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

CISS Free Software

FrontISTR

Ver. 3.5

User's Manual

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

Contents

1. Introduction 1
1.1 Position of this Manual 1
1.2 Purpose of this Manual 1
2. Finite Element Method Analysis Theory 2
2.1 Infinitesimal Deformation Linear Elasticity Static Analysis 2
2.1.1 Basic Equation 2
2.1.2 Principle of Virtual Work 3
2.1.3 Formulation 4
2.2 Nonlinear Static Analysis Method 5
2.2.1 Geometric Nonlinear Analysis Method 6
2.2.1.1 Decomposition of Increments of Virtual Work Equation 6
2.2.1.2 Principle of Virtual Work 6
2.2.1.3 Formulation of Total Lagrange Method 7
2.2.1.4 Formulation of Updated Lagrange Method 9
2.2.2 Material Nonlinear Analysis Method 10
2.2.2.1 Hyperelastic Material 10
2.2.2.2 Elastoplastic Material 11
2.2.2.3 Viscoelastic Material 14
2.2.2.4 Creep Material 14
2.2.3 Contact Analysis Method 16
2.3 Eigenvalue Analysis 17
2.3.1 Generalized Eigenvalue Problem 17
2.3.2 Problem Settings 18
2.3.3 Shifted Inverse Iteration Method 18
2.3.4 Algorithm to Solve Eigenvalues 19
2.3.5 Lanczos Method 19
2.3.6 Geometric Meaning in the Lanczos Method 19
2.3.7 Tridiagonalization 21
2.4 Heat Conduction Analysis 22
2.4.1 Basic Equation 22
2.4.2 Discretization 23
2.5 Dynamic Analysis Method 26
2.5.1 Formulation of Implicit Method 26
(1) Handling of Mass Term 27
(2) Handling of Attenuation Term 27
2.5.2 Formulation of Explicit Method 27

2.6 Frequency Response Analysis 29
2.6.1 Formulation 29
3. Analysis Flow and Input/Output File 32
3.1 Analysis Flow 32
3.2 Overall Control Data 33
3.3 Mesh Data 33
3.4 Analysis Control Data 34
3.5 Output File 35
3.6 Execution Procedure 35
(1) Preparation of FrontISTR 35
(2) Preparation of Input Files 35
(3) Execution of Single Domain Analysis 36
(4) Parallel Execution on Linux 36
(5) Parallel Execution on Windows 36
(6) Excution of Parallel Domain Contact Analysis 36
3.7 Restrictions at Time of Execution 37
4. Element Library and Material Data 39
4.1 Element Library 39
4.2 Material Data 48
4.2.1 Elastic Static Analysis, Linear Dynamic Analysis and Eigenvalue Analysis 48
4.2.2 Heat Conduction Analysis 49
(1) In the case of link, plane surface and solid element 49
(2) In the case of an interface element 50
(3) In the case of a shell element 50
4.2.3 Nonlinear Static Analysis 53
5. Overall Control Data 54
5.1 Outline of Overall Control Data 54
5.2 Input Rules 54
5.3 Header List 55
(1) !CONTROL 56
(2) !MESH 57
(3) !RESTRAT 58
(4) !RESULT 59
(5) !SUBDIR 60
6. Single Domain Mesh Data 61
6.1 Outline of Single Mesh Data 61
6.2 Input Rules 61
6.3 Header List of Single Domain Mesh Data 63

(1) !AMPLITUDE (M1-1) 65
(2) !EGROUP (M1-2) 66
(3) !ELEMENT (M1-3) 68
(4) !EQUATION (1-4) 70
(5) !HEADER (M1-5) 72
(6) !INITIAL CONDITION (M1-6) 73
(7) !MATERIAL (M1-7) 74
(8) !NGROUP (M1-8) 83
(9) !NODE (M1-9) 85
(10) !SECTION (M1-10) 87
(11) !SGROUP (M1-11) 90
(12) !ZERO (M1-12) 92
(13) !CONTACT PAIR (M1-13) 93
(14) !END (M1-14) 94
7. Analysis Control Data 95
7.1 Outline of Analysis Control Data 95
7.2 Input Rules 97
7.3 Analysis Control Data 99
7.3.1 Header List of Computing Control Data 99
(1) Control data common to all analyses 102
(2) Static analysis control data 103
(3) Eigenvalue analysis control data 105
(4) Heat conduction analysis control data 106
(5) Dynamic analysis control data 109
(6) Dynamic analysis (Frequency Response Analysis) Control Data 111
7.3.2 Solver Control Data 114
7.3.3 Post Process (Visualization) Control Data 115
7.4 Details of Analysis Control Data Parameters 123
7.4.1 Common Control Data 123
(1) !VERSION (1-1) 123
(2) !SOLUTION (1-2) 123
(3) !WRITE, VISUAL (1-3) 123
(4) !WRITE, RESULT (1-4) 124
(5) !WRITE, LOG (1-5) 124
(6) !OUTPUT_VIS (1-6) 124
(7) !OUTPUT_RES (1-7) 125
(8) !RESTART (1-8) 126
(9) !ECHO (1-9) 126

(10) !ORIENTATION (1-10) 126
(11) !SECTION (1-11) 127
(12) !END (1-12) 127
7.4.2 Control Data for Static Analysis 127
(1) !STATIC (2-1) 127
(2) !MATERIAL (2-2) 128
(3) !ELASTIC (2-2-1) 128
(4) !PLASTIC (2-2-2) 129
(5) !HYPERELASTIC (2-2-3) 131
(6) !VISCOELASTIC (2-2-4) 132
(7) !CREEP (2-2-5) 132
(8) !DENSITY (2-2-6) 133
(9) !EXPANSION_COEFF (2-2-7) 133
(10) !USER_MATERIAL (2-2-8) 134
(11) !BOUNDARY (2-3) 134
(12) !SPRING (2-3-1) 134
(13) !CLOAD (2-4) 135
(14) !DLOAD (2-5) 135
(15) !ULOAD (2-6) 137
(16) !CONTACT_ALGO (2-7) 137
(17) !CONTACT (2-8) 137
(18) !TEMPERATURE (2-9) 138
(19) !REFTEMP (2-10) 138
(20) !STEP (2-11) 139
(21) !TRS (2-12) 140
7.4.3 Control Data for Eigenvalue Analysis 140
(1) !EIGEN (3-1) 140
7.4.4 Control Data for Heat Conduction Analysis 140
(1) !HEAT (4-1) 140
(2) !FIXTEMP (4-2) 141
(3) !CFLUX (4-3) 142
(4) !DFLUX (4-4) 142
(5) !SFLUX (4-5) 143
(6) !FILM (4-6) 144
(7) !SFILM (4-7) 144
(8) !RADIATE (4-8) 145
(9) !SRADIATE (4-9) 146
(10) !WELD_LINE (4-10) 146

7.4.5 Control Data for Dynamic Analysis 147
(1) !DYNAMIC (5-1) 147
(2) !VELOCITY (5-2) 151
(3) !ACCELERATION (5-3) 152
(4) !COUPLE (5-4) 153
(5) !EIGENREAD(5-5) 153
(6) !FLOAD(5-6) 154
7.4.6 Solver Control Data 155
(1) !SOLVER (6-1) 155
7.4.7 Post Process (Visualization) Control Data 156
(1) !VISUAL (P1-0) 156
(2) !surface_num, !surface, !surface_style (P1-1～3) 157
(3) !display_method (P1-4) 159
(4) !color_comp_name !color_comp !color_subcomp (P1-5 P1-7 P1-8) 159
(5) !isoline_number !isoline_color (P1-9 P2-22) 161
(6) !initial_style !deform_style (P1-15 P1-16) 161
(7) !deform_scale (P1-14) 161
(8) !output_type (P1-19) 164
(9) !x_resolution !y_resolution (P2-1 P2-2) 164
(10) !viewpoint !look_at_point !up_direction (P2-5 P2-6 P2-7) 165
(11) !ambient_coef !diffuse_coef !specular_coef (P2-8 P2-9 P2-10) 166
(12) !color_mapping_bar_on !scale_marking_on !num_of_scales (P2-16 P2-17 P2-18) 167
(13) !font_size !font_color !backgroud_color (P2-19 P2-20 P2-21) 167
(14) !data_comp_name !data_comp !data_subcomp (P3-1 P3-3 P3-4) 168
(15) !method (P4-1) 168
8. User Subroutines 169
8.1 Input of User Defined Material 169
8.2 Subroutine regarding Elastoplasticity Deformation (Uyield.F90) 169
8.3 Subroutine regarding Elastic Deformation (Uelastic.F90) 170
8.4 Subroutine regarding User Defined Materials (umat.f) 171
8.5 Process Subroutine of User Defined External Load (uload.f) 172
9. Example Verification 173
9.1 Verification by Simple Geometric Model 173
(1) Elastic static analysis 173
(2) Nonlinear static analysis 179
(3) Eigenvalue analysis 185
(4) Heat conduction analysis 189

(5) Linear dynamic analysis 195
(6) Frequency Response Analysis 200
9.2 Example of Actual Model for Elastic Static Analysis 202
9.2.1 Analysis Model 202
9.2.2 Analysis Results 205
9.2.2.1 Example of Analysis Results 205
9.2.2.2 Verification Results of Analysis Performance by Verification Example EX02. 207
9.2.2.3 Comparison of Computing Time by Verification Example EX01A 209
9.3 Example of Actual Model for Eigenvalue Analysis 210
9.3.1 Analysis Model 210
9.3.2 Analysis Results 213
(1) EX06 Turbine blade 213
(2) EX07 Turbine rotor 214
(3) EX08 Spring 215
(4) EX09 Cylindrical shell 216
(5) EX10A Wine glass 217
9.4 Example of Actual Model for Heat Conduction Analysis 218
9.4.1 Analysis Model 218
9.4.2 Analysis Results 221
9.5 Example of Actual Model for Linear Dynamic Analysis 223
9.5.1 Analysis Model 223
9.5.2 Analysis Results 224

1. Introduction

1.1 Position of this Manual

This manual describes the data input method regarding the analysis range applicable to FrontISTR, and the execution procedures of FrontISTR.

1.2 Purpose of this Manual

The purpose of this manual is to describe the basic contents of the data structure peculiar to the programs and the analysis functions, in order for the user to execute FrontISTR. Regarding the analysis execution control in FrontISTR, it is necessary to specify the overall control data and computing control data. Moreover, analysis of the mesh data is executed by inputting the distributed mesh file. The details of the relationships between the input methods and input data of these control data are described from the following Chapter.

2. Finite Element Method Analysis Theory

Chapter 2 describes the analysis method by the finite element method (FEM) used in this development code. Regarding the stress analysis method of solids, the infinitesimal deformation linear elasticity static analysis method is described first, and the geometric nonlinear analysis method and the 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.

2.1 Infinitesimal Deformation Linear Elasticity Static Analysis

The formulation for the elastic static analysis based on the infinitesimal deformation theory is described in this section. The linear elasticity is assumed as the stress and strain relationship.

2.1.1 Basic Equation

The equilibrium equation of solid mechanics, dynamic boundary conditions and the geometric boundary conditions (basic boundary conditions) are given by the following equation (Refer to Figure 2.1.1).

$$\nabla \bullet \boldsymbol{\sigma} + \bar{\mathbf{b}} = 0 \quad in\, V \tag{2.1.1}$$

$$\boldsymbol{\sigma} \bullet \mathbf{n} = \bar{\mathbf{t}} \quad on\, S_t \tag{2.1.2}$$

$$\mathbf{u} = \bar{\mathbf{u}} \quad on\, S_u \tag{2.1.3}$$

Herein, $\boldsymbol{\sigma}$ is the stress, $\bar{\mathbf{t}}$ is the surface force, $\bar{\mathbf{b}}$ is the body force, and S_t expresses the dynamic boundary and S_u expresses the geometric boundary.

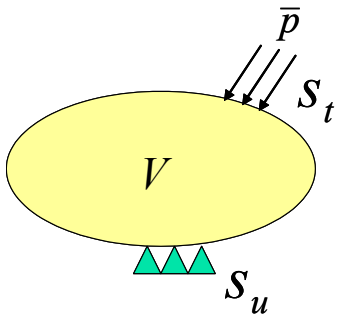


Figure 2.1.1: Boundary Value Problem in Solid Mechanics (Infinitesimal Deformation Problem)

The strain and displacement relational expression in the infinitesimal deformation problem is given by the following equation.

$$\boldsymbol{\varepsilon} = \nabla_s \mathbf{u} \tag{2.1.4}$$

The stress and strain relational expression (constitutive equation) in the linear elastic body is given by the following equation.

$$\boldsymbol{\sigma} = \mathbf{C} : \boldsymbol{\varepsilon} \tag{2.1.5}$$

Herein, C is the fourth order elastic tensor.

2.1.2 Principle of Virtual Work

The principle of the virtual work regarding the infinitesimal deformation linear elasticity problem equivalent to the basic equations (2.1), (2.1.2) and (2.1.3), is expressed as in the following equation.

$$\int_V \boldsymbol{\sigma} : \delta\boldsymbol{\varepsilon}\, dV = \int_{St} \bar{\mathbf{t}} \bullet \delta\mathbf{u} dS + \int_V \bar{\mathbf{b}} \bullet \delta\mathbf{u} dV \tag{2.1.6}$$

$$\delta\mathbf{u} = 0 \quad on \quad S_u \tag{2.1.7}$$

Furthermore, in consideration of the constitutive equation (2.1.5), equation (2.1.6) is expressed as in the following equation.

$$\int_V \left(\mathbf{C} : \boldsymbol{\varepsilon}\right) : \delta\boldsymbol{\varepsilon}\, dV = \int_{St} \bar{\mathbf{t}} \bullet \delta\mathbf{u} dS + \int_V \bar{\mathbf{b}} \bullet \delta\mathbf{u} dV \tag{2.1.8}$$

In equation (2.1.8), $\boldsymbol{\varepsilon}$ is the strain tensor and C is the forth order elastic tensor. In this case, when stress tensor $\boldsymbol{\sigma}$ and strain tensor $\boldsymbol{\varepsilon}$ are expressed by each vector forms $\hat{\boldsymbol{\sigma}}$ and $\hat{\boldsymbol{\varepsilon}}$, the constitutive equation (2.1.5) is expressed as in the following equation.

$$\hat{\boldsymbol{\sigma}} = \mathbf{D}\hat{\boldsymbol{\varepsilon}} \tag{2.1.9}$$

Herein, D is the elastic matrix.

