element

Parameter

AMP = Flux history table name (specified in !AMPLITUDE)

** 2nd line or later **

(2nd line) ELEMENT_GRP_NAME, LOAD_type, Value

Paramater Name	Attributions	Contents
ELEMENT_GRP_NAME	C/I	Element group name or element ID
LOAD_type	C	Load type No.
Value	R	Heat flux value

Parameter

```
!DFLUX
  ALL, S1, 1.0
!DFLUX, AMP=FLUX2
  ALL, S0, 1.0
```

Load Parameters

Load Type No.	Applied Surface	Parameter
BF	Element overall	Calorific value
S1	Surface No. 1	Heat flux value
S3	Surface No. 2	Heat flux value
S4	Surface No. 3	Heat flux value
S5	Surface No. 4	Heat flux value
S6	Surface No. 5	Heat flux value
S2	Surface No. 6	Heat flux value
S3	Shell surface	Heat flux value

1.8.6.4.5 (5) !SFLUX (4-5)

Definition of distributed heat flux by surface group

Parameter

AMP = Flux history table name (specified in !AMPLITUDE)

** 2nd line or later **

(2nd line) SURFACE_GRP_NAME, Value

Parameter Name	Attributions	Contents
SURFACE_GRP_NAME	C	Surface group name
Value	R	Heat flux value

Example of Use

```
!SFLUX
  SURF, 1.0
!SFLUX, AMP=FLUX3
  SURF, 1.0
```

1.8.6.4.6 (6) !FILM (4-6)

Definition of heat transfer coefficient given to the boundary plane

Parameter

AMP1 = Heat transfer coefficient history table name (specified in !AMPLITUDE)
AMP2 = Ambient temperature history table name (specified in !AMPLITUDE)

** 2nd line or later **

(2nd line) ELEMENT_GRP_NAME, LOAD_type, Value, Sink

Parameter Name	Attributions	Contents
ELEMENT_GRP_NAME	C/I	Element group name or element ID
LOAD_type	C	Load type No.

Parameter Name	Attributions	Contents
Value	R	Heat transfer coefficient
Sink	R	Ambient temperature

Example of Use

```
!FILM
  FSURF, F1, 1.0, 800.0
!FILM, AMP1=TFILM
  FSURF, F1, 1.0, 1.0
```

Load Parameters

Load Type No.	Applied Surface	Parameter
F1	Surface No. 1	Heat transfer coefficient and ambient temperature
F2	Surface No. 2	Heat transfer coefficient and ambient temperature
F3	Surface No. 3	Heat transfer coefficient and ambient temperature
F4	Surface No. 4	Heat transfer coefficient and ambient temperature
F5	Surface No. 5	Heat transfer coefficient and ambient temperature
F6	Surface No. 6	Heat transfer coefficient and ambient temperature
F0	Shell Surface	Heat transfer coefficient and ambient temperature

1.8.6.4.7 (7) !SFILM (4-7)

Definition of heat transfer coefficient by surface group

Parameter

AMP1 = Heat transfer coefficient history table name (specified in !AMPLITUDE)
AMP2 = Ambient temperature history table name (specified in !AMPLITUDE)

** 2nd line or later **

```
(2nd line) SURFACE_GRP_NAME, Value, Sink
```

Parameter Name	Attributions	Contents
SURFACE_GRP_NAME	C	Surface group name
Valu	R	Heat Transfer Rate
Sink	R	Ambient Temperature

Example of Use

```
!SFILM
  SFSURF, 1.0, 800.0
!SFILM, AMP1=TSFILM, AMP2=TFILM
  SFSURF, 1.0, 1.0
```

1.8.6.4.8 !RADIADE (4-8)

Definition of radiation factor given to boundary plane

Parameter

AMP1 = Radiation factor history table name (specified in !AMPLITUDE)
AMP2 = Ambient temperature history table name (specified in !AMPLITUDE)

** 2nd line or later **

(2nd line) ELEMENT_GRP_NAME, LOAD_type, Value, Sink

Parameter Name	Attributions	Contents
ELEMENT_GRP_NAME	C/I	Element group name or element ID
LOAD_type	C	Load type No.
Value	R	Radiation factor
Sink	R	Ambient temperature

Example of Use

```
!RADIADE
  RSURF, R1, 1.0E-9, 800.0
!RADIADE, AMP2=TRAD
  RSURF, R1, 1.0E-9, 1.0
```

Load Type No.	Applied Surface	Parameter
R1	Surface No. 1	Radiation factor and ambient temperature
R2	Surface No. 2	Radiation factor and ambient temperature
R3	Surface No. 3	Radiation factor and ambient temperature
R4	Surface No. 4	Radiation factor and ambient temperature
R5	Surface No. 5	Radiation factor and ambient temperature
R6	Surface No. 6	Radiation factor and ambient temperature
R0	Shell Surface	Radiation factor and ambient temperature

1.8.6.4.9 (9) !SRADIATE (4-9)

Definition of radiation factor by surface group

Parameter

AMP1 = Radiation factor history table name (specified in !AMPLITUDE)
AMP2 = Ambient temperature history table name (specified in !AMPLITUDE)

** 2nd line or later **

(2nd line) SURFACE_GRP_NAME, Value, Sink

Parameter Name	Attributions	Contents
SURFACE_GRP_NAME	C	Surface group name
Value	R	Radiation factor
Sink	R	Ambient temperature

Example of Use

```
!SRADIATE
  RSURF, 1.0E-9, 800.0
!SRADIATE, AMP2=TSRAD
  RSURF, 1.0E-9, 1.0
```

1.8.6.4.10 (10) !WELD_LINE (4-10)

Definition of weld line (Linear)

Parameter

N/A

** 2nd line **

(2nd line) I, U, Coef, V

Parameter Name	Attributions	Contents
I	R	Current
U	R	Voltage
Coef	R	Heat input coefficient
V	R	Movement speed of the welding torch

** 3rd line **

(3rd line) EGROUP, XYZ, C1, C2, H, tstart

Parameter Name	Attributions	Contents
EGROUP	C	Element group name for heat input
XYZ	I	Movement direction of welding torch (Degree of freedom No.)
C1	R	Starting point coordinates of welding torch
C2	R	Ending point coordinates of welding torch
H	R	Width of welding torch, inside which thermo energy inputted
tstart	R	Welding start time

1.8.6.5 Control Data for Dynamic Analysis

1.8.6.5.1 (1) DYNAMIC

Dynamic analysis control

Time t for each !AMPLITUDE specified in !BOUNDARY, !CLOAD and !DLOAD must be started from 0.0.