In consideration of stress $\hat{\boldsymbol{\sigma}}$ and $\hat{\boldsymbol{\varepsilon}}$ expressed by the vector forms and equation (2.1.9), equation (2.1.8) is expressed as in the following equation.

$$\int_V \hat{\boldsymbol{\varepsilon}}^T \mathbf{D}\delta\hat{\boldsymbol{\varepsilon}} dV = \int_{St} \delta\mathbf{u}^T \bar{\mathbf{t}} dS + \int_V \delta\mathbf{u}^T \bar{\mathbf{b}} dV \tag{2.1.10}$$

Equation (2.1.10) and equation (2.1.7) are the principles of the virtual work discretized in this development code.

2.1.3 Formulation

The principle equation (2.1.10) of the virtual work is discretized for each finite element to acquire the following equation.

$$\sum_e \int_{V^e} \hat{\boldsymbol{\varepsilon}}^T \mathbf{D}\delta\hat{\boldsymbol{\varepsilon}}\, dV = \sum_e \int_{S_t^e} \delta\mathbf{u}^T \bar{\mathbf{t}} dS + \sum_e \int_{V^e} \delta\mathbf{u}^T \bar{\mathbf{b}} dV \tag{2.1.11}$$

Using the displacement of the nodes which consist of elements, the displacement field is interpolated for each element as in the following equation.

$$\mathbf{u} = \sum_{i=1}^{m} N_i \mathbf{u}_i = \mathbf{NU} \tag{2.1.12}$$

The strain in this case, is given as in the following equation using equation (2.1.4).

$$\hat{\boldsymbol{\varepsilon}} = \mathbf{BU} \tag{2.1.13}$$

Equations (2.1.12) and (2.1.13) are substituted with equation (2.1.11) to acquire the following equation.

$$\sum_e \delta\mathbf{U}^{\mathbf{T}} \left(\int_{V^e} \mathbf{B}^{\mathbf{T}} \mathbf{D}\mathbf{B} dV \right) \mathbf{U} = \sum_e \delta\mathbf{U}^T \bullet \int_{S_t^e} \mathbf{N}^T \bar{\mathbf{t}} dS + \sum_e \delta\mathbf{U}^T \int_{V^e} \mathbf{N}^T \bar{\mathbf{b}} dV \tag{2.1.14}$$

Equation (2.1.14) can be summarized as in the following equation.

$$\delta\mathbf{U}^{\mathbf{T}}\mathbf{K}\mathbf{U} = \delta\mathbf{U}^{\mathbf{T}}\mathbf{F} \tag{2.1.15}$$

Where,

$$\mathbf{K} = \sum_e \int_{V^e} \mathbf{B}^{\mathbf{T}} \mathbf{D}\mathbf{B} dV \tag{2.1.16}$$

$$\mathbf{F} = \sum_e \int_{S_t^e} \mathbf{N}^T \bar{\mathbf{t}} dS + \int_{V^e} \mathbf{N}^T \bar{\mathbf{b}} dV \tag{2.1.17}$$

The components of the matrix and vectors defined by the following equations (2.1.16) and (2.1.17) can be calculated and overlapped for each finite element.

The following equation can be acquired by forming equation (2.1.15) for the arbitrary virtual displacement $\delta\mathbf{U}$.

$$\mathbf{KU} = \mathbf{F} \tag{2.1.18}$$

On the other hand, the displacement boundary condition equation (2.1.3) is expressed as in the following equation.

$$\mathbf{U} = \overline{\mathbf{U}} \tag{2.1.19}$$

Node displacement $\mathbf{U}$ can be determined by solving equation (2.1.18) by the restriction condition equation (2.1.19).

2.2 Nonlinear Static Analysis Method

As mentioned above, in the analysis of the infinitesimal deformation problem, the finite element analysis can be performed by discretizing this equation with the finite element, using the principle of virtual work equivalent to a basic equation, such as the equilibrium equation. Even in the analysis of a finite deformation problem which handles finite deformation of structures, the point of using the principle of virtual work is basically the same. However, in a finite deformation problem, even though linearity of the material is assumed, the principle equation of virtual work will become a nonlinear equation regarding the displacement. In order to solve the nonlinear equation, repeated calculations by an iterative method is generally used. In the iterative calculation, an incremental analysis method is used, where a calculation is sectionally performed for certain small load increments, and repeating this calculation results in a final deformed state. When an infinitesimal deformation problem is assumed, the layout before and after deformation to define the strain and stress has not been distinguished in particular. Thus, when an infinitesimal deformation is assumed, the layout to describe the basic equation has not been a problem, even though it was before or after the deformation. However, when implementing an incremental analysis in a finite deformation problem, whether to refer to the initial status as a reference layout, or refer to the starting point of the increments can be selected. The former is called the total Lagrange method, and the latter is called the updated Lagrange method. For details, refer to the references and etc. at the end of this Chapter.

Both the total Lagrange method and updated Lagrange method have been adopted for this development code.

2.2.1 Geometric Nonlinear Analysis Method

2.2.1.1 Decomposition of Increments of Virtual Work Equation

The status to time t is already known. The incremental analysis assumed here is where the status of t' = t + Δ t is unknown. (Refer to Figure 2.2.1) The equilibrium equation, dynamic boundary conditions and geometric boundary conditions (basic boundary conditions) of the static boundary value problem are as follows.

$$\nabla_{{}^{t'}\mathbf{x}} \bullet {}^{t'}\boldsymbol{\sigma} + {}^{t'}\bar{\mathbf{b}} = 0 \quad in\ V \tag{2.2.1}$$

$${}^{t'}\boldsymbol{\sigma} \bullet {}^{t'}\mathbf{n} = {}^{t'}\bar{\mathbf{t}} \quad on\ {}^{t'}S \tag{2.2.2}$$

$${}^{t'}\mathbf{u} = {}^{t'}\bar{\mathbf{u}} \quad on\ {}^{t'}S_u \tag{2.2.3}$$

However, ${}^{t'}\boldsymbol{\sigma}, {}^{t'}\bar{\mathbf{b}}, {}^{t'}\mathbf{n}, {}^{t'}\bar{\mathbf{t}}, {}^{t'}\bar{\mathbf{u}}$ are the Cauchy stress (true stress), body force, outward normal vector of the object's surface, fixed surface force and fixed displacement in each time t'. These equations are described for the layout of ${}^{t'}V, {}^{t'}S_t, {}^{t'}S_u$ in time t'.

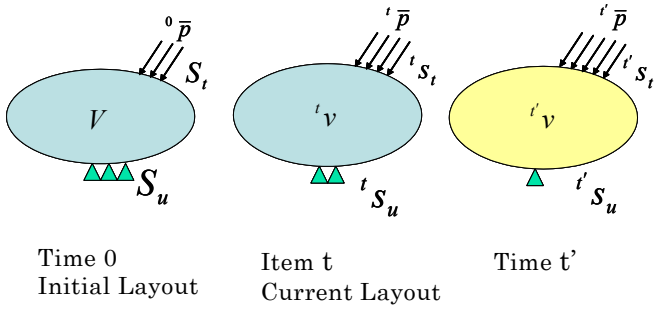


Figure 2.2.1: Concept of Incremental Analysis

2.2.1.2 Principle of Virtual Work

The principle of virtual work equivalent to the equilibrium equation of equation (2.2.1) and the dynamic boundary conditions of equation (2.2.2) is given by the following equation.

$$\int_{{}^{t'}V} {}^{t'}\boldsymbol{\sigma} : \delta\, {}^{t'}\mathbf{A}_{(L)}\, d\,{}^{t'}V = \int_{{}^{t'}S_t} {}^{t'}\bar{\mathbf{t}} \bullet \delta\mathbf{u}\, d\,{}^{t'}S + \int_{V} {}^{t'}\bar{\mathbf{b}} \bullet \delta\mathbf{u}\, d\,{}^{t'}V \tag{2.2.4}$$

Herein, ${}^{t'}\mathbf{A}_{(L)}$ is the linear portion of the Almansi strain tensor, and is specifically expressed by the following equation.

$${}^{t'}\mathbf{A}_{(L)} = \frac{1}{2}\left\{ \frac{\partial\, {}^{t'}\mathbf{u}}{\partial\, {}^{t'}\mathbf{x}} + \left(\frac{\partial\, {}^{t'}\mathbf{u}}{\partial\, {}^{t'}\mathbf{x}} \right)^{T} \right\} \tag{2.2.5}$$

Equation (2.2.4) 2.4) should be solved with the geometric boundary conditions, strain

displacement relational expression and the stress strain relational expression; however, equation (2.2.4) 2.4) is described in the layout of time t', and the layout of time t' is unknown at the present stage. Therefore, the formulation is performed referring to layout V of time 0, or layout ${}^{t}v$ at time t.

2.2.1.3 Formulation of Total Lagrange Method

The formulation based on the total Lagrange method used in the development code is described in this section.

The principle equation of the virtual work at time t' assuming the initial layout of time 0 is the reference, is given by the following equation.

$$\int_V {}_0^{t'}\mathbf{S} : \delta\, {}_0^{t'}\mathbf{E}\, dV = {}^{t'}\delta\mathbf{R} \tag{2.2.6}$$

$${}^{t'}\delta\mathbf{R} = \int_{S_t} {}_0^{t'}\bar{\mathbf{t}} \bullet \delta\mathbf{u}\, dS + \int_V {}_0^{t'}\bar{\mathbf{b}} \bullet \delta\mathbf{u}\, dV \tag{2.2.7}$$

However, ${}_0^{t'}\mathbf{S}$, ${}_0^{t'}\mathbf{E}$ respectively express the 2nd Piola-Kirchhoff stress tensor and the Green-Lagrange strain tensor at time t', assuming the initial layout of time 0 is the reference. Moreover, ${}_0^{t'}\bar{\mathbf{t}}$, ${}_0^{t'}\bar{\mathbf{b}}$ is the body force converted per unit volume of the nominal surface force vector and the initial layout, and is given by the following equation in connection with equations (2.2.1), (2.2.2) and (2.2.3).

$${}_0^{t'}\bar{\mathbf{t}} = \frac{d^{t'}s}{dS}\, {}^{t'}\bar{\mathbf{t}} \tag{2.2.8}$$

$${}_0^{t'}\bar{\mathbf{b}} = \frac{d^{t'}v}{dV}\, {}^{t'}\bar{\mathbf{b}} \tag{2.2.9}$$

The Green-Lagrange strain tensor at time t is defined by the following equation.

$${}_0^{t}\mathbf{E} = \frac{1}{2}\left\{ \frac{\partial {}^{t}\mathbf{u}}{\partial \mathbf{X}} + \left(\frac{\partial {}^{t}\mathbf{u}}{\partial \mathbf{X}} \right)^{T} + \left(\frac{\partial {}^{t}\mathbf{u}}{\partial \mathbf{X}} \right)^{T} \bullet \frac{\partial {}^{t}\mathbf{u}}{\partial \mathbf{X}} \right\} \tag{2.2.10}$$

Thus, the displacement and the 2nd Piola-Kirchhoff stress ${}^{t'}\mathbf{u}, {}_0^{t'}\mathbf{S}$ at time t' are expressed by the decomposed increments as in the following equation.

$${}^{t'}\mathbf{u} = {}^{t}\mathbf{u} + \Delta\mathbf{u} \tag{2.2.11}$$

$${}_0^{t'}\mathbf{S} = {}_0^{t}\mathbf{S} + \Delta\mathbf{S} \tag{2.2.12}$$

In this case, in relation to the displacement increment, the increment of the Green-Lagrange strain is defined by the following equation.

$$ {}_0^{t'}\mathbf{E} = {}_0^{t}\mathbf{E} + \Delta\mathbf{E} \tag{2.2.13} $$

$$ \Delta\mathbf{E} = \Delta\mathbf{E}_L + \Delta\mathbf{E}_{NL} \tag{2.2.14} $$

$$ \Delta\mathbf{E}_L = \frac{1}{2}\left\{\frac{\partial\Delta\mathbf{u}}{\partial\mathbf{X}} + \left(\frac{\partial\Delta\mathbf{u}}{\partial\mathbf{X}}\right)^T + \left(\frac{\partial\Delta\mathbf{u}}{\partial\mathbf{X}}\right)^T \bullet \frac{\partial {}^t\mathbf{u}}{\partial\mathbf{X}} + \left(\frac{\partial {}^t\mathbf{u}}{\partial\mathbf{X}}\right)^T \bullet \frac{\partial\Delta\mathbf{u}}{\partial\mathbf{X}}\right\} \tag{2.2.15} $$

$$ \Delta\mathbf{E}_{NL} = \frac{1}{2}\left(\frac{\partial\Delta\mathbf{u}}{\partial\mathbf{X}}\right)^T \bullet \frac{\partial\Delta\mathbf{u}}{\partial\mathbf{X}} \tag{2.2.16} $$

Equations (2.2.11)(2.2.12)(2.2.13)(2.2.14)(2.2.15) and (2.2.16) を、 are substituted with equations (2.2.6) and (2.2.7) to acquire the following equation.

$$ \int_V \Delta\mathbf{S} : \left(\delta\Delta\mathbf{E}_L + \delta\Delta\mathbf{E}_{NL}\right)dV + \int_V {}_o^t\mathbf{S} : \delta\Delta\mathbf{E}_{NL}\,dV = {}^{t'}\delta\mathbf{R} - \int_V {}_o^t\mathbf{S} : \delta\Delta\mathbf{E}_L\,dV \tag{2.2.17} $$

Herein, $\Delta\mathbf{S}$ is assumed to be expressed as in the following equation in connection with $\Delta\mathbf{E}_L$ and the forth order tensor ${}_0^t\mathbf{C}$.

$$ \Delta\mathbf{S} = {}_0^t\mathbf{C} : \Delta\mathbf{E}_L \tag{2.2.18} $$

Equation (2.2.17) is substituted with equation (2.2.18) and $\Delta\mathbf{S} : \delta\Delta\mathbf{E}_{NL}$ having two or more polynomials of Δu are omitted to acquire the following equation.

$$ \int_V ({}_0^t\mathbf{C} : \Delta\mathbf{E}_L) : \delta\Delta\mathbf{E}_L\,dV + \int_V {}_o^t\mathbf{S} : \delta\Delta\mathbf{E}_{NL}\,dV = {}^{t'}\delta\mathbf{R} - \int_V {}_o^t\mathbf{S} : \delta\Delta\mathbf{E}_L\,dV \tag{2.2.19} $$

Equation (2.2.19) is discretized by the finite element to acquire the following equation.

$$ \delta\mathbf{U}^T\left({}_0^t\mathbf{K}_L + {}_0^t\mathbf{K}_{NL}\right)\Delta\mathbf{U} = \delta\mathbf{U}^T\,{}_0^{t'}\mathbf{F} - \delta\mathbf{U}^T\,{}_0^t\mathbf{Q} \tag{2.2.20} $$

Herein, ${}_0^t\mathbf{K}_L, {}_0^t\mathbf{K}_{NL}, {}_0^{t'}\mathbf{F}, {}_0^t\mathbf{Q}$ are the initial displacement matrix, initial stress matrix, external force vector and internal force vector respectively.

Therefore, the recurrence equation to acquire the time t' status from the time t status is given by the following equation.

Step1： $i = 0$

$$ {}_0^{t'}\mathbf{K}^{(0)} = {}_0^t\mathbf{K}_L + {}_0^t\mathbf{K}_{NL};\ {}_0^{t'}\mathbf{Q}^{(0)} = {}_0^t\mathbf{Q};\ {}^{t'}\mathbf{U}^{(0)} = {}^t\mathbf{U} $$

Step2： $ {}_0^{t'}\mathbf{K}^{(i)}\Delta\mathbf{U}^{(i)} = {}_0^{t'}\mathbf{F} - {}_0^{t'}\mathbf{Q}^{(i-1)} $

Step3： $ {}^{t'}\mathbf{U}^{(i)} = {}^{t'}\mathbf{U}^{(i-1)} + \Delta\mathbf{U}^{(i)} $

$i = 0$

2.2.1.4 Formulation of Updated Lagrange Method

The principle equation of the virtual work at time t' assuming the current layout of time t is the reference, is given by the following equation.

$$\int_V {}^{t'}_{t}\mathbf{S} : \delta\, {}^{t'}_{t}\mathbf{E} dV = {}^{t'}\delta\mathbf{R} \tag{2.2.21}$$

$${}^{t'}\delta\mathbf{R} = \int_{S_t} {}^{t'}_{t}\bar{\mathbf{t}} \bullet \delta\mathbf{u} dS + \int_V {}^{t'}_{t}\bar{\mathbf{b}} \bullet \delta\mathbf{u} dV \tag{2.2.22}$$

However,

$${}^{t'}_{t}\bar{\mathbf{t}} = \frac{d^{t'}s}{d^{t}s} {}^{t'}\bar{\mathbf{t}} \tag{2.2.23}$$

$${}^{t'}_{t}\bar{\mathbf{b}} = \frac{d^{t'}v}{d^{t}v} {}^{t'}\bar{\mathbf{b}} \tag{2.2.24}$$

although tensor ${}^{t'}_{t}\mathbf{S}$, ${}^{t'}_{t}\mathbf{E}$ and vector ${}^{t'}_{t}\bar{\mathbf{t}}$, ${}^{t'}_{t}\bar{\mathbf{b}}$ are using the current layout of time t as the reference, the Green-Lagrange strain does not include the initial displacement (displacement to time t) ${}^{t}\mathbf{u}$;

$${}^{t'}_{t}\mathbf{E} = \Delta_t \mathbf{E}_L + \Delta_t \mathbf{E}_{NL} \tag{2.2.25}$$

however, the equation becomes as follows.

$$\Delta_t \mathbf{E}_L = \frac{1}{2}\left\{ \frac{\partial \Delta\mathbf{u}}{\partial^t x} + \left(\frac{\partial \Delta\mathbf{u}}{\partial^t x} \right)^T \right\} \tag{2.2.26}$$

$$\Delta_t \mathbf{E}_{NL} = \frac{1}{2}\left(\frac{\partial \Delta\mathbf{u}}{\partial^t x} \right)^T \bullet \frac{\partial \Delta\mathbf{u}}{\partial^t x} \tag{2.2.27}$$

On the other hand,

$${}^{t'}_{t}\mathbf{S} = {}^{t}_{t}\mathbf{S} + \Delta_t \mathbf{S} \tag{2.2.28}$$

since the equation becomes as above, when this is arranged by substituting with equations (2.2.21) and (2.2.22), and equation (2.2.25), the equation which must be solved is given as follows.

$$\int_{{}^{t}v} \Delta_t \mathbf{S} : \left(\delta\Delta_t \mathbf{E}_L + \delta\Delta_t \mathbf{E}_{NL} \right) d^t v + \int_V {}^{t}_{t}\mathbf{S} : \delta\Delta_t \mathbf{E}_{NL} d^t v = {}^{t'}\delta\mathbf{R} - \int_{{}^{t}v} {}^{t}_{t}\mathbf{S} : \delta\Delta_t \mathbf{E}_L d^t v \tag{2.2.29}$$

In this case, $\Delta_t \mathbf{S}$ is assumed to be expressed as in the following equation in connection with $\Delta_t \mathbf{E}_L$ and the forth order tensor ${}^{t}_{t}\mathbf{C}$.

$$\Delta_t \mathbf{S} = {}^{t}_{t}\mathbf{C} : \Delta_t \mathbf{E}_L \tag{2.2.30}$$

This is substituted with equation (2.2.29) to acquire the following equation.

$$\int_V ({}_t^t\mathbf{C} : \Delta_t\mathbf{E}_L) : \delta\Delta_t\mathbf{E}_L dV + \int_V {}_t^t\mathbf{S} : \delta\Delta_t\mathbf{E}_{NL} dV = {}^{t'}\delta\mathbf{R} - \int_V {}_t^t\mathbf{S} : \delta\Delta_t\mathbf{E}_L dV \tag{2.2.31}$$

Equation (2.2.31) is discretized by the finite element to acquire the following equation.

$$\delta\mathbf{U}^T \left({}_t^t\mathbf{K}_L + {}_t^t\mathbf{K}_{NL} \right) \Delta\mathbf{U} = \delta\mathbf{U}^T {}_t^{t'}\mathbf{F} - \delta\mathbf{U}^T {}_t^t\mathbf{Q} \tag{2.2.32}$$

Herein, ${}_t^t\mathbf{K}_L, {}_t^t\mathbf{K}_{NL}, {}_t^{t'}\mathbf{F}, {}_t^t\mathbf{Q}$ are the initial displacement matrix, initial stress matrix, external force vector and internal force vector respectively.

Therefore, the recurrence equation to acquire the time t' status from the time t status is given by the following equation.

Step1 : $i = 0$

$${}_t^{t'}\mathbf{K}^{(i)} = {}_t^t\mathbf{K}_L + {}_t^t\mathbf{K}_{NL}; {}_t^{t'}\mathbf{Q}^{(i)} = {}_t^t\mathbf{Q}; {}^{t'}\mathbf{U}^{(i)} = {}^t\mathbf{U}$$

Step2 : ${}_t^{t'}\mathbf{K}^{(i)}\Delta\mathbf{U}^{(i)} = {}_t^{t'}\mathbf{F} - {}_t^{t'}\mathbf{Q}^{(i-1)}$

Step3 : ${}^{t'}\mathbf{U}^{(i)} = {}^{t'}\mathbf{U}^{(i-1)} + \Delta\mathbf{U}^{(i)}$

$i = i + 1$

2.2.2 Material Nonlinear Analysis Method

In this development code, 2 types of analysis, such as isotropic hyperelasticity and the elastoplasticity can be performed for nonlinear materials. When the material applicable for analysis is an elastoplastic material, the updated Lagrange method is applied, and the total Lagrange method is applied for hyperelastic material. Moreover, the Newton-Raphson method is applied to the repetitive analysis method.

The outline of these constitutive equations of materials is shown in the following.

2.2.2.1 Hyperelastic Material

The elastic potential energy in isotropic hyperelastic material can acquire the isotropic response from the initial state without the activation of stress. Therefore, the function of the main invariable of the right Cauchy-Green deformation tensor C ($\mathbf{I_1}, \mathbf{I_2}, \mathbf{I_3}$), or the main invariable of the deformation tensor excluding the change in volume ($\bar{\mathbf{I}}_\mathbf{1}, \bar{\mathbf{I}}_\mathbf{2}, \bar{\mathbf{I}}_\mathbf{3}$), can be expressed as $\mathbf{W} = \mathbf{W}(\mathbf{I_1}, \mathbf{I_2}, \mathbf{I_3})$, or $\mathbf{W} = \mathbf{W}(\bar{\mathbf{I}}_\mathbf{1}, \bar{\mathbf{I}}_\mathbf{2}, \bar{\mathbf{I}}_\mathbf{3})$.

The constitutive equation of hyperelastic material is defined by the relationship between the 2nd Piola-Kirchhoff stress and the Green-Lagrange strain, and the total Lagrange method is applied for the deformation analysis.

The elastic potential energy W of the hyperelasticity model included in this development code is listed in the following. If the elastic potential energy W is known, the 2nd Piola-Kirchhoff stress and the stress-strain relationship can be calculated as follows.

$$S = 2\frac{\partial W}{\partial C} \tag{2.2.33}$$

$$C = 4\frac{\partial^2 W}{\partial C\,\partial C} \tag{2.2.34}$$

(1) Neo Hookean Hyperelasticity Model

The Neo-Hookean hyperelasticity model is a material model with an expanded linear rule (Hooke rule) having isotropy so that it can respond to finite deformation problems. The elastic potential is as follows.

$$W = C_{10}(\bar{I}_1 - 3) + \frac{1}{D_1}(J-1)^2 \tag{2.2.35}$$

Herein, C_{10} and D_1 are the material constants.

(2) Mooney Rivlin Hyperelasticity Model

$$W = C_{10}(\bar{I}_1 - 3) + C_{01}(\bar{I}_2 - 3) + \frac{1}{D_1}(J-1)^2 \tag{2.2.36}$$

Herein, $C_{10,}\, C_{01}$ and D_1 are the material constants.

(3) Arruda Boyce Hyperelasticity Model

$$W = \mu\left[\frac{1}{2}(\bar{I}_1 - 3) + \frac{1}{20\lambda_m^2}(\bar{I}_1^2 - 9) + \frac{11}{1050\lambda_m^2}(\bar{I}_1^3 - 27) + \frac{19}{7000\lambda_m^2}(\bar{I}_1^4 - 81) + \frac{519}{673750\lambda_m^2}(\bar{I}_1^5 - 243)\right] + \frac{1}{D}\left(\frac{J^2-1}{2} - \ln J\right) \tag{2.2.37}$$

$$\mu = \frac{\mu_0}{1 + \frac{3}{5\lambda_m{}^2} + \frac{99}{175\lambda_m{}^4} + \frac{513}{875\lambda_m{}^6} + \frac{42039}{67375\lambda_m{}^8}} \tag{2.2.38}$$

Herein, μ, λ_m and D are the material constants.

2.2.2.2 Elastoplastic Material

In this development code, the elastoplasticity constitutive equation according to the associated flow rule is applied. Moreover, the constitutive equation expresses the relationship between the Jaumman rate and the deformation rate tensor of the Kirchhoff stress, and the updated Lagrange method is applied in the deformation analysis.

(1) Elastoplastic Constitutive Equation

The yield criteria of an elasto-plastic solid is assumed to be given as follows.

Initial Yield Criteria

$$F(\boldsymbol{\sigma}, \sigma_{y0}) = 0 \tag{2.2.39}$$

Consecutive Yield Criteria

$$F(\boldsymbol{\sigma}, \sigma_y(\bar{e}^p)) = 0 \tag{2.2.40}$$

Where,

F: Yield function

σ_{y0}: Initial yield stress, σ_y: Consecutive yield stress

$\boldsymbol{\sigma}$: Stress tensor, $\mathbf{e}$: Infinitesimal strain tensor

$\mathbf{e}^p$: Plastic strain tensor, $\bar{e}^p$: Equivalent plastic strain

The yield stress-equivalent plastic strain relationship is assumed to conform to the stress-plastic strain relationship in a single axis state.

Stress-plastic strain relationship in a single axis state:

$$\sigma = H(e^p) \tag{2.2.41}$$

$$\frac{d\sigma}{de^p} = H' \tag{2.2.42}$$

Where,

H' : Strain hardening factor

Equivalent stress-equivalent plastic strain relationship:

$$\bar{\sigma} = H(\bar{e}^p) \tag{2.2.43}$$

$$\dot{\bar{\sigma}} = H' \dot{\bar{e}}^p \tag{2.2.44}$$

The consecutive yield function is generally a function of temperature and plastic strain work. However, for simplification, the function is only assumed to be the equivalent plastic strain $\bar{e}^p$ in this section. Since F=0 continues to be satisfied during the progression of the plastic deformation, the following equation must be established.

$$\dot{F} = \frac{\partial F}{\partial \boldsymbol{\sigma}} : \dot{\boldsymbol{\sigma}} + \frac{\partial F}{\partial \boldsymbol{e}^p} : \dot{\boldsymbol{e}}^p = 0 \tag{2.2.44}$$

$\dot{F}$ in equation (2.2.44) expresses the time derivative function of F, and the time derivative function of a certain amount of A is expressed as $\dot{A}$ hereafter.