Parameter

TYPE = LINEAR : Linear dynamic analysis
NONLINEAR : Nonlinear dynamic analysis

** 2nd line or later **

(2nd line) idx_eqa, idx_resp

Parameter	Attributions	Contents
idx_eqa	I	Solution of equation of motion (Direct time integration)(Default: 1): Implicit method (Newmark- method)11: Explicit method (Center difference method)
idx_resp	I	Analysis type (Default: 1): Time history response analysis2: Frequency response analysis (Not included)

idx_resp=1 (Time history response analysis)

(3rd line) t_start, t_end, n_step, t_delta

Parameter Name	Attributions	Contents
t_start	R	Analysis start time (Default: 0.0), not used

Parameter Name	Attributions	Contents
t_end	R	Analysis end time (Default: 1.0), not used
n_step	I	Overall No. of steps (Default: 1)
t_delta	R	Time increment (Default: 1.0)

(4th line) gamma , beta

Parameter Name	Attributions	Contents
gamma	R	Parameter of Newmark- method (Default: 0.5)
beta	R	Parameter of Newmark- method (Default: 0.25)

(5th line) idx_mas ,idx_dmp , ray_m ,ray_k

Parameter Name	Attributions	Contents
idx_mas	I	Type of mass matrix (Default: 1):1: Lumped mass matrix2: Consistent mass matrix
idx_dmp	I	1: Rayleigh damping (Default: 1)
ray_m	R	Parameter Rm of Rayleigh damping (Default: 0.0)
ray_k	R	Parameter Rk of Rayleigh damping (Default: 0.0)

(6th line) nout , node_monit_1 , nout_monit

Parameter Name	Attributions	Contents
nout	I	not used
node_monit_1	I	Monitoring node ID (Global) or node group name
nout_monit	I	Results output interval of displacement monitoring(Default: 1)

Note: Regarding the information of the monitoring node specified in this line, the displacement is output to the file <dyna_disp_NID.txt>, where NID is the global ID of the monitoring node, and each line includes the step number, time of the step, NID, u1, u2, and u3 in this order. The velocity and acceleration are also output to <dyna_velo_NID.txt> and <dyna_acce_NID.txt>, respectively, in the same format as the displacement. The nodal strain is output to <dyna_strain_NID.txt> and each line includes the step number, time of the step, NID, e11, e22, e33, e12, e23, and e13 in this order. The nodal stress is output to <dyna_stress_NID.txt> and each line includes the step number, time of the step, NID, s11, s22, s33, s12, s23, s13, and s_mises in this order. When monitoring nodes are specified by a node group, each of the files stated above is separately output for each node. When this output is specified, the kinetic energy, deformation energy and the overall energy of the overall analytic model will also be output to <dyna_energy.txt>.

(7th line) iout_list(1) ,iout_list(2) ,iout_list(3) ,iout_list(4) ,iout_list(5) ,iout_list(6)

Parameter Name	Attributions	Contents
iout_list(1)	I	Displacement output specification (Default: 0):0: Not output, 1: Output
iout_list(2)	I	Velocity output specification (Default: 0):0: Not output, 1: Output
iout_list(3)	I	Acceleration output specification (Default: 0):0: Not output, 1: Output
iout_list(4)	I	Reaction force output specification (Default: 0):0: Not output, 1: Output
iout_list(5)	I	Strain output specification (Default: 0):1: Output2: Output (Node base)3: Output (Element base)

Parameter Name	Attribu-	Contents
iout_list(6)	I	Stress output specification (Default: 0) 0: Not output (Element base and node base) 1: Output 2: Output (Node base) 3: Output (Element base)

Example of Use

```
!DYNAMIC, TYPE=NONLINEAR
1 , 1
0.0 , 1.0 , 500 , 1.0000e-5
0.5 , 0.25
1 , 1 , 0.0 , 0.0
100 , 55 , 1
0 , 0 , 0 , 0 , 0 , 0
```

idx_resp=2 (Frequency response analysis)

(3rd line) f_start , f_end , n_freq , f_disp

Parameter Name	Attributions	Contents
f_start	R	Minimum frequency
f_end	R	Maximum frequency
n_freq	I	Number of divisions for the frequency range
f_disp	R	Frequency to obtain displacement

(4th line) t_start , t_end

Parameter Name	Attributions	Contents
t_start	R	Analysis start time
t_end	R	Analysis end time

(5th line) idx_mas , idx_dmp , ray_m ,ray_k

Parameter Name	Attributions	Contents
idx_mas	I	Type of mass matrix (Default: 1) 1: Lumped mass matrix
idx_dmp	I	1: Rayleigh damping (Default: 1)
ray_m	R	Parameter Rm of Rayleigh damping (Default: 0.0)
ray_k	R	Parameter Rk of Rayleigh damping (Default: 0.0)

(6th line) nout , vistype , nodeout

Parameter Name	Attribu-	Contents
nout	I	Results output interval in time domain
vistype	I	Visuzalization type 1:Mode shapes 2:Time history results at f_disp
nodeout	I	Monitoring NODE ID in frequency domain

(7th line) iout_list(1) ,iout_list(2) ,iout_list(3) ,iout_list(4) ,iout_list(5) ,iout_list(6)

Parameter Name	Attributions	Contents
iout_list(1)	I	Displacement output specification (Default: 0)0: Not output, 1: Output
iout_list(2)	I	Velocity output specification (Default: 0)0: Not output, 1: Output
iout_list(3)	I	Acceleration output specification (Default: 0)0: Not output, 1: Output
iout_list(4)	I	not used
iout_list(5)	I	not used
iout_list(6)	I	not used

Example of Use

```
!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
```

1.8.6.5.2 (2) !VELOCITY (5-2)

Definition of velocity boundary conditions

Parameter

TYPE = INITIAL (Initial velocity boundary conditions)
 = TRANSIT (Time history velocity boundary conditions
 specified in !AMPLITUDE; Default)
AMP = Time function name (specified in !AMPLITUDE)
 Provides the relationship between time t and factor f(t) in !AMPLITUDE.
 The time multiplied by factor f(t) to the following value
 becomes the restrained value of that time
 (which not specified: time and factor relationship becomes f(t) = 1.0).

** 2nd line or later **

(2nd line) NODE_ID, DOF_idS, DOF_idE, Value

Parameter Name	Attributions	Contents
NODE_ID	I/C	Node ID or node group name
DOF_idS	I	Start No. of restricted degree of freedom
DOF_idE	I	End No. of restricted degree of freedom
Value	R	Restricted value (Default: 0)

Example of Use

```
!VELOCITY, TYPE=TRANSIT, AMP=AMP1
1, 1, 1, 0.0
ALL, 3, 3
* Restricted value is 0.0
!VELOCITY, TYPE=INITIAL
1, 3, 3, 1.0
2, 3, 3, 1.0
3, 3, 3, 1.0
```

Note: The velocity boundary conditions are different than the displacement boundary conditions, and the multiple

degrees of freedom can not be defined collectively. Therefore, the same number must be used for DOF_idS and DOF_idE. When the TYPE is INITIAL, AMP becomes invalid.

1.8.6.5.3 (3) !ACCELERATION (5-3)

Definition of acceleration boundary conditions

Parameter

```
TYPE = INITIAL (Initial acceleration boundary conditions)
      = TRANSIT ((Time history acceleration boundary conditions
                   specified in !AMPLITUDE; Default)
AMP = Time function name (specified in !AMPLITUDE)
      Provides the relationship between time t and factor f(t) in !AMPLITUDE.
      The time multiplied by factor f(t) to the following Value
      becomes the restrained value of that time (when not specified:
      time and factor relationship becomes f(t) = 1.0).
```

** 2nd line or later **

(2nd line) NODE_ID, DOF_idS, DOF_idE, Value

Parameter Name	Attributions	Contents
NODE_ID	I/C	Node ID or node group name
DOF_idS	I	Start No. of restricted degree of freedom
DOF_idE	I	End No. of restricted degree of freedom
Value	R	Restricted value (Default: 0)

Example of Use

```
!ACCELERATION, TYPE=TRANSIT, AMP=AMP1
  1, 1, 3, 0.0
  ALL, 3, 3
  i* Restricted value is 0.0
!ACCELERATION, TYPE=INITIAL
  1, 3, 3, 1.0
  2, 3, 3, 1.0
  3, 3, 3, 1.0
```

Note: The acceleration boundary conditions are different than the displacement boundary conditions, and the multiple degrees of freedom can not be defined collectively. Therefore, the same number must be used for DOF_idS and DOF_idE.