In this case, assuming the existence of plastic potentialΘ, the plastic strain rate is expressed by the following equation.

$$\dot{\boldsymbol{e}}^{p} = \dot{\lambda}\frac{\partial \Theta}{\partial \boldsymbol{\sigma}} \tag{2.2.45}$$

Herein, $\dot{\lambda}$ is the factor.

Furthermore, assuming that plastic potential$\Theta$is equivalent to yield function F, the associated flow rule is assumed as in the following equation.

$$\dot{\boldsymbol{e}}^{p} = \dot{\lambda}\frac{\partial F}{\partial \boldsymbol{\sigma}} \tag{2.2.46}$$

When this equation is substituted with equation (2.4.44), the following equation can be acquired.

$$\dot{\lambda} = \frac{\mathbf{a}^{T} : \mathbf{d}_{D}}{A + \mathbf{a}^{T} : \mathbf{D} : \mathbf{a}}\dot{\mathbf{e}} \tag{2.2.47}$$

Where, $\mathbf{D}$ is the elastic matrix,

$$\mathbf{a}^{T} = \frac{\partial F}{\partial \boldsymbol{\sigma}} \qquad \mathbf{d}_{D} = \mathbf{D}\mathbf{a}^{T} \qquad A = -\frac{1}{\dot{\lambda}}\frac{\partial F}{\partial \boldsymbol{e}^{p}} : \dot{\boldsymbol{e}}^{p} \tag{2.2.48}$$

the stress-strain relational expression of the elastoplasticity can be written as follows.

$$\dot{\boldsymbol{\sigma}} = \left\{ \boldsymbol{D} - \frac{\mathbf{d}_{D} \otimes \mathbf{d}_{D}^{T}}{A + \mathbf{d}_{D}^{T}\mathbf{a}} \right\} : \dot{\boldsymbol{e}} \tag{2.2.49}$$

When the yield function (2.2.49) of an elastoplastic material is known, the constitutive equation can be acquired from this equation.

(1) Yield Function

The elastoplastic yield functions included in this development code are listed in the following.

・Von Mises Yield Function

$$\mathrm{F} = \sqrt{3\mathbf{J}_{2}} - \sigma_{\mathrm{y}} = 0 \tag{2.2.50}$$

・Mohr-Coulomb Yield Function

$$\mathrm{F} = \sigma_{1} - \sigma_{3} + (\sigma_{1} + \sigma_{3})\sin\phi - 2\,\mathrm{c}\cos\phi = 0 \tag{2.2.51}$$

・Drucker-Prager Yield Function

$$\mathrm{F} = \sqrt{\mathbf{J}_{2}} - \alpha\,\boldsymbol{\sigma} : \mathbf{I} - \sigma_{\mathrm{y}} = 0 \tag{2.2.52}$$

In this case, material constant α and σ_y are calculated as follows from the viscosity and friction angle of the material.

$$\alpha = \frac{2\sin\phi}{3+\sin\phi}\ , \qquad \sigma_y = \frac{6\,c\cos\phi}{3+\sin\phi} \tag{2.2.53}$$

2.2.2.3 Viscoelastic Material

A generalized Maxwell model is applied in this development code. As shown in the following, the constitutive equation becomes a function of deviatoric strain $\mathbf{e}$ and deviatoric viscosity strain $\mathbf{q}$.

$$\boldsymbol{\sigma}(t) = K\mathrm{tr}\boldsymbol{\varepsilon}\mathbf{I} + 2G\left(\mu_0\mathbf{e} + \mu\mathbf{q}\right) \tag{2.2.54}$$

Which becomes,

$$\mu\mathbf{q} = \sum_{m=1}^{M}\mu_m\,\mathbf{q}^{(m)}; \qquad \sum_{m=0}^{M}\mu_m = 1 \tag{2.2.55}$$

Moreover, q can be calculated from the following equation.

$$\dot{\mathbf{q}}^{(m)} + \frac{1}{\lambda_m}\mathbf{q}^{(m)} = \dot{\mathbf{e}} \tag{2.2.56}$$

Herein, λ_m is the relaxation. Relaxation factor G is expressed by the following Prony series.

$$G(t) = G\left[\mu_0 + \sum_{i=1}^{M}\mu_m\exp(-t/\lambda_m)\right] \tag{2.2.57}$$

2.2.2.4 Creep Material

Time dependent displacement under constant stress conditions is a phenomenon called "creep". The viscoelasticity behavior mentioned above can also be considered as a type of linear creep phenomenon. Several types of nonlinear creeping are described in this section. The method to form the constitutive equation by adding to the strain generated momentarily is generally used for this phenomenon, and the strain while a certain constant load is continued, is assumed to be creep strain $\boldsymbol{\varepsilon}^c$. The creep strain rate $\dot{\boldsymbol{\varepsilon}}^c$ which is defined as a function of the stress and overall creep strain, is generally used in the constitutive equation in consideration of the creep.

$$\dot{\boldsymbol{\varepsilon}}^c \equiv \frac{\partial\boldsymbol{\varepsilon}^c}{\partial t} = \boldsymbol{\beta}\left(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^c\right) \tag{2.2.58}$$

In this case, assuming the strain generated momentarily is the elastic strain $\boldsymbol{\varepsilon}^e$, the overall strain can be expressed as in the following equation where the creep strain is added.

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}^{e} + \boldsymbol{\varepsilon}^{c} \tag{2.2.59}$$

Which becomes,

$$\boldsymbol{\varepsilon}^{e} = \boldsymbol{c}^{e^{-1}} : \boldsymbol{\sigma} \tag{2.2.60}$$

As mentioned in the above plastic material, the time integration method for the numerical analysis must be indicated for the constitutive equation which indicates the creep. The constitutive equation when creep is taken into consideration is,

$$\boldsymbol{\sigma}_{n+1} = \boldsymbol{c} : (\boldsymbol{\varepsilon}_{n+1} - \boldsymbol{\varepsilon}^{c}_{n+1}) \tag{2.2.61}$$

$$\boldsymbol{\varepsilon}^{c}_{n+1} = \boldsymbol{\varepsilon}^{c}_{n} + \Delta t \, \boldsymbol{\beta}_{n+\theta} \tag{2.2.62}$$

where, $\boldsymbol{\beta}_{n+\theta}$ becomes as follows.

$$\boldsymbol{\beta}_{n+\theta} = (1 - \theta) \, \boldsymbol{\beta}_{n} + \theta \, \boldsymbol{\beta}_{n+1} \tag{2.2.63}$$

Moreover, the creep strain increment $\Delta\boldsymbol{\varepsilon}^{c}$ is assumed to be a simplified nonlinear equation.

$$\mathbf{R}_{n+1} = \boldsymbol{\varepsilon}_{n+1} - \boldsymbol{c}^{-1} : \boldsymbol{\sigma}_{n+1} - \boldsymbol{\varepsilon}^{c}_{n} - \Delta t \, \boldsymbol{\beta}_{n+\theta} = \mathbf{0} \tag{2.2.64}$$

In the iterative calculation of the Newton-Raphson method, the following equation is used for the iterative solution and the increment solution as an incremental strain where the initial value is calculated by $\boldsymbol{\sigma}_{n+1} = \boldsymbol{\sigma}_{n}$ and the finite element method.

$$\mathbf{R}^{(k+1)}_{n+1} = \mathbf{0} = \mathbf{R}^{(k)}_{n+1} - (\boldsymbol{c}^{-1} + \Delta t \, \boldsymbol{c}^{c}_{n+1}) \, d\boldsymbol{\sigma}^{(k)}_{n+1} \tag{2.2.65}$$

Which becomes,

$$\boldsymbol{c}^{c}_{n+1} = \frac{\partial \boldsymbol{\beta}}{\partial \boldsymbol{\sigma}} \Big|_{n+\theta} = \theta \frac{\partial \boldsymbol{\beta}}{\partial \boldsymbol{\sigma}} \Big|_{n+1} \tag{2.2.66}$$

When the solution of equation (2.2.66) and equation (2.2.67) are used to perform the iterative solution method until the residual $\mathbf{R}$ becomes $\mathbf{0}$, stress $\boldsymbol{\sigma}_{n+1}$ and the tangent tensile modulus are used.

$$\boldsymbol{c}^{*}_{n+1} = [\boldsymbol{c}^{-1} + \Delta t \, \boldsymbol{c}^{c}_{n+1}]^{-1} \tag{2.2.67}$$

As a detailed equation of equation (2.2.57), the following Norton model is applied in this development code. In the constitutive equation, the equivalent clip strain $\dot{\varepsilon}^{cr}$ as in the following equation expresses the function of the Mises stress q and time t.

$$\dot{\varepsilon}^{cr} = A q^{n} t^{m} \tag{2.2.68}$$

Herein, A, m and n are the material constants.

2.2.3 Contact Analysis Method

When two objects are in contact, the contact force $\mathbf{t_c}$ is transmitted via the contact surface. The principle equation (2.2.4) of the virtual work can be rewritten as follows.

$$\int_{V}^{t'} \boldsymbol{\sigma} : \delta^{t'}\mathbf{A}_{(L)} d^{t'}v = \int_{S_t}^{t'} \bar{\mathbf{t}} \bullet \delta\mathbf{u} d^{t'}s + \int_{V}^{t'} \bar{\mathbf{b}} \bullet \delta\mathbf{u} d^{t'}v + \int_{S_c}^{t'} \mathbf{t}_c \left[\delta\mathbf{u}^{(1)} - \delta\mathbf{u}^{(2)}\right] \tag{2.2.69}$$

In this case, Sc expresses the contact area, and $\mathbf{u}^{(1)}$ and $\mathbf{u}^{(2)}$ express the displacement of contact object 1 and contact object 2 respectively.

In the contact analysis, the surfaces which may contact are specified in a pair. One of these surfaces is called the master surface, and the other surface is the slave surface. In this master slave analysis method, the contact restriction conditions are assumed as follows.

1) The slave node does not penetrate the master surface.

2) When there is contact, the slave node becomes the contact position, and the master surface and the slave surface mutually transmit the contact force and the frictional force through this point of contact.

The last term of equation (2.2.54) is discretized by the finite element to acquire the following equation.

$$\int_{S_c}^{t'} \mathbf{t}_c \left[\delta\mathbf{u}^{(1)} - \delta\mathbf{u}^{(2)}\right] \approx \delta\mathbf{U}\mathbf{K}_C\Delta\mathbf{U} + \delta\mathbf{U}\mathbf{F}_c \tag{2.2.70}$$

In this case, $\mathbf{K}_c$ and $\mathbf{F}_c$ express the contact rigid matrix and contact force respectively. When this equation is substituted with equation (2.2.20) or (2.2.32), the finite element method of the total Lagrange method and the updated Lagrange method in consideration of the contact restraint becomes as follows.

$$\delta\mathbf{U}^T \left({}_0^t\mathbf{K}_L + {}_0^t\mathbf{K}_{NL} + \mathbf{K}_c\right)\Delta\mathbf{U} = \delta\mathbf{U}^T {}_0^{t'}\mathbf{F} - \delta\mathbf{U}^T {}_0^t\mathbf{Q} + \delta\mathbf{U}^T\mathbf{F}_c \tag{2.2.71}$$

$$\delta\mathbf{U}^T \left({}_t^t\mathbf{K}_L + {}_t^t\mathbf{K}_{NL} + \mathbf{K}_c\right)\Delta\mathbf{U} = \delta\mathbf{U}^T {}_t^{t'}\mathbf{F} - \delta\mathbf{U}^T {}_t^t\mathbf{Q} + \delta\mathbf{U}^T\mathbf{F}_C \tag{2.2.72}$$

This development software allows for contact deformation analysis between deformable bodies, and the following analysis functions can be selected by the user.

- Infinitesimal sliding contact problem: This analysis assumes that there is no position change of the point of contact.
- Limited sliding contact problem: This analysis can be used when there is a change of the point of contact accompanying the deformation.
- Frictionless contact problem
- Friction contact problem: This analysis supports the Coulomb friction rule.

However, when the infinitesimal deformation linear elastic analysis is selected, it becomes an infinitesimal sliding frictionless problem.

Moreover, it only corresponds to the contact analysis of a linear solid element (element numbers 341,351,361) at present.

2.3 Eigenvalue Analysis

2.3.1 Generalized Eigenvalue Problem

When conducting a free oscillation analysis of the continuum, spatial discretization is performed, and a model is created with the multi degrees of freedom system by a central mass point as shown in Figure 2.3.1. In the case of the free oscillation problem without attenuation, the governing equation (equation of motion) is as follows.

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{K}\mathbf{u} = 0 \tag{2.3.1}$$

However, $\mathbf{u}$ is the generalized displacement vector, $\mathbf{M}$ is the mass matrix and $\mathbf{K}$ is the stiffness matrix. Now, the function is defined assuming the natural angular frequency as ω, a and b as the arbitrary constants, and $\mathbf{x}$ as the vector as follows.

$$\mathbf{u}(t) = (a\sin\omega t + b\cos\omega t)\mathbf{x} \tag{2.3.2}$$

Herein, when this equation and this second order differential,

$$\ddot{\mathbf{u}}(t) = \omega(a\sin\omega t - b\sin\omega t)\mathbf{x} \tag{2.3.3}$$

is substituted with equation (2.3.1), it will become as follows.

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{K}\mathbf{u} = (a\sin\omega t + b\cos\omega t)(-\omega^2\mathbf{M} + \mathbf{K}\mathbf{x}) = (-\lambda\mathbf{M}\mathbf{x} + \mathbf{K}\mathbf{x}) = 0 \tag{2.3.4}$$

Which becomes,

$$\mathbf{K}\mathbf{x} = \lambda\mathbf{M}\mathbf{x} \tag{2.3.5}$$

That is to say, if factor λ (= ω 2) and vector $\mathbf{x}$ which satisfies equation (2.3.5) can be found, function $\mathbf{u}$(t) becomes the solution of equation (2.3.1). Factor λ is called the eigenvalue, vector $\mathbf{x}$ is called the eigenvector, and the problem in which these values are calculated by equation (2.3.1) is called the generalized eigenvalue problem.