When the TYPE is INITIAL, AMP becomes invalid.

1.8.6.5.4 (4) !COUPLE (5-4)

Definition of coupled surface (Used only in coupled analysis)

Parameter

```
TYPE = 1: One-way coupled (FrontISTR starts from receiving data)
      2: One-way coupled (FrontISTR starts from sending data)
      3: Staggered two-way coupled (FrontISTR starts from receiving data)
      4: Staggered Two-way coupled (FrontISTR starts from sending data)
      5: Iterative partitioned two-way coupled (FrontISTR starts from receiving data)
      6: Iterative partitioned two-way coupled (FrontISTR starts from sending data)
```

ISTEP = Step No.

From the beginning of analysis to the step specified here, a linearly increasing function from 0 to 1 is multiplied to the input fluid traction.

After this step , the input fluid traction is directly applied.
 WINDOW => 0 Multiply window function (*) to input fluid traction

(*) $\frac{1}{2}(1 - \cos \frac{2\pi i}{N})$, i : current step, N : no. of steps of current analysis

** 2nd line or later **

(2) COUPLING_SURFACE_ID

Parameter Name	Attributions	Contents
SURFACE_ID	C	Surface group name

Example of Use

```
!COUPLE , TYPE=1
SCOUPLE1
SCOUPLE2
```

1.8.6.5.5 (5) !EIGENREAD (5-5)

Controlling the input file for frequency response analysis

Parameter

N/A

** 2nd line or later **

Parameter Name	Attributions	Contents
eigenlog_filename	C	The name of eigenvalue analysis log

(3rd line) start_mode , end_mode

Parameter Name	Attributions	Contents
start_mode	I	lowest mode to be used in frequency response analysis
end_mode	I	highest mode to be used in frequency response analysis

Example of Use

```
!EIGENREAD
eigen_0.log
1, 5
```

1.8.6.5.6 (6) !FLOAD (5-6)

Defining external forces applied in frequency response analysis

Parameter

LOAD CASE = (1: Real part , 2: Imaginary part)

** 2nd line or later **

(2nd line) NODE_ID, DOF_id, Value

Parameter Name	Attributions	Contents
NODE_ID	I/C	Node ID, node group name or surface group name
DOF_id	I	Degree of freedom No.
Value	R	Load value

Example of Use

```
!FLOAD, LOAD CASE=2
    _PickedSet5 , 2, 1.0
```

1.8.6.6 Solver Control Data

1.8.6.6.1 (1) !SOLVER (6-1)

Control of solver

Mandatory control data

1.8.6.6.2 Parameter

METHOD = Method(CG, BiCGSTAB, GMRES, GPBiCG, DIRECT, DIRECTmkl, MUMPS)
 DIRECT: Direct method other than contact analysis (serial processing only)
 DIRECTmkl: Direct method by Intel MKL in contact analysis (serial processing only)
 MUMPS : Parallel direct method by MUMPS
 When any of direct methods is selected, the data lines will be disregarded.
 In 1D and 2D problems, only CG, DIRECT and MUMPS are valid.
 In shell problems, only DIRECT and MUMPS are valid.
 Thread-parallel computation by OpenMP is available in iterative methods
 for 3D problems.

PRECOND = Preconditioner(1, 2, 3, 5, 10, 11, 12)
 1, 2 : (Block) SSOR
 3 : (Block) Diagonal Scaling
 5 : AMG by multigrid preconditioner package ML (experimental)
 10 : Block ILU(0)
 11 : Block ILU(1)
 12 : Block ILU(2)
 1011 and 12 are available only in 3D problems.
 In thread-parallel computation, SSOR or Diagonal Scaling is recommended.

ITERLOG = Whether solver convergence history is output (YES/NO) (Default: NO)

TIMELOG = Whether solver computation time is output (YES/NO) (Default: NO)

USEJAD = Whether matrix ordering optimized for vector processors are performed
 (YES/NO) (Default: NO)
 Valid only in 3D problems with iterative solvers.

SCALING = Whether matrix is scaled so that each diagonal element becomes 1 (YES/NO)
 (Default: NO)
 Valid only in 3D problems with iterative solvers.

DUMPTYPE = Type of matrix dumping (NONE, MM, CSR, BSR) (Mainly for debugging)
 NONE : no dumping (Default)
 MM : matrix is dumped in Matrix Market format
 CSR : matrix is dumped in Compressed Sparse Row (CSR) format
 BSR : matrix is dumped in Blocked CSR format

DUMPEXIT = Whether the program terminates right after matrix dumping (YES/NO)
 (Default: NO)

MPCMETHOD = Method for multipoint constraints

1: Penalty method

2: MPC-CG method

3: Explicit master-slave elimination (Default)

Valid only in 3D problems with iterative solvers.

(Penalty method is always used for direct solvers and 1D and 2D iterative solvers , and MPC-CG method is always used for 4D and 6D iterative solvers.)

ESTCOND = Frequency of condition number estimation (experimental)

Estimation is performed at every specified number of iterations and at the last iteration . No estimation when 0 is specified .

** 2nd line or later **

(2nd line) NIER, iterPREmax, NREST, NCOLOR_IN

Parameter Name	Attribu-tions	Contents
NIER	I	No. of iterations (Default: 100)
iterPREmax	I	No. of iteration of preconditioning based on Additive Schwarz(Default: 1)(recommended value : 1 for serial computation, parallel computation with Diagonal scaling, and serial/parallel computation of problems with MPC, 2 for other parallel computation)
NREST	I	No. of Krylov subspaces (Default: 10)(Valid only when GMRES is selected as the solution)
NCOLOR_IN	I	No. of Colors for Multi-Color ordering (Default: 10)(Valid only when no. of OpenMP threads >= 2)

(3rd line) RESID, SIGMA_DIAG, SIGMA

Parameter Name	Attribu-tions	Contents
RESID	R	Truncation error (Default: 1.0e-8)
SIGMA_DIAG	R	Scale factor for diagonal elements when computing preconditioning matrix (Default: 1.0)
SIGMA	R	Not used (Default: 0.0)

Example of Use

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

1.8.6.7 Post Process (Visualization) Control Data

1.8.6.7.1 (1) !VISUAL (P1-0)

Specifies the visualization method.

```
METHOD = PSR           : Surface rendering
visual_start_step      : Specification of time step number which starts the visualization
                           (Default: 1)
visual_end_step         : Specification of time step number which ends the visualization pr
                           (Default: All)
visual_interval_step    : Specification of time step interval which performs the visualizati
                           (Default: 1)
```

1.8.6.7.2 (2) !surface_num, !surface, !surface_style (P1-1 - 3)

!surface_num (P1-1)

No. of surfaces in one surface rendering

Ex.: There are four surfaces in Figure 7.4.1, which includes two isosurfaces pressure = 1000.0 and pressure = -1000.0, and two cut end plane surfaces z = -1.0 and z = 1.0.