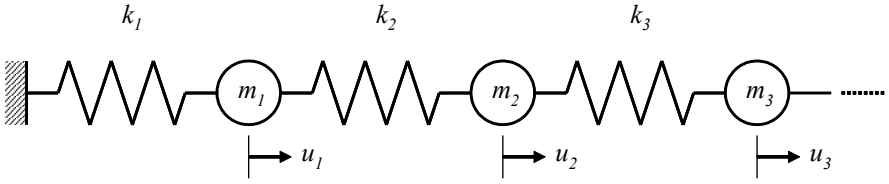


Figure 2.3.1: Example of Multi Degrees of Freedom System of Free Oscillation without Attenuation

2.3.2 Problem Settings

Equation (2.3.5) can be expanded to arbitrary order frequencies, and appears in many situations. In order to handle many physical problems, there are many cases where the matrix is Hermitian (symmetrical). Thus, in a complex matrix, the transposed matrix is transposed into a conjugate complex number, and a symmetric matrix in the real matrix. That is to say, when the ij component of matrix K is assumed to be k_{ij}, if the conjugate complex number of k is transposed to $\bar{k}$, the relationship becomes as in the following equation.

$$k_{ij} = \bar{k}_{ji} \tag{2.3.6}$$

In this manual, the matrix is symmetrical, and assumes a positive definite. Positive definite means that all the eigenvalues are positive, in other words, it refers to a matrix which always satisfies the following equation (2.3.7).

$$\mathbf{x}^{\mathrm{T}}\mathbf{A}\mathbf{x} > 0 \tag{2.3.7}$$

2.3.3 Shifted Inverse Iteration Method

In the structural analysis by the finite element method, all the eigenvalues are not required in practical use, and there are many cases where an extensive number of low order eigenvalues is sufficient. Now, the handling of large-scale problems is assumed in HEC-MW, and the size of the matrix is large and extremely sparse (many zero elements). Therefore, considering this fact, it is important to efficiently calculate the eigenvalues of the low order modes.

When the lower limit of the eigenvalue is set too, equation (2.3.5) is deformed as in the following equation (mathematically equivalent equation).

$$\left(\mathbf{K} - \sigma\mathbf{M}\right)^{-1}\mathbf{M}\mathbf{x} = \left[1/\left(\lambda - \sigma\right)\right]\mathbf{x} \tag{2.3.8}$$

In this case, there are the following convenient characteristics for calculation.

(1) The mode is reversed.
(2) The eigenvalue around ρ is maximized.

In actual calculations, there are many cases where the maximum eigenvalue is calculated first. For this reason, the main convergence calculation is applied to equation (2.3.8), rather than equation (2.3.5), aiming to calculate from the eigenvalues around ρ. This method is called the shifted inverse iteration.

2.3.4 Algorithm to Solve Eigenvalues

The Jacobi method is well known as a classic method. This method is effective when the size of the matrix is small and dense. However, since large-scale and sparse matrices are handled by HEC-MW, this method is not used, and the Lanczos iterative solution method is adopted.

2.3.5 Lanczos Method

This method was proposed by C. Lanczos in the 1950s, and is a calculation algorithm which performs tridiagonalization of matrices, and has the following features.

(1) This is an iterative convergence solution, where calculations can be advanced with sparse matrices.
(2) The algorithm is mainly structured with matrix and vector products, and is suitable for parallelization.
(3) This is suitable for the geometric domain decomposition method (DDM) accompanying the finite element mesh.
(4) The number of the eigenvalues to be calculated and the mode range can be restricted, and efficient calculations can be performed.

The Lanczos method is for performing calculations to acquire the base of partial spaces, by sequentially creating orthogonal vectors starting from the initial vector. This method is said to be faster than the sub space method which is another iterative solution method, and is extensively used in the finite element method program. However, this method is easily affected by errors of the calculator, which impairs the orthogonality of the vector, and the risks of failing in the middle can not be avoided. Therefore, it is essential to take measure against errors.

2.3.6 Geometric Meaning in the Lanczos Method

Transforming the variables of equation (2.3.8) as follows,

$$\mathbf{A} = \left(\mathbf{K} - \sigma\mathbf{M}\right)^{-1}\mathbf{M}, \quad \left[1/\left(\lambda - \sigma\right)\right] = \xi \tag{2.3.9}$$

to rewrite the problem, the following equation can be acquired.

$$\mathbf{Ax} = \xi\mathbf{x} \tag{2.3.10}$$

The linear transformation with matrix $\mathbf{A}$ is performed for proper vector **qo** (refer to Figure 2.3.2).

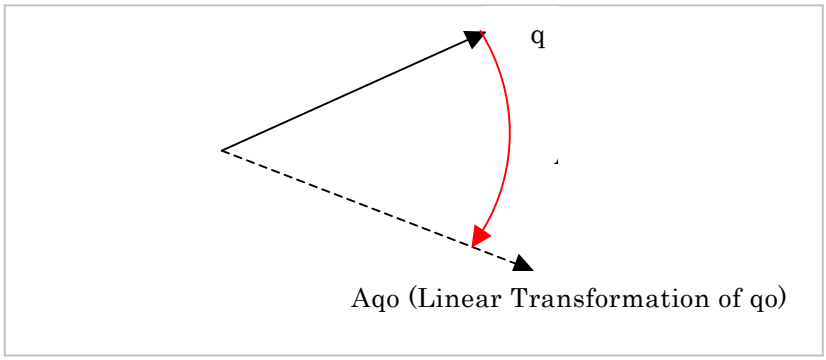


Figure 2.3.2: Linear Transformation of $\mathbf{q}_0$ with Matrix $\mathbf{A}$

The transformed vector is orthogonalized in the space created with the original vector. Thus, as in Figure 2.3.2 the so called Gram-Schmidt orthogonalization is performed. The vector acquired in such a way, is normalized as $\mathbf{r}_1$ (to length 1), to acquire $\mathbf{q}_1$ (Figure 2.3.3). $\mathbf{q}_2$ is acquired from $\mathbf{q}_1$ with the same algorithm. At this time, $\mathbf{q}_2$ is orthogonal to both $\mathbf{q}_1$ and $\mathbf{q}_0$ (Figure 2.3.4). When the same calculation is continued, the mutually orthogonal vectors can be acquired up to the order frequency of the maximum matrix.

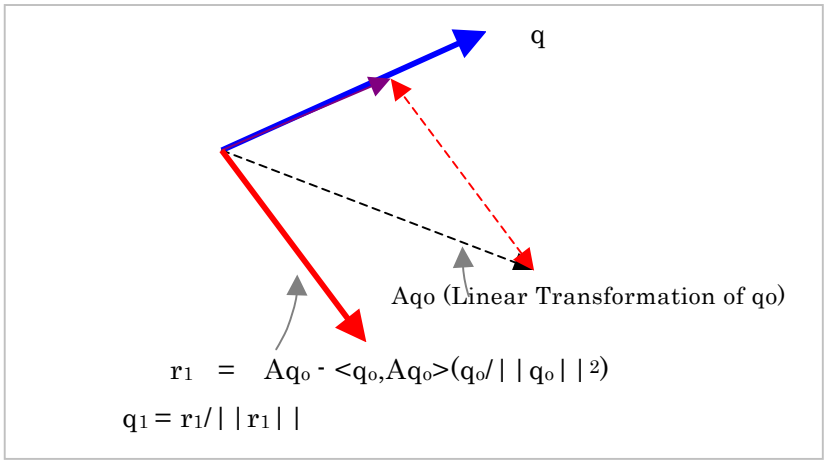


Figure 2.3.3: Vector $\mathbf{q}_1$ Orthogonal to $\mathbf{q}_o$

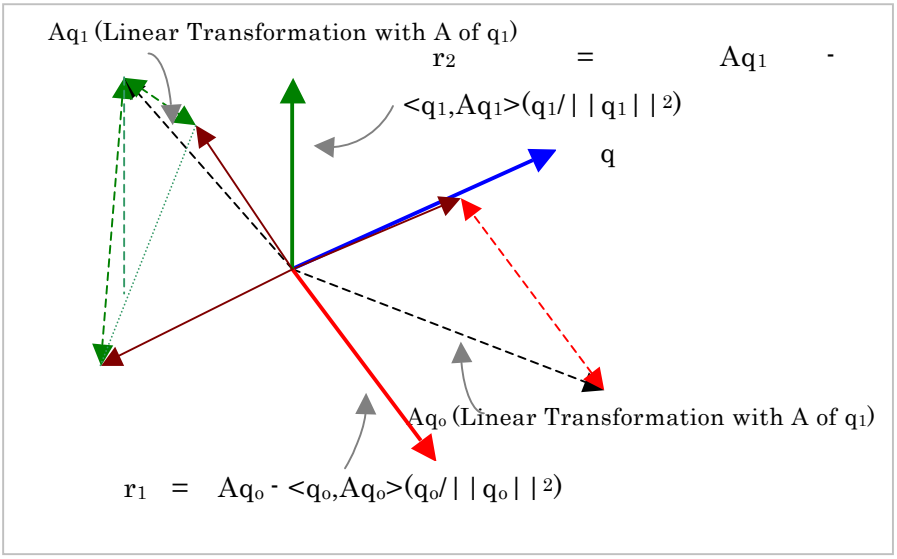


Figure 2.3.4: Vector $\mathbf{q}_2$ Orthogonal to $\mathbf{q}_1$ and $\mathbf{q}_0$

Particularly, the algorithm of the Lanczos method is the Gram-Schmidt orthogonalization for column vectors $\{\mathbf{Aq}_0, \mathbf{A^2q_0}, \mathbf{A^3q_0}, \ldots, \mathbf{A^nq_0}\}$ rephrased from $\{\mathbf{Aq_0}, \mathbf{Aq_1}, \mathbf{Aq_2}, \ldots\}$. This column vector is called the columns of Krylov, and the space created by this is called the Krylov subspace. When the Gram-Schmidt orthogonalization is performed in this space, the vectors can be acquired using the two nearest vectors. This is called the principle of Lanczos.

2.3.7 Tridiagonalization

While repeating the above calculation, the i + 1th calculation can be expressed as follows.

$$\beta_{i+1}q_{i+1} + \alpha_{i+1}q_i + \gamma_{i+1}q_{i-1} = Aq_i \tag{2.3.11}$$

However,

$$\beta_{i+1} = \frac{1}{|r_{i+1}|}, \quad \alpha_{i+1} = \frac{(q_i, Aq_i)}{(q_i, q_i)}, \quad \gamma_{i+1} = \frac{(q_{i-1}, Aq_i)}{(q_{i-1}, q_{i-1})} \tag{2.3.12}$$

When the above equation is indicated in the matrix, it becomes as follows.

$$AQ_m = Q_m T_m \tag{2.3.13}$$

Which becomes,

$$Q_m = \left[q_1 q_2 q_3 \cdots q_m\right] \quad T_m = \begin{pmatrix} \alpha_1 & \gamma_1 & & \\ \beta_2 & \alpha_2 & \gamma_2 & \\ & \cdots & & \\ & & \beta_m & \alpha_m \end{pmatrix} \tag{2.3.14}$$

Thus, the eigenvalue can be acquired by performing the eigenvalue calculation for the tridiagonal matrix acquired by equation (2.3.13).

2.4 Heat Conduction Analysis

The heat conduction analysis method for solids by the finite element method used in this development code is shown in the following.

2.4.1 Basic Equation

The heat conduction equation during the continuum is as follows.

$$\rho \mathrm{c} \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(kxx \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(kyy \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(kxx \frac{\partial T}{\partial z}\right) + Q \tag{2.4.1}$$

However,

$\rho = \rho(x)$ Mass (Density)

$c = c(x,T)$ Specific heat

$T = T(x,t)$ Temperature

$\mathrm{K} = k(x,T)$ Thermal conductivity

$Q = Q(x,T,t)$ Calorific value

Herein, x expresses the position, T is the temperature and t is the time.

The area being considered is S and the periphery is Γ. When assuming the boundary conditions of either the Dirichet type or the Neumann type is given everywhere on Γ, the boundary conditions become as follows.

$$T = T_1(x,t) \qquad X \in \Gamma_1 \tag{2.4.2}$$

$$k\frac{\partial T}{\partial n} = q(x,T,t) \qquad X \in \Gamma_2 \tag{2.4.3}$$

However, the function form T_1, q is already known. q is the heat flux outflow from the boundary. Three types of heat flux can be considered in this program.

$$q = -q_s + q_c + q_r \tag{2.4.4}$$

$$q_s = q_s(x,t) \tag{2.4.5}$$

$$q_c = hc(T - Tc) \tag{2.4.6}$$

$$q_r = hr(T^4 - Tr^4) \tag{2.4.7}$$

q_s is the distributed heat flux, q_c is the heat flux by the convective heat transfer, and q_r is the heat flux by the radiant heat transfer.