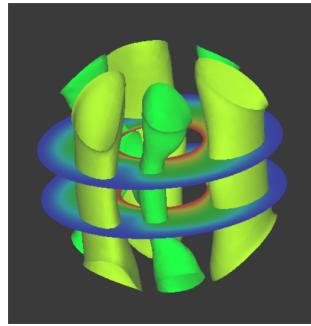


Figure 19: Example of surface_num Setting

Figure 7.4.1: Example of surface_num Setting

!surface (P1-2)

Sets the contents of the surface.

Ex: Then contents of the four surface in Figure 7.4.2 are as follows.

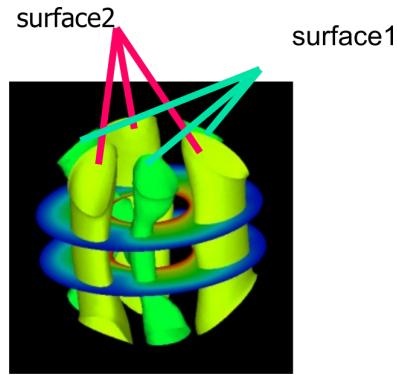


Figure 20: Example of Surface Setting

Figure 7.4.2: Example of Surface Setting

```
!surface_num = 2
!SURFACE
!surface_style = 2
!data_comp_name = press
```

```

!iso_value = 1000.0
!display_method = 4
!specified_color = 0.45
!output_type = BMP
!SURFACE
!surface_style = 2
!data_comp_name = press
!iso_value = -1000.0
!display_method = 4
!specified_color = 0.67

!surface_style (P1-3)

```

Specifies the style of the surface.

1. Boundary plane
2. Isosurface
3. Arbitrary quadric surface $\text{coef}[1]x^2 + \text{coef}[2]y^2 + \text{coef}[3]z^2 + \text{coef}[4]xy + \text{coef}[5]xz + \text{coef}[6]yz + \text{coef}[7]x + \text{coef}[8]y + \text{coef}[9]z + \text{coef}[10]=0$

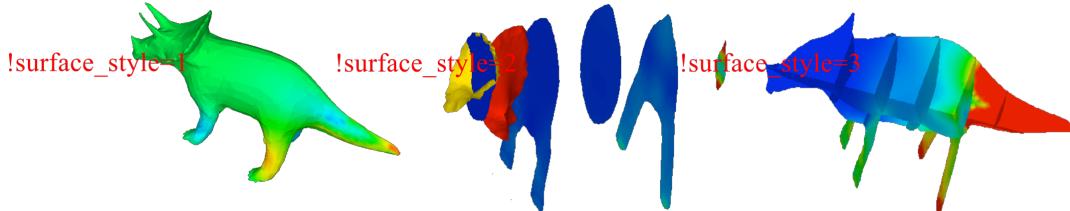


Figure 21: Example of surface_style Setting

Figure 7.4.3: Example of surface_style Setting

1.8.6.7.3 (3) !display_method (P1-4)

Display method (Default: 1)

1. Color code display
2. Boundary line display
3. Color code and boundary line display
4. Display of 1 specified color
5. Isopleth line display by classification of color

Figure 7.4.4: Example of display_method Setting

1.8.6.8 (2) !color_comp_name, !color_comp, !color_subcomp (P1-5, P1-7, P1-8)

Specifies the selections for the color map from the physical values. Provides the names to the necessary physical values and the degree of freedom numbers. Accordingly, the names will be entered for the structure node_label(:) and nn_dof(:) of the results data.

Then you can define which one you hope to map into color by

!color_comp_name (Character string, default: 1st parameter)

Example

```

!color_comp_name = pressure
In static analysis;           = DISPLACEMENT : Specification
                                of the results displacement data
                                = STRAIN      : Specification of strain data
                                = STRESS      : Specification of stress data

```

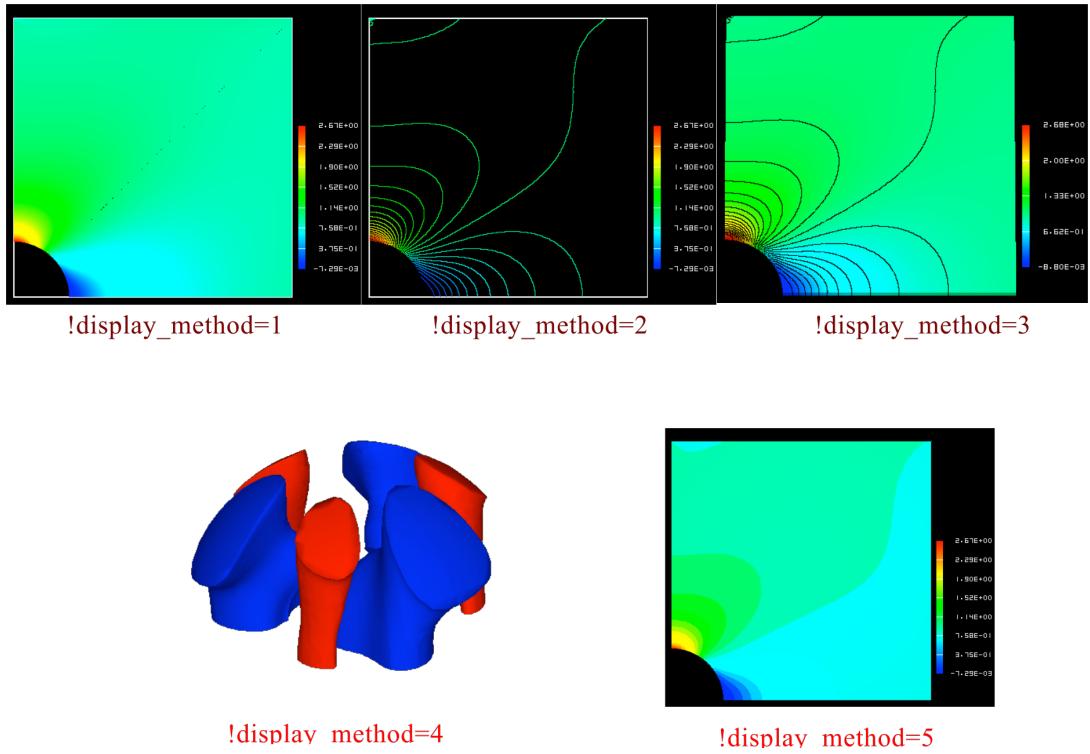


Figure 22: Example of display_method Setting

In heat transfer analysis; = TEMPERATURE : Specification
of the results temperature data

!color_comp (Integer, default: 0)

Physical value ID number (Integers above 0)

Example

!color_comp = 2

This is the specification of the ID number and component name of the results data type; however, this is not included.

!color_subcomp (Integer, default: 0)

When the physical value is 1 degree of freedom or more like the vector quantity, it's the number of the degree of freedom.

Example:

!color_subcomp = 0

When **!color_comp_name=DISPLACEMENT** is specified

1: X Component , 2: Y Component , 3: Z Component

When **!color_comp_name=STRAIN** is specified

1: x , 2: y , 3: z
4: xy , 5: yz , 6: zx

When **!color_comp_name=STRESS** is specified

1: x , 2: y , 3: z
4: xy , 5: yz , 6: zx

When **!color/comp_name=TEMPERATURE** is specified

1: Temperature

In the structural analysis, for example;

Physical Value	Displacement	Strain	Stress
No. of degrees of freedom	3	6	7

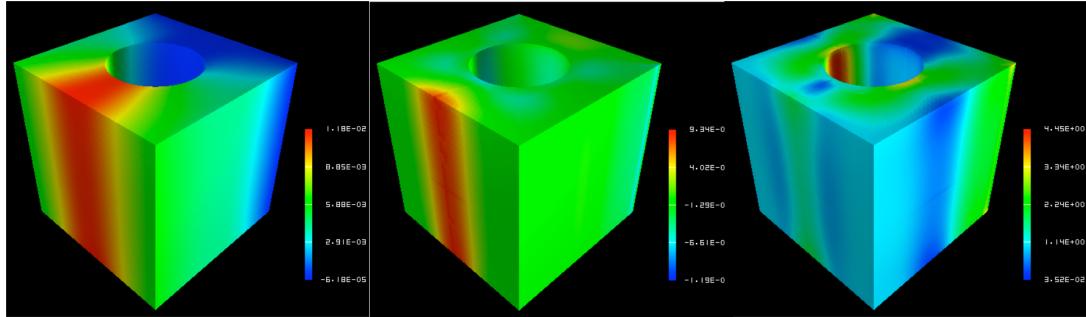


Figure 23: Example of color_comp, color_subcomp and color_comp_name Setting

Figure 7.4.5: Example of color_comp, color_subcomp and color_comp_name Setting

1.8.6.8.1 (5) !isoline_number, !isoline_color (P1-9, P2-22)

When display_method=2,3 or 5

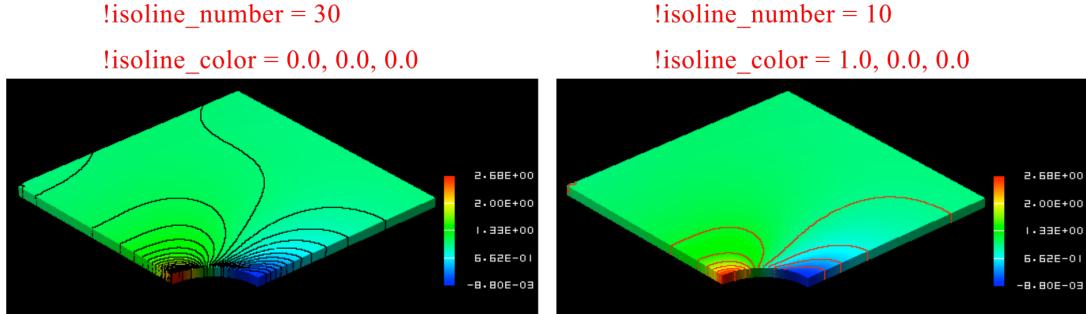


Figure 24: Example of isoline_number and isoline_color Setting

Figure 7.4.6: Example of isoline_number and isoline_color Setting

1.8.6.8.2 (6) !initial_style, !deform_style (P1-15, P1-16)

Specifies the display style of the initial shape and the deformed shape.

0. Not specified
1. Solid line mesh (Displayed in blue if not specified)
2. Gray filled pattern
3. Shading (Let the physical attributions respond to the color)
4. Dotted line mesh (Displayed in blue if not specified)

(7) !deform_scale (P1-14)

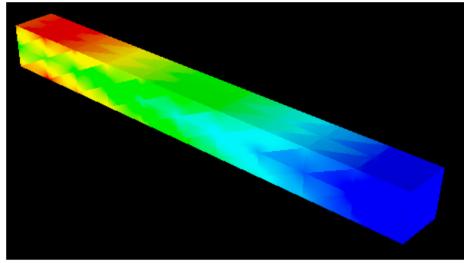
Specifies the displacement scale when displaying deformation.

Default: Auto

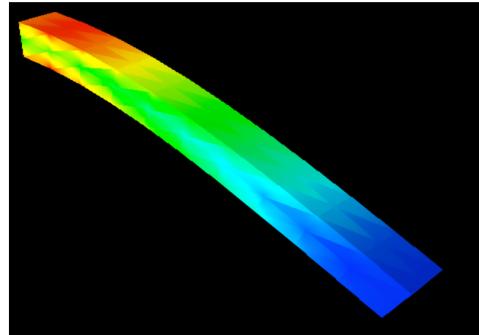
standard_scale = 0.1 * sqrt(x_range2 + y_range2 + z_range2) / max_deform

Figure 7.4.7: Example of display_styles Setting

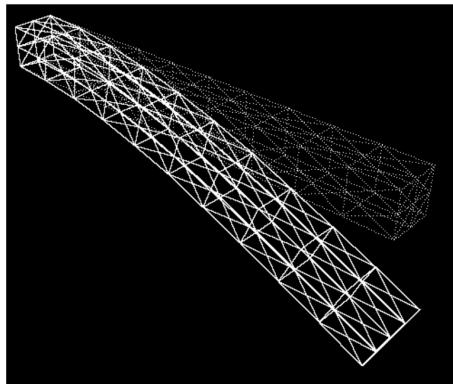
Figure 7.4.8: Example of deform_scale Setting



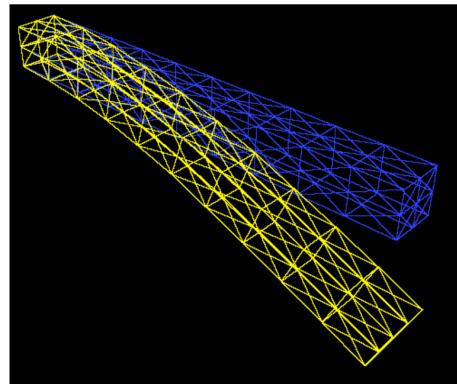
```
!initial_style=2  
!deform_style = 0
```



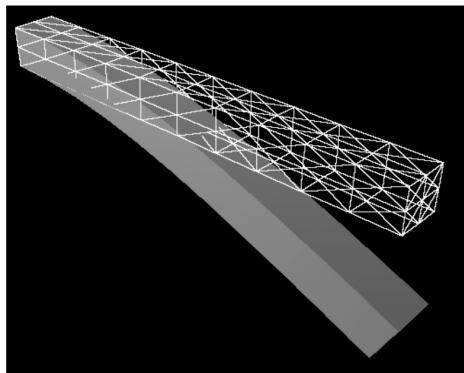
```
!initial_style=0  
!deform_style = 2
```



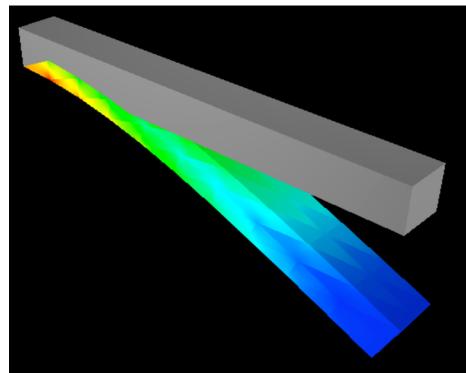
```
!initial_style=4  
!deform_style = 1  
!initial_line_color = 1.0, 1.0, 1.0
```



```
!initial_style=1  
!deform_style = 1  NASTRAN style  
!initial_line_color = default
```