However,

$Tc = Tc(x,t)$ Convective heat transfer coefficient ambient temperature

$hc = hc(x,t)$ Convective heat transfer factor

$Tr = Tr(x,t)$ Radiant heat transfer coefficient ambient temperature

$hr = \varepsilon\sigma F = hr(x,t)$ Radiant heat transfer factor

ε: Radiant rate, σ: Stefan-Boltzmann constant, F : Shape factor

2.4.2 Discretization

When equation (2.4.1) is discretized by the Galerkin method, it becomes as follows.

$$[\mathrm{K}]\{T\} + [\mathrm{M}]\frac{\partial T}{\partial t} = \{F\} \tag{2.4.8}$$

However,

$$[\mathrm{K}] = \int\left(kxx\frac{\partial\{N\}^T}{\partial x}\frac{\partial\{N\}}{\partial x} + kyy\frac{\partial\{N\}^T}{\partial y}\frac{\partial\{N\}}{\partial y} + kzz\frac{\partial\{N\}^T}{\partial z}\frac{\partial\{N\}}{\partial z}\right)dV \tag{2.4.9}$$

$$+\int hc\{N\}^T\{N\}ds+\int hr\{N\}^T\{N\}ds$$

$$[\mathrm{M}]=\int \rho c\{N\}^T\{N\}dV \tag{2.4.10}$$

$$\{F\}=\int Q\{N\}^T dV-\int q_s\{N\}^T dS+\int hcTc\{N\}^T dS$$
$$+\int hrTr(T+Tr)(T^2+Tr^2)\{N\}^T dS \tag{2.4.11}$$

$$\{N\}=(N^1,N^2\cdots\cdots),\ Ni=Ni(x)\ :\ \text{Form function} \tag{2.4.12}$$

Equation (2.4.8) is a nonlinear and unsteady equation. Now, when the time is discretized by the backward Euler's rule and the temperature at time $t=t_0$ is already known, the temperature at $t=t_{0+\triangle t}$ is calculated using the following equation.

$$[\mathrm{K}]_{t=t_{0+\triangle t}}\{T\}_{t=t_{0+\triangle t}}+[\mathrm{M}]_{t=t_{0+\triangle t}}\frac{\{\mathrm{T}\}_{t=t_{0+\triangle t}}-\{\mathrm{T}\}_{t=t_0}}{\triangle t}=\{F\}_{t=t_{0+\triangle t}} \tag{2.4.13}$$

The acquisition of a solution with better accuracy $\{T\}^{(i)+1}_{t=to+\triangle t}$, is considered by improving the temperature vector $\{T\}^{(i)}_{t=to+\triangle t}$ which satisfies the approximation of equation (2.4.13).

Therefore, first, the temperature vector is expressed as follows.

$$\{T\}_{t=to+\triangle t}=\{T\}^{(i)}_{t=to+\triangle t}+\{\triangle T\}^{(i)}_{t=to+\triangle t} \tag{2.4.14}$$

The product of the heat conduction matrix and temperature vector, mass matrix and etc. are expressed in approximation as in the following equation.

$$[\mathrm{K}]_{t=t_{0+\triangle t}}\{T\}_{t=t_{0+\triangle t}}\cong[\mathrm{K}]^{(i)}_{t=to+\triangle t}\{T\}^{(i)}_{t=to+\triangle t}$$
$$+\frac{\partial[\mathrm{K}]^{(i)}_{t=to+\triangle t}\{T\}^{(i)}_{t=to+\triangle t}}{\partial\{T\}^{(i)}_{t=to+\triangle t}}\{\triangle T\}^{(i)}_{t=to+\triangle t} \tag{2.4.15}$$

$$[\mathrm{M}]_{t=t_{0+\triangle t}}\cong[\mathrm{M}]^{(i)}_{t=to+\triangle t}+\frac{\partial[\mathrm{M}]^{(i)}_{t=to+\triangle t}}{\partial\{T\}^{(i)}_{t=to+\triangle t}}\{\triangle T\}^{(i)}_{t=to+\triangle t} \tag{2.4.16}$$

When equations (2.4.14) (2.4.15) and (2.4.16) are substituted with equation (2.4.13) and two or more polynomials are omitted, the following equation can be acquired.

$$\left(\frac{[\mathrm{M}]^{(i)}_{t=to+\triangle t}}{\triangle t}+\frac{\partial[\mathrm{M}]^{(i)}_{t=to+\triangle t}\{T\}^{(i)}_{t=to+\triangle t}}{\partial\{T\}^{(i)}_{t=to+\triangle t}}\frac{\{T\}^{(i)}_{t=to+\triangle t}-\{T\}_{t=t0}}{\triangle t}+\frac{\partial[\mathrm{K}]^{(i)}_{t=to+\triangle t}\{T\}^{(i)}_{t=to+\triangle t}}{\partial\{T\}^{(i)}_{t=to+\triangle t}}\right)\{\triangle T\}^{(i)}_{t=to+\triangle} \tag{2.4.17}$$

$$= \{F\}_{t=t0+\Delta t} - [M]^{(i)}_{t=to+\Delta t} \frac{\{T\}^{(i)}_{t=to+\Delta t} - \{T\}_{t=t0}}{\Delta t} - [K]^{(i)}_{t=to+\Delta t} \{T\}^{(i)}_{t=to+\Delta t}$$

Furthermore, an approximation evaluation is performed for the left factor matrix using the following equation.

$$[K^*]^{(i)} = \frac{[M]^{(i)}_{t=to+\Delta t}}{\Delta t} + \frac{\partial [K]^{(i)}_{t=to+\Delta t}}{\partial \{T\}^{(i)}_{t=to+\Delta t}} \{T\}^{(i)}_{t=to+\Delta t}$$

$$= \frac{[M]^{(i)}_{t=to+\Delta t}}{\Delta t} + [K_T]^{(i)}_{t=to+\Delta t} \tag{2.4.18}$$

In this case, $[K_T]^{(i)}_{t=to+\Delta t}$ is the tangent stiffness matrix.

Finally, the temperature at time $t = t0 + \Delta t$ can be calculated by performing an iterative calculation using the following equation.

$$[K^*]^{(i)} \{T\}^{(i)}_{t=to+\Delta t}$$

$$= \{F\}_{t=t0+\Delta t} - [M]^{(i)}_{t=to+\Delta t} \frac{\{T\}^{(i)}_{t=to+\Delta t} - \{T\}_{t=t0}}{\Delta t} - [K]^{(i)}_{t=to+\Delta t} \{T\}^{(i)}_{t=to+\Delta t} \tag{2.4.19}$$

$$\{T\}^{(i+1)}_{t=to+\Delta t} = \{T\}^{(i)}_{t=to+\Delta t} + \{\Delta T\}^{(i)}_{t=to+\Delta t}$$

Particularly, in the analysis for a steady state, the following equation is used to perform the iterative calculation.

$$[K_T]^{(i)} \{\Delta T\}^{(i)}_{t=\infty} = \{F\}_{t=\infty} - [K_T]^{(i)} \{\Delta T\}^{(i)}_{t=\infty}$$

$$\{T\}^{(i+1)}_{t=\infty} = \{T\}^{(i)}_{t=\infty} + \{\Delta T\}^{(i)}_{t=\infty} \tag{2.4.20}$$

Since an implicit method is used for the discretization regarding the time to select time increment Δt in the unsteady state analysis, generally, there are no concerns in the restrictions of the size. However, if time increment Δt is too large, the convergence frequency will increase in the iterative calculation. Therefore, this program is equipped with an automatic increment function to always monitor the size of the residual vectors during the iterative calculation process, and if the convergence of the iterative calculation is too slow, time increment Δt is decreased, and if the iterative calculation frequency decreases, time increment Δt is increased.

2.5 Dynamic Analysis Method

The dynamic problem analysis method with direct time integration applied is described in this section. As shown in the following, time history response analysis with an implicit method and explicit method can be performed in this development code.

2.5.1 Formulation of Implicit Method

Direct time integration is applied to the solution of the equation of motion as shown in the following equation, targeting dynamic problems.

$$\mathbf{M}(t+\Delta t)\ddot{\mathbf{U}}(t+\Delta t)+\mathbf{C}(t+\Delta t)\dot{\mathbf{U}}(t+\Delta t)+\mathbf{Q}(t+\Delta t)=\mathbf{F}(t+\Delta t) \tag{2.5.1}$$

Herein, $\mathbf{M}$ and $\mathbf{C}$ is the mass matrix and attenuation matrix, and $\mathbf{Q}$ and $\mathbf{F}$ are the internal force vector and external force vector. In addition, the change of mass is not considered in this software, and the mass matrix becomes constant regardless of the deformation in non-linearity.

The displacement within time increment Δt, and the change of rate and acceleration are approximated as shown in equation (2.5.2) and equation (2.5.3) using the Newmark-β method.

$$\dot{\mathbf{U}}(t+\Delta t)=\frac{\gamma}{\beta\Delta t}\Delta\mathbf{U}(t+\Delta t)-\frac{\gamma-\beta}{\beta}\dot{\mathbf{U}}(t)-\Delta t\frac{\gamma-2\beta}{2\beta}\ddot{\mathbf{U}}(t) \tag{2.5.2}$$

$$\ddot{\mathbf{U}}(t+\Delta t)=\frac{1}{\beta\Delta t^{2}}\Delta\mathbf{U}(t+\Delta t)-\frac{1}{\beta\Delta t}\dot{\mathbf{U}}(t)-\frac{1-2\beta}{2\beta}\ddot{\mathbf{U}}(t) \tag{2.5.3}$$

Where,

γ,β : Parameter

As it is already known, when γ andβare substituted into the following values, it will match the linear acceleration method, or the trapezoid rule.

γ=1/2、β=1/6 (Linear acceleration method)

γ=1/2、β=1/4 (Trapezoid rule)

When equation (2.5.2) and equation (2.5.3) are substituted with equation (2.5.1), the following equation can be acquired.

$$\begin{aligned}\left(\frac{1}{\beta\Delta t^{2}}\mathbf{M}+\frac{\gamma}{\beta\Delta t}\mathbf{C}+\mathbf{K}\right)\Delta\mathbf{U}(t+\Delta t)&\\ =\mathbf{F}(t+\Delta t)-\mathbf{Q}(t+\Delta t)+\frac{1}{\beta\Delta t}\mathbf{M}\dot{\mathbf{U}}(t)+\frac{1-2\beta}{2\beta}\mathbf{M}\ddot{\mathbf{U}}(t)+\frac{\gamma-\beta}{\beta}\mathbf{C}\dot{\mathbf{U}}(t)&\\ +\Delta t\frac{\gamma-2\beta}{2\beta}\mathbf{C}\ddot{\mathbf{U}}(t)&\end{aligned} \tag{2.5.4}$$

Particularly, when $\mathbf{K}_L$ is assumed as the linear stiffness matrix for a linear problem, the equation becomes $\mathbf{Q}(t+\Delta t)=\mathbf{K}_L\mathbf{U}(t+\Delta t)$. When this equation is substituted with the above equation, the following equation can be acquired.

$$\left\{\boldsymbol{M}\left(-\frac{1}{(\Delta t)^2\beta}\boldsymbol{U}(t)-\frac{1}{(\Delta t)\,\beta}\dot{\boldsymbol{U}}(t)-\frac{1-2\beta}{2\beta}\ddot{\boldsymbol{U}}(t)\right)\right.$$
$$\left.+\boldsymbol{C}\left(-\frac{\gamma}{(\Delta t)\,\beta}\boldsymbol{U}(t)+\left(1-\frac{\gamma}{\beta}\right)\dot{\boldsymbol{U}}(t)+\Delta t\frac{2\beta-\gamma}{2\beta}\ddot{\boldsymbol{U}}(t)\right)\right\} \tag{2.5.5}$$
$$+\left\{\frac{1}{(\Delta t)^2\beta}\boldsymbol{M}+\frac{\gamma}{(\Delta t)\,\beta}\boldsymbol{C}+\boldsymbol{K}_L\right\}\boldsymbol{U}(t+\Delta t)=\boldsymbol{F}(t+\Delta t)$$

In addition, in locations where the acceleration is specified as a geometric boundary condition, the displacement of the following equation can be acquired from equation (2.5.2).

$$u_{is}(t+\Delta t)=u_{is}(t)+\Delta t\,\dot{u}_{is}(t)+(\Delta t)^2\left(\frac{1}{2}-\beta\right)\ddot{u}_{is}(t)+(\Delta t)^2\beta\,\ddot{u}_{is}(t+\Delta t) \tag{2.5.6}$$

Similarly, in locations where the rate is specified, the displacement of the following equation can be acquired from equation (2.76).

$$u_{is}(t+\Delta t)=u_{is}(t)+\Delta t\frac{\gamma-\beta}{\gamma}\dot{u}_{is}(t)+(\Delta t)^2\frac{\gamma-2\beta}{2\gamma}\ddot{u}_{is}(t)+\Delta t\frac{\beta}{\gamma}\dot{u}_{is}(t+\Delta t) \tag{2.5.7}$$

Where,

$u_{is}(t+\Delta t)$: Node displacement at time $t+\Delta t$

$\dot{u}_{is}(t+\Delta t)$: Node rate at time $t+\Delta t$

$\ddot{u}_{is}(t+\Delta t)$: Node acceleration at time $t+\Delta t$

i : Node degree of freedom number (Number of degrees of freedom per 1 - 1 node)

s : Node number

The handling of the mass terms and the attenuation terms are as follows.

(1) Handling of Mass Term

Regarding mass matrices, it is handled as lumped mass matrices as a general rule.

(2) Handling of Attenuation Term

Regarding the attenuation term, it is handled as the Rayleigh attenuation expressed in equation (2.5.8).

$$\boldsymbol{C}=R_m\boldsymbol{M}+R_k\boldsymbol{K}_L \tag{2.5.8}$$

ここで、

R_m, R_k : パラメータ

2.5.2 Formulation of Explicit Method

The equation of motion at time t shown in the following equation is used as a reference in the explicit method.

$$\mathbf{M}\ddot{\mathbf{U}}(t) + \mathbf{C}(t)\dot{\mathbf{U}}(t) + \mathbf{Q}(t) = \mathbf{F}(t) \tag{2.5.9}$$

In this case, when the displacement at time t+Δt and time t-Δt is expressed by the Taylor expansion at time t, and is taken to the second polynomial regardingΔt, the equation becomes as follows.

$$\boldsymbol{U}(t+\Delta t) = \boldsymbol{U}(t) + \dot{\boldsymbol{U}}(t)(\Delta t) + \frac{1}{2!}\ddot{\boldsymbol{U}}(t)(\Delta t)^2 \tag{2.5.10}$$

$$\boldsymbol{U}(t-\Delta t) = \boldsymbol{U}(t) - \dot{\boldsymbol{U}}(t)(\Delta t) + \frac{1}{2!}\ddot{\boldsymbol{U}}(t)(\Delta t)^2 \tag{2.5.11}$$

The following equation can be acquired from the difference and sum of equation (2.83) and equation (2.84).

$$\dot{\boldsymbol{U}}(t) = \frac{1}{2\Delta t}\left(\boldsymbol{U}(t+\Delta t) - \boldsymbol{U}(t-\Delta t)\right) \tag{2.5.12}$$

$$\ddot{\boldsymbol{U}}(t) = \frac{1}{(\Delta t)^2}\left(\boldsymbol{U}(t+\Delta t) - 2\boldsymbol{U}(t) + \boldsymbol{U}(t-\Delta t)\right) \tag{2.5.13}$$

When equation (2.5.12) and equation (2.5.13) are substituted with equation (2.5.9), the following equation can be acquired.

$$\left(\frac{1}{\Delta t^2}\mathbf{M} + \frac{1}{2\Delta t}\mathbf{C}\right)\mathbf{U}(t+\Delta t) = \mathbf{F}(t) - \mathbf{Q}(t) - \frac{1}{\Delta t^2}\mathbf{M}[2\mathbf{U}(t) - \mathbf{U}(t-\Delta t)] - \frac{1}{2\Delta t}\mathbf{C}\mathbf{U}(t-\Delta t) \tag{2.5.14}$$

Particularly, the equation becomes $\mathbf{Q}(t) = \mathbf{K}_L\mathbf{U}(t)$ for a linear problem, and the above equation becomes as follows.

$$\left(\frac{1}{\Delta t^2}\mathbf{M} + \frac{1}{2\Delta t}\mathbf{C}\right)\mathbf{U}(t+\Delta t) = \mathbf{F}(t) - \mathbf{K}_L\mathbf{U}(t) - \frac{1}{\Delta t^2}\mathbf{M}[2\mathbf{U}(t) - \mathbf{U}(t-\Delta t)] - \frac{1}{2\Delta t}\mathbf{C}(t-\Delta t)\mathbf{U} \tag{2.5.15}$$

In this case, when the mass matrix and the attenuation matrix are set as follows, problem solving operations of simultaneous equations is not required in equation (2.5.15).

$\boldsymbol{M}$: Mass matrix
Lumped mass matrix
$\boldsymbol{C}$: Attenuation matrix
Proportional attenuation $\boldsymbol{C} = \mathrm{R_m}\,\boldsymbol{M}$
$\mathrm{R_m}$: Parameter (2.5.16)

Therefore, from equation (2.5.15), $\mathbf{U}(t+\Delta t)$ can be calculated by the following equation.

$$\mathbf{U}(t+\Delta t) = \frac{1}{\left(\frac{1}{\Delta t^2}\mathbf{M} + \frac{1}{2\Delta t}\mathbf{C}\right)}\left\{\mathbf{F}(t) - \mathbf{Q}(t) - \frac{1}{\Delta t^2}\mathbf{M}[2\mathbf{U}(t) - \mathbf{U}(t-\Delta t)] - \frac{1}{2\Delta t}\mathbf{C}(t-\Delta t)\mathbf{U}\right\} \tag{2.5.17}$$

2.6 Frequency Response Analysis

2.6.1 Formulation

When damping is not considered, the equatios of motion for natural frequency analysis are given by

$$\mathbf{M}\,\ddot{\mathbf{U}} + \mathbf{K}\mathbf{U} = \mathbf{0}. \tag{2.6.1}$$

In the frequency domain, the solution of Eq. (2.6.1) can be expressed by

$$\mathbf{U} = \mathbf{U}_j e^{i\omega_j t}. \tag{2.6.2}$$

Substituting Eq. (2.6.2) into Eq. (2.6.1) gives

$$\mathbf{K}\mathbf{U}_j = \omega_j^2 \mathbf{M}\mathbf{U}_j. \tag{2.6.3}$$

The following derivation shows that the natural frequency is a real value. Substituting $\omega_j^2 = \lambda_j$ into Eq. (2.6.3) and complex conjugates are given by

$$\begin{aligned} \mathbf{K}\,\mathbf{U}_{\mathrm{j}} &= \lambda_j \mathbf{M}\mathbf{U}_j \\ \mathbf{K}\overline{\mathbf{U}_j} &= \overline{\lambda_j}\mathbf{M}\overline{\mathbf{U}_j}. \end{aligned} \tag{2.6.4}$$

Multiplying Eq. (2.6.4) by $\overline{\mathbf{U}_j}^{\mathrm{T}}$ gives

$$\begin{aligned} \mathbf{U}_j^{\mathrm{T}}\mathbf{K}\overline{\mathbf{U}_j} &= \overline{\lambda_j}\mathbf{U}_j^{\mathrm{T}}\mathbf{M}\overline{\mathbf{U}_j} \\ \overline{\mathbf{U}_j}^{\mathrm{T}}\mathbf{K}\mathbf{U}_j &= \lambda_j \overline{\mathbf{U}_j}^{\mathrm{T}}\mathbf{M}\mathbf{U}_j. \end{aligned} \tag{2.6.5}$$

Accordingly, we get

$$0 = \left(\lambda_j - \overline{\lambda_j}\right)\overline{\mathbf{U}_j}^{\mathrm{T}}\mathbf{M}\mathbf{U}_j. \tag{2.6.6}$$

Since mass matrix is a positive definite, we have the following relation for non-zero eigen vector.

$$\overline{\mathbf{U}_j}\,\mathbf{M}\mathbf{U}_j > 0 \tag{2.6.7}$$

Consequently, we get

$$\lambda_j = \overline{\lambda_j}. \tag{2.6.8}$$

It also gives that $\omega_j^2 = \lambda_j$ is a real value. Let us consider two distinct eigenvalues.

$$\begin{aligned} \mathbf{K}\mathbf{U}_i &= \lambda_i\,\mathbf{M}\,\mathbf{U}_i \\ \mathbf{K}\mathbf{U}_j &= \lambda_j\,\mathbf{M}\,\mathbf{U}_j \end{aligned} \tag{2.6.9}$$

From Eq. (2.6.9), we get

$$\left(\lambda_i - \lambda_j\right)\mathbf{U}_j^{\mathrm{T}}\mathbf{M}\mathbf{U}_i = 0. \tag{2.6.10}$$

In case the eigen values are distinct, we get

$$\mathbf{U}_j^{\mathrm{T}}\mathbf{M}\,\mathbf{U}_i = 0. \tag{2.6.11}$$

which means that the eigenvectors are said to be orthonormal with respsect to the mass matrix. For the same eigenvectors, it can be easily handled by normalizing with respect to the mass matrix as shown by

$$\mathbf{U}_i^{\mathrm{T}}\,\mathbf{M}\mathbf{U}_i = 1. \tag{2.6.12}$$

In the next derivation, we formulate the case considering a damping matrix in the equations of motion.

$$\mathbf{M}\ddot{\mathbf{U}}+\mathbf{C}\dot{\mathbf{U}}+\mathbf{K}\mathbf{U}=\mathbf{F} \tag{2.6.13}$$

Here we uses Rayleigh damping and the damping matrix can be obtained by

$$\mathbf{C} = \alpha\mathbf{M} + \beta\mathbf{K}. \tag{2.6.14}$$

By using the eigenvector obtained by eigenvalue analysis, the displacement in time domain can be expressed by

$$\mathbf{U}(t) = \sum_i b_i(t)\mathbf{U}_i. \tag{2.6.15}$$

Now, we determine $b_i(t)$ in case the external forces is a harmonic function.

$$\mathbf{F}(t) = \{\mathbf{F}_R + i\mathbf{F}_I\}e^{i\Omega t} \tag{2.6.16}$$

Eq. (2.6.13) can be considered forced vibration

$$b_j(t) = (b_{jR} + b_{jI})e^{i\Omega t} \tag{2.6.17}$$

The real and imaginary parts of $b_i(t)$ are given by

$$b_{jR} = \frac{\mathbf{U}_j{}^{\mathrm{T}}\mathbf{F}_R(\omega_j{}^2 - \Omega^2) + \mathbf{U}_j{}^T\mathbf{F}_I(\alpha + \beta\omega_j{}^2)\Omega}{(\omega_j{}^2 - \Omega^2)^2 + (\alpha + \beta\omega_j{}^2)^2\Omega^2}, \text{ and} \tag{2.6.18}$$

$$b_{jI} = \frac{\mathbf{U}_j{}^{\mathrm{T}}\mathbf{F}_I(\omega_j{}^2 - \Omega^2) - \mathbf{U}_j{}^T\mathbf{F}_R(\alpha + \beta\omega_j{}^2)\Omega}{(\omega_j{}^2 - \Omega^2)^2 + (\alpha + \beta\omega_j{}^2)^2\Omega^2}. \tag{2.6.19}$$

References

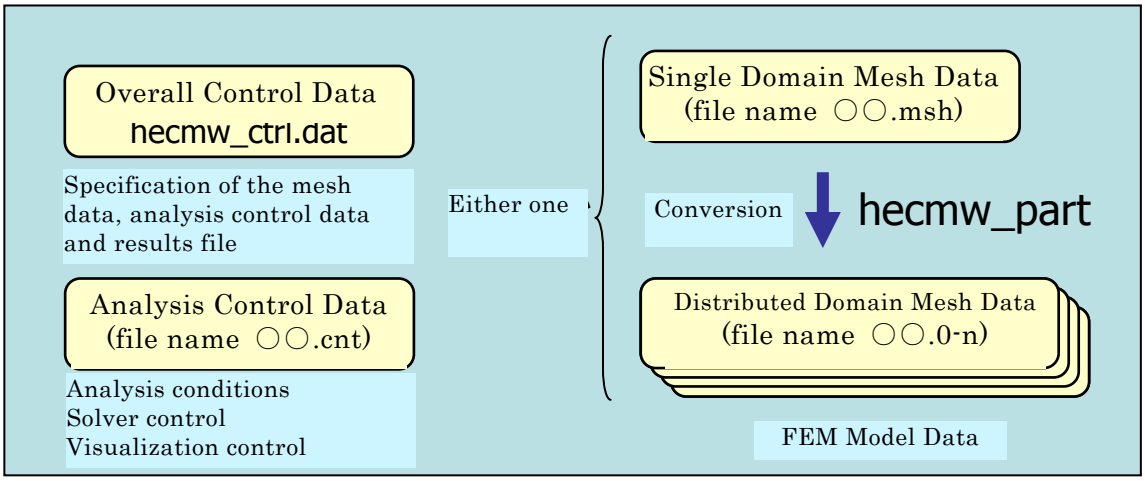
- Hisada, Noguchi: *Foundations and Applications of Nonlinear Finite Element Method*, Maruzen, 1995 (in Japanese)
- O.C.Zienkiewicz, R.L.Taylor: *The Finite Element Method*, 6th Ed., Vol.2: McGraw-Hill, 2005
- JSME Computational Mechanics Handbook (Vol.I) *Finite Element Method (Structure Ver.)*, Japan Society of Mechanical Engineers, 1998 (in Japanese)

・Kyuichiro Washizu, Hiroshi Miyamoto, Yoshiaki Yamada, Yoshiyuki Yamamoto, Tadahiko Kawai: *A Handbook of Finite Element Method*, (I Theory), BAIFUKAN, 1982 (in Japanese)
・Masatake Mori, Masaaki Sugihara, Kazuo Murota: *Linear Computation*, Iwanami Shoten, 1994 (in Japanese)
・Lois Komzsik: *The Lanczos Method Evolution and Application*: Siam, 2003.
・Hayato Togawa: *Vibration Analysis by Finite Element Method*, SAIENSU-SHA Co., Ltd., 1997 (in Japanese)
・Genki Yagawa, Noriyuki Miyazaki: *Thermal Stress, Creep and Heat Transfer Analysis by Finite Element Method*, SAIENSU-SHA Co., Ltd., 1985 (in Japanese)

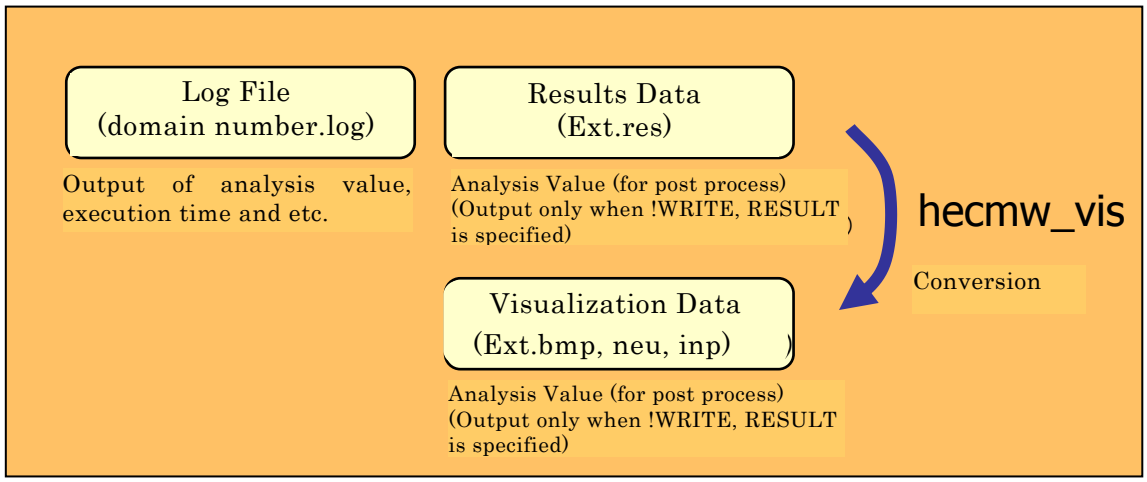
3. Analysis Flow and Input/Output File

3.1 Analysis Flow

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



(a) Input File



(b) 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.

The outline of the above input/output files is described in the following.

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
```

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
```

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 the solver and the control data of the visualizer are also included in the analysis control data.

The details of the analysis control data are 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) conditions
```

```
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
```

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
- Eigenvector values

3.6 Execution Procedure

(1) Preparation of FrontISTR

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

(2) Preparation of Input Files

Prepare three types of input files hecmw_ctrl.dat, analysis control data 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.

(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
```

(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 in four domains is shown in the following.