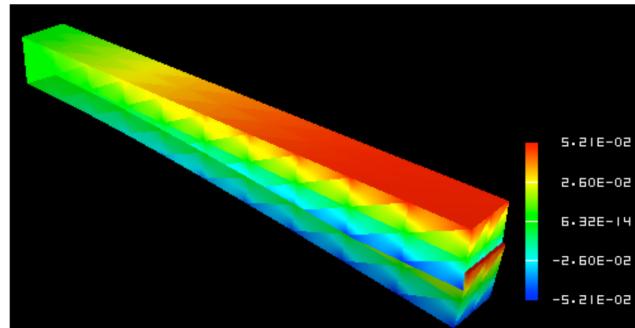


```
!initial_style=1  
!deform_style = 2
```

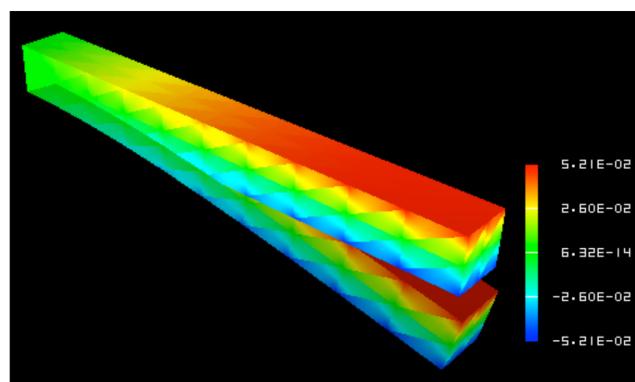


```
!initial_style=2  
!deform_style = 3
```

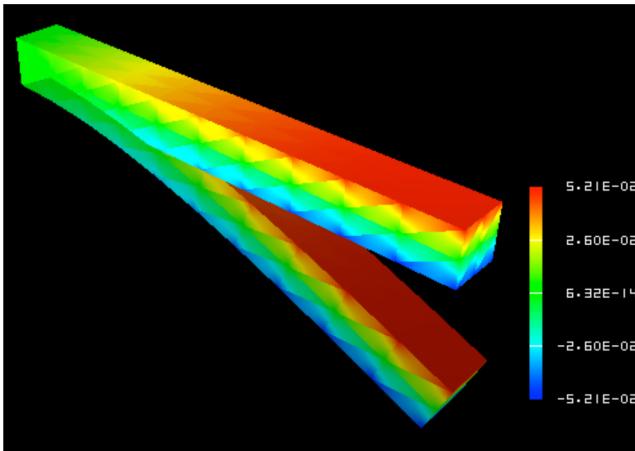
Figure 25: Example of Display Styles Setting



`!deform_scale=1.0`



`!deform_scale=2.0`



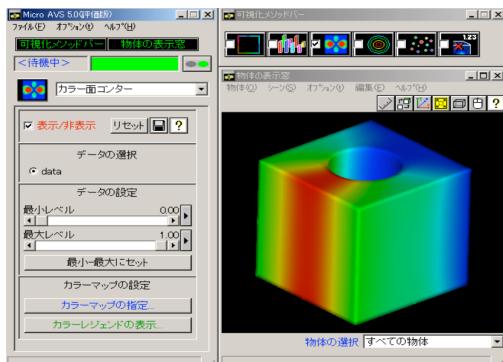
`!deform_scale=4.0`

Figure 26: Example of deform_scale Setting

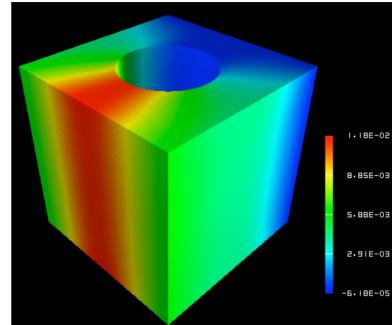
(8) !output_type (P1-19)

Specifies the type of output file. (Default: AVS)

AVS	: UCD data for AVS (only on object surface)
BMP	: Image data (BMP format)
COMPLETE_AVS	: UCD data for AVS
COMPLETE_REORDER_AVS	: Rearranges the node and element ID in the UCD data for AVS
SEPARATE_COMPLETE_AVS	: UCD data for AVS for each decomposed domain
COMPLETE_MICROAVS	: Outputs the physical values in the scalar in the UCD data for AVS
BIN_COMPLETE_AVS	: Outputs COMPLETE_AVS in binary format
FSTR_FEMAP_NEUTRAL	: Neutral file for FEMAP



!output_type = AVS



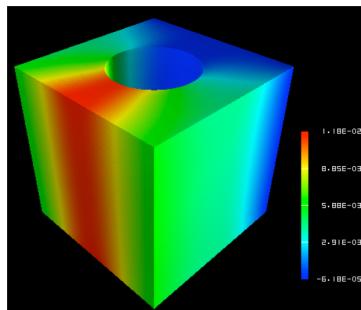
!output_type=BMP

Figure 27: Example of output_type

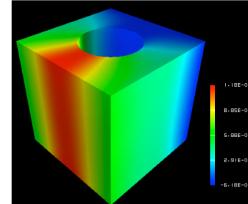
Figure 7.4.9: Example of output_type

1.8.6.8.3 (9) !x_resolution, !y_resolution (P2-1, P2-2)

Specifies the resolution when output_type=BMP



!y_resolution=500
!x_resolution=500



!x_resolution=300
!y_resolution=300

Figure 28: Example of x_resolution and y_resolution Setting

Figure 7.4.10: Example of x_resolution and y_resolution Setting

1.8.6.8.4 (10) !viewpoint, !look_at_point, !up_direction (P2-5, P2-6, P2-7)

viewpoint

Specifies the viewpoint position by coordinates.

Default: $x = (\text{xmin} + \text{xmax})/2.0$, $y = \text{ymin} + 1.5 (\text{ymax} - \text{ymin})$, $z = \text{zmin} + 1.5 (\text{zmax} - \text{zmin})$

look_at_point

Specifies the look at point position.

Default: Center of data

up_direction

Specifies the view frame in viewpoint, look_at_point and up_direction.

default: 0.0 0.0 1.0

View coordinate frame

- Origin: look_at_point
- Z-axis: viewpoint - look_at_point
- X-axis: up_direction \times z axis

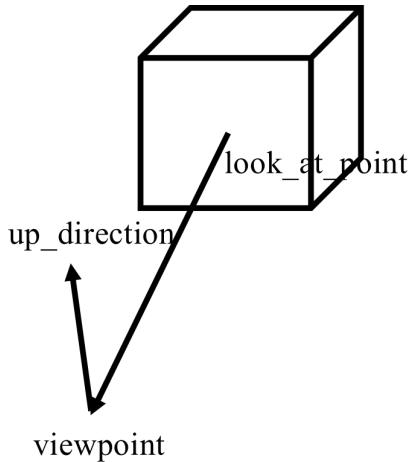


Figure 29: View Frame Determination Method

Figure 7.4.11: View Frame Determination Method

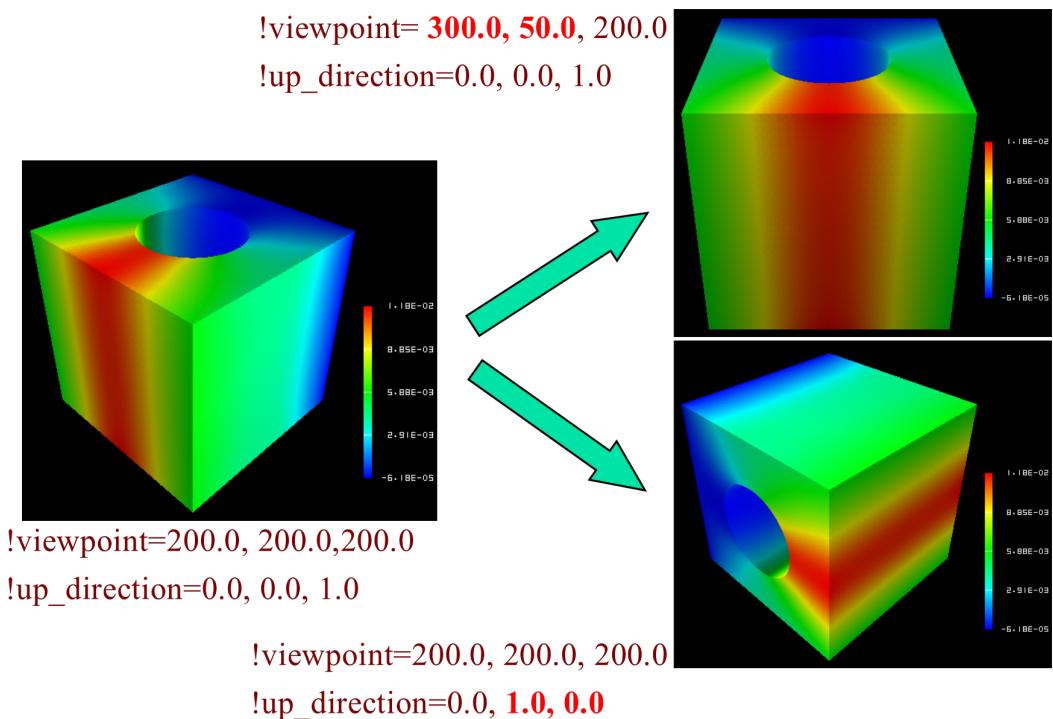


Figure 30: Example of !viewpoint, look_at_point and up_direction Setting

Figure 7.4.12: Example of !viewpoint, look_at_point and up_direction Setting

1.8.6.8.5 (11) !ambient_coef !diffuse_coef !specular_coef (P2-8 P2-9 P2-10)

Coefficient setting of lighting model

When the ambient_coef is increased, information on the 3D depth direction is impaired.

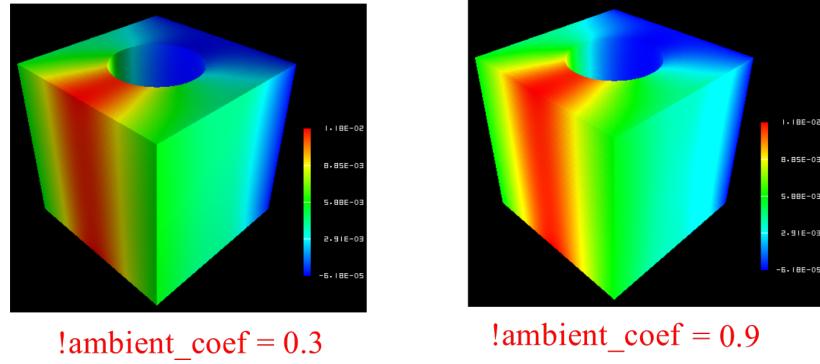


Figure 31: Example of Lighting Model Parameter Setting

1.8.6.8.6 (12) '!color_mapping_bar_on' '!scale_marking_on' '!num_of_scales' (P2-16 P2-17 P2-18)

!color_mapping_bar_on
Specifies whether to display the color mapping bar.
0: off
1: on (Default:
0)
!scale_marking_on
0: off 1: on (:
0)
!num_of_scales (3)

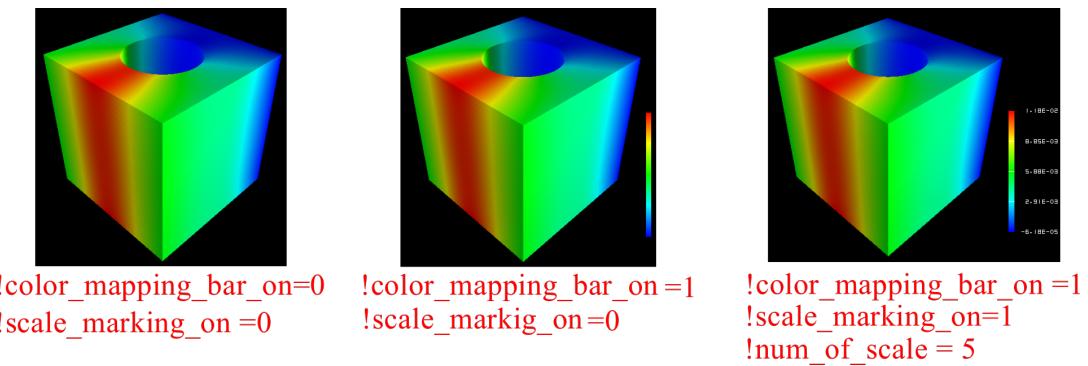
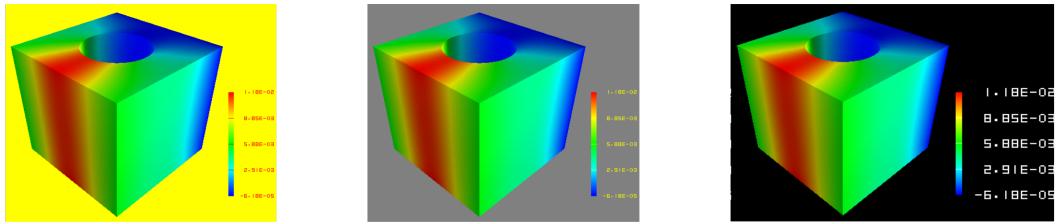


Figure 32: Example of Color Mapping Bar Display

1.8.6.8.7 (13) !font_size !font_color !background_color (P2-19 P2-20 P2-21)

Specifies the background color and character font.

Figure 7.4.15: Example of Background and Font Setting



```
!background_color =1.0,1.0,0.0 !background_color =0.5, 0.5, 0.5 !background_color=0.0, 0.0,0.0
!font_color=1.0, 0.0, 0.0 !font_color=1.0, 1.0, 0.0 !font_color=1.0,1.0, 1.0
!font_size=1.5 !font_size =1.5 !font_size=2.5
```

Figure 33: Example of Background and Font Setting

1.8.6.8.8 (14) !data_comp_name, !data_comp, !data_subcomp (P3-1, P3-3, P3-4)

Specifies the physical values of the isosurface to be visualized when surface_style=2.



```
!data_comp_name=pressure
```

```
!data_comp_name=vorticity
!data_subcomp=3
```

Figure 34: Example of data_comp, data_subcomp and data_comp_name Setting

Figure 7.4.16: Example of data_comp, data_subcomp and data_comp_name Setting

1.8.6.8.9 (15) !method (P4-1)

When specifying the surfaces and cut end, specifies the setting method of the surface.

```
!surface_num =2
!surface
!surface_style=3
!method=5
!coef=0.0, 0.0, 0.0, 0.0, 0.0, 0.0, 0.0, 0.0, 1.0, -0.35
!color_comp_name = temperature
```

Figure 7.4.17: Example of Setting Method

Accordingly, the cut end of the plane surface $z = 0.35$ and $z = -0.35$ will be visualized.

1.9 User Subroutines

An interface is provided for users to expand the FrontISTR functions by programming. These interfaces are basically FORTRAN subroutines which include the subroutine header, and are a description of the I/O parameters and the declaration statement of these parameters. The main portion of the routine must be written by the user.

FrontISTR provides the following user subroutine interfaces.

1.9.1 Input of User Defined Material

When using user defined materials, up to a maximum of 100 material constants defined by the user can be used. As shown in the following, up to 10 values per line and a maximum of 10 lines of material constants can be input in the control data file.

2nd line - 10th line maximaum

v1, v2, v3, v4, v5, v6, v7, v8, v9, v10

1.9.2 Subroutine regarding Elastoplasticity Deformation (uyield.f90)

The subroutines are provided in order to calculate the elastoplasticity stiffness matrix and stress return mapping. When using the user defined yield function, first, it is necessary to set the !PLASTIC, TYPE=USER in the input file, input the required material constants and then, create the subroutines uElastoPlasticMatrix and uBackwardEuler.

1.9.2.1 (1) Calculation subroutines of elastoplasticity stiffness matrix

```
subroutine uElastoPlasticMatrix( matl, stress, istat, fstat, D )
  REAL(KIND=kreal), INTENT(IN) :: matl(:)
  REAL(KIND=kreal), INTENT(IN) :: stress(6)
  INTEGER, INTENT(IN) :: istat
  REAL(KIND=kreal), INTENT(IN) :: fstat(:)
  REAL(KIND=kreal), INTENT(OUT) :: D(:, :)
```

- matl: Array to save the material constants (100 max)
- stress: 2nd Piola-Kirchhoff stress
- istat: Yield state (0: not yielded; 1: yielded)
- fstat: State variable, fstat(1) = plastic strain, fstat(2:7) = back stress (while moving or complex hardening)
- D: Elastoplasticity matrix

1.9.2.2 (2) Return mapping calculation subroutine of stress

```
subroutine uBackwardEuler ( matl, stress, istat, fstat )
  REAL(KIND=kreal), INTENT(IN) :: matl(:)
  REAL(KIND=kreal), INTENT(INOUT) :: stress(6)
  INTEGER, INTENT(INOUT) :: istat
  REAL(KIND=kreal), INTENT(IN) :: fstat(:)
```

- matl: Array to save the material constants (100 max)
- stress: 2nd Piola-Kirchhoff stress acquired by assuming trial stress elastic deformation
- istat: Yield state (0: not yielded; 1: yielded)
- fstat: State variable, fstat(1) = plastic strain, fstat(2:7) = back stress (while moving or complex hardening)

1.9.3 Subroutine regarding Elastic Deformation (uelastic.f90)

The subroutines are provided in order to perform update calculations of the elastic stiffness matrix and stress of the elasticity and hyperelasticity problems. When using the user elasticity, or a hyperelasticity constitutive equation, first, it is necessary to set the !ELASTIC, TYPE=USER or the !HYPERELASTIC, TYPE=USER in the input file, input the required material constants and then, create the subroutines uElasticMatrix and uElasticUpdate.

1.9.3.1 (1) Calculation subroutine of elastic stiffness matrix

```
subroutine uElasticMatrix( matl, strain, D )
  REAL(KIND=kreal), INTENT(IN) :: matl(:)
  REAL(KIND=kreal), INTENT(IN) :: strain(6)
  REAL(KIND=kreal), INTENT(OUT) :: D(6,6)
```

- matl: Array to save the material constants (100 max)
- strain: Green-Lagrange strain
- D: Elastic matrix

1.9.3.2 (2) Calculation subroutine of stress

```
subroutine uElasticUpdate ( matl, strain, stress )
  REAL(KIND=kreal), INTENT(IN) :: matl(:)
  REAL(KIND=kreal), INTENT(IN) :: strain(6)
  REAL(KIND=kreal), INTENT(OUT) :: stress(6)
```

- matl: Array to save the material constants (100 max)
- strain: Green-Lagrange strain
- stress: Stress

1.9.4 Subroutine regarding User Defined Materials (**umat.f**)

The interface of the deformation analysis of general materials is provided irrespective of elastic, hyperelastic and elastoplastic materials.

1.9.4.1 (1) Calculation subroutine of stiffness matrix

```
subroutine uMatlMatrix( mname, matl, ftn, stress, fstat, D, temperature, dtime )
  CHARACTER(len=*) , INTENT(IN) :: mname
  REAL(KIND=kreal), INTENT(IN) :: matl(:)
  REAL(KIND=kreal), INTENT(IN) :: ftn(3,3)
  REAL(KIND=kreal), INTENT(IN) :: stress(6)
  REAL(KIND=kreal), INTENT(IN) :: fstat(:)
  REAL(KIND=kreal), INTENT(OUT):: D(:, :)
  REAL(KIND=kreal), optional :: temperature
  REAL(KIND=kreal), optional :: dtime
```

- mname: Material name
- matl: Array to save the material constants (100 max)
- ftn: Deformation gradient tensor
- stress: 2nd Piola-Kirchhoff stress
- fstat: State variable
- D: Constitutive equation
- temperature: Temperature
- dtime: Time increment

1.9.4.2 (2) Update calculation subroutine of strain and stress

```
subroutine uUpdate( mname, matl, ftn, strain, stress, fstat, temperature, dtime )
  character(len=*) , intent(in) :: mname
  real(KIND=kreal), intent(in) :: matl
  real(kind=kreal), intent(in) :: ftn(3,3)
  real(kind=kreal), intent(inout) :: strain(6)
  real(kind=kreal), intent(inout) :: stress(6)
  real(kind=kreal), intent(inout) :: fstat(:)
  real(KIND=kreal), optional :: temperature
  real(KIND=kreal), optional :: dtime
```

- mname: Material name
- matl: Array to save the material constants (100 max)
- ftn: Deformation gradient tensor
- strain: Strain
- stress: 2nd Piola-Kirchhoff stress

- fstat: State variable
- temperature: Temperature
- dtim: Time increment

1.9.5 Process Subroutine of User Defined External Load (`uload.f`)

An interface is provided to process the external load defined by the user.

In order to use the external load defined by the user, first, numerical structure tULoad is defined in order to define the external load, and the definition is read using the !ULOAD of the input file. Subsequently, the external load is incorporated using the following interfaces.

1.9.5.1 (1) Subroutine for reading external load

```
integer function ureadload( fname )
  character(len=*) , intent(in) :: fname
```

- fname: External file name. The user defined external load is read from this file.

1.9.5.2 (2) Subroutine for incorporating the external load into the overall load vector

```
subroutine uloading( cstep , factor , exForce )
  integer , INTENT(IN) :: cstep
  REAL(KIND=kreal) , INTENT(IN) :: factor
  REAL(KIND=kreal) , INTENT(INOUT) :: exForce(:)
```

- cstep: Current number of analysis steps
- factor: Load factor of current step
- exForce: Overall load vector

1.9.5.3 (3) Calculation subroutine of residual stress

```
subroutine uResidual( cstep , factor , residual )
  integer , INTENT(IN) :: cstep
  REAL(KIND=kreal) , INTENT(IN) :: factor
  REAL(KIND=kreal) , INTENT(INOUT) :: residual(:)
```

- cstep: Current number of analysis steps
- factor: Load factor of current step
- residual: Overall residual stress vector