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

(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 the MPICH2 Manual.

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

(6) Excution 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 analyses.

3.7 Restrictions at Time of Execution

The functions which are normally executed in FrontISTR Ver.3.5 and the element types are shown in Table 3.7.1.

Table 3.7.1: Element List for each Analysis Function

Element No.	Linear Elastic Static Analysis	Eigen value Analysis	Heat Conduction Analysis	Linear Elastic Dynamic Analysis	Frequency Response Analysis	Geometric/Material Nonlinear Static Analysis	Static/Dynamic Contact Analysis
111	×	×	○	×	×	×	×
112	×	×	×	×	×	×	×
231	○	○	○	○	×	×	×
232	○	○	○	○	×	×	×
241	○	○	○	○	×	×	×
242	○	○	○	○	×	×	×
301	○	×	×	○	×	○	○
341	○	○	○	○	○	○	○
342	○	○	○	○	○	○	×
351	○	○	○	○	○	○	○
352	○	○	○	○	○	○	×
361	○	○	○	○	○	○	○
362	○	○	○	○	○	○	×
541	×	×	○	×	×	×	×
542	×	×	×	×	×	×	×
611	○	×	×	○	×	×	×
641	○	×	×	○	×	×	×
731	○	○	○	○	×	×	×
732	×	×	×	×	×	×	×
741	○	○	○	○	×	×	×
743	○	○	×	○	×	×	×
761	○	×	×	×	×	×	×
781	○	×	×	×	×	×	×

Note: ○: Compatible; ×: Not compatible

- In the linear dynamic analysis, element numbers 731, 741 and 743 are not compatible to parallel computing; however, all other element numbers are compatible to parallel computing.
- The direct method is only compatible to parallel computing for contact analysis.
- Element number 611 is not compatible with stress analysis considering thermal stress,

gravity, pressure, or centrifugal loads.

- Element number 641 is not compatible with stress analysis considering pressure and centrifugal loads.
- When analyzing beam elements with solid elements, the comopatible beam element is 641.
- When analyzing shell elements with solid elements, the compatible shell elements are 761 and 781.

4. Element Library and Material Data

4.1 Element Library

In FrontISTR, the element groups shown 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 the 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 element
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
	362	Twenty node hexahedral quadratic element
Interface 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)
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)

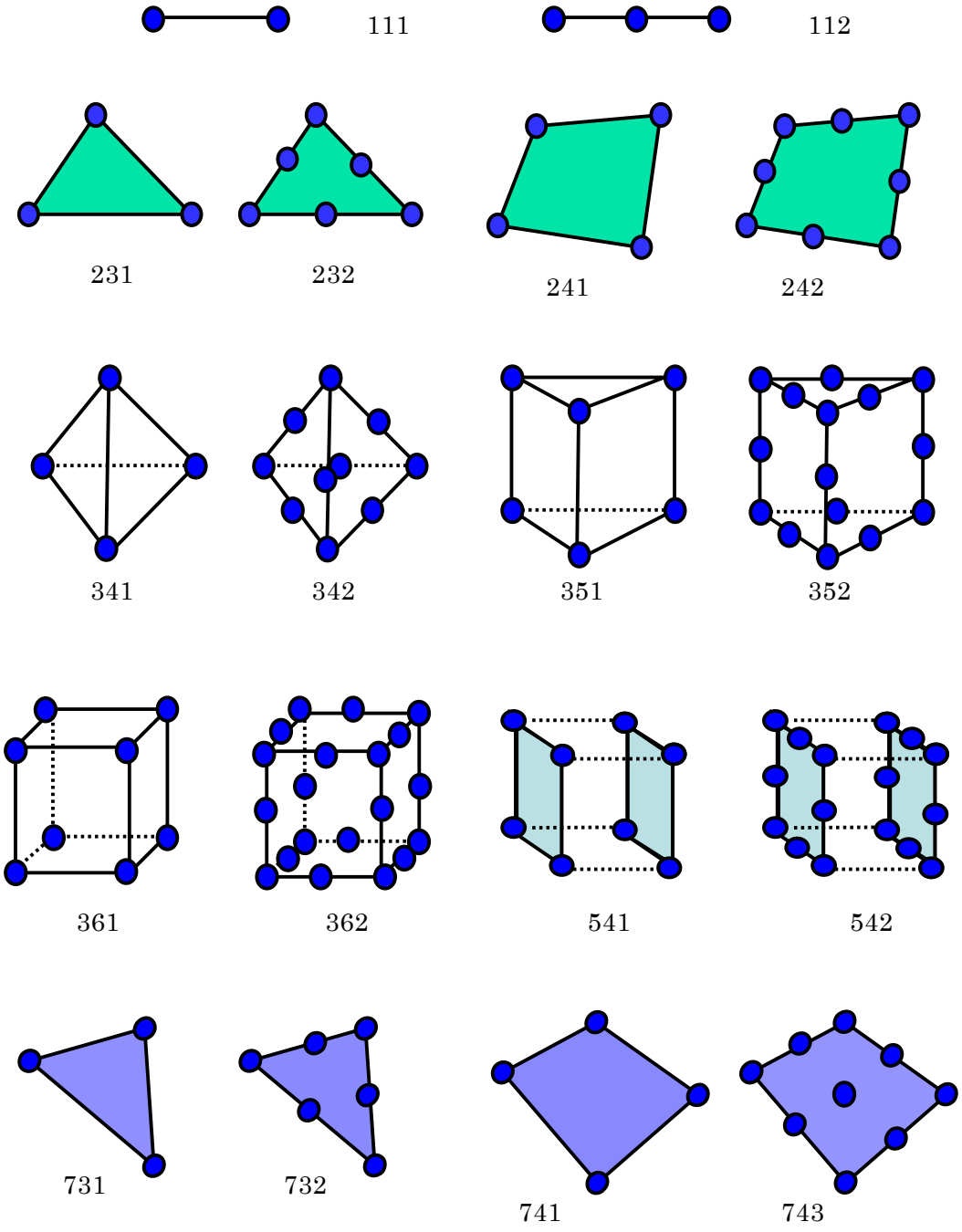
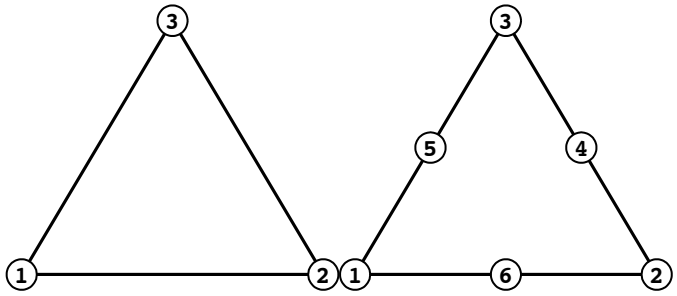


Figure 4.1.1: Element Library

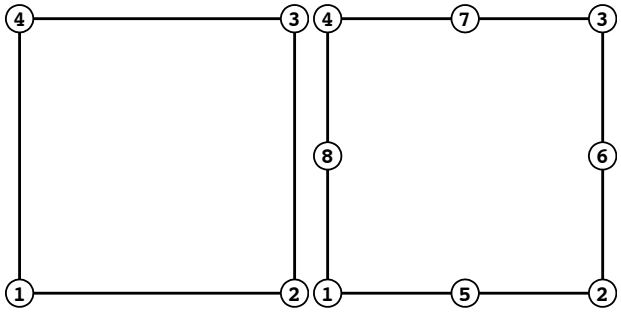
(Line Element)

(Triangular Plane Element)



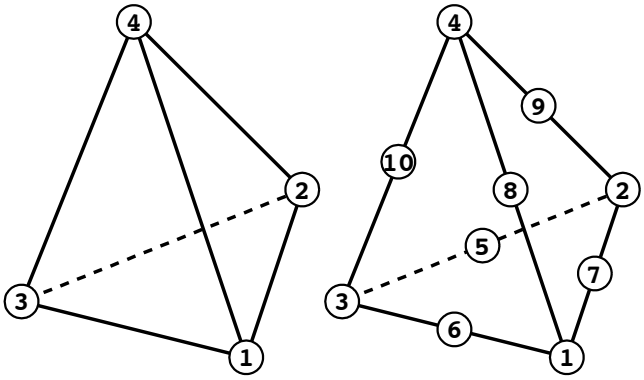
Surface No.	Linear	Quadratic
1	1 - 2	1 - 6 - 2
2	2 - 3	2 - 4 - 3
3	3 - 1	3 - 5 - 1

(Quadrilateral Plane Element)



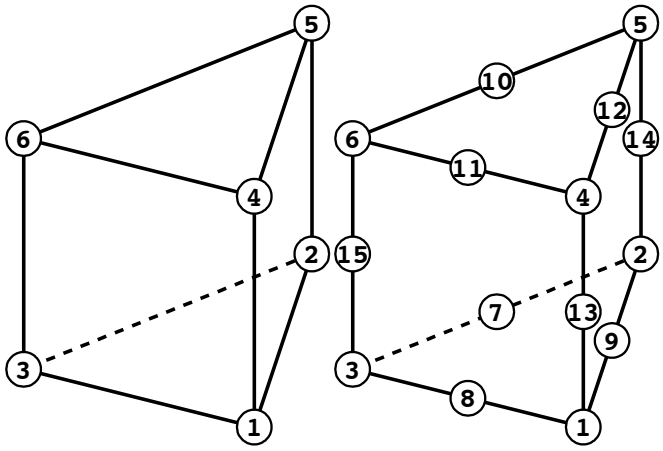
Surface No.	Linear	Quadratic
1	1 - 2	1 - 5 - 2
2	2 - 3	2 - 6 - 3
3	3 - 4	3 - 7 - 4
4	4 - 1	4 - 8 - 1

(Tetrahedral Element)



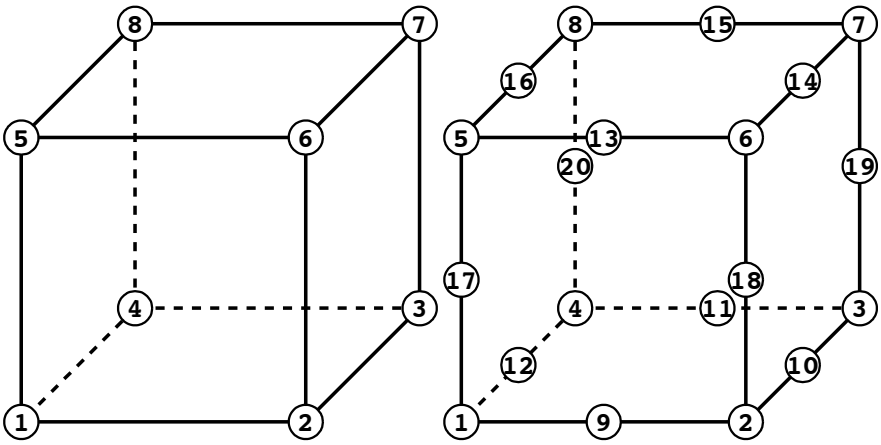
Surface No.	Linear	Quadratic
1	1 - 2 - 3	1 - 7 - 2 - 5 - 3 - 6
2	1 - 2 - 4	1 - 7 - 2 - 9 - 4 - 8
3	2 - 3 - 4	2 - 5 - 3 - 10 - 4 - 9
4	3 - 1 - 4	3 - 6 - 1 - 10 - 4 - 8

(Pentahedral Element)



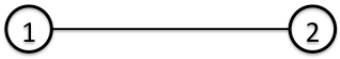
Surface No.	Linear	Quadratic
1	1 - 2 - 3	1 - 9 - 2 - 7 - 3 - 8
2	4 - 5 - 6	4 - 12 - 5 - 10 - 6 - 11
3	1 - 2 - 5 - 4	1 - 9 - 2 - 14 - 5 - 12 - 4 - 13
4	2 - 3 - 6 - 5	2 - 7 - 3 - 15 - 6 - 10 - 5 - 14
5	3 - 1 - 4 - 6	3 - 8 - 1 - 13 - 4 - 11 - 6 - 15

(Hexahedral Element)

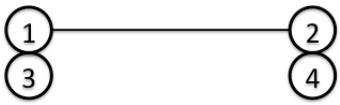


Surface No.	Linear	Quadratic
1	1 - 2 - 3 - 4	1 - 9 - 2 - 10 - 3 - 11 - 4 - 12
2	5 - 6 - 7 - 8	5 - 13 - 6 - 14 - 7 - 15 - 8 - 16
3	1 - 2 - 6 - 5	1 - 9 - 2 - 18 - 6 - 13 - 5 - 17
4	2 - 3 - 7 - 6	2 - 10 - 3 - 19 - 7 - 14 - 6 - 18
5	3 - 4 - 8 - 7	3 - 11 - 4 - 20 - 8 - 15 - 7 - 19
6	4 - 1 - 5 - 8	4 - 12 -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

(Beam Element)

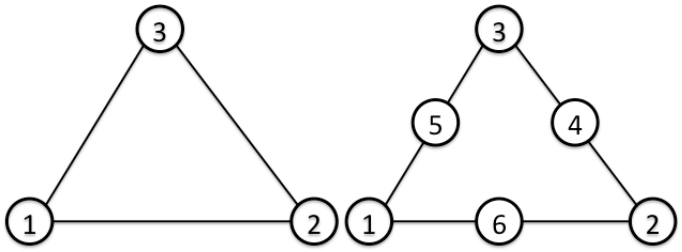


(Beam Element with 3-dof nodes)



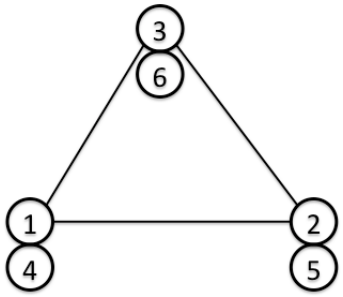
Nodes 1 and 2 for translational DOF, nodes 3 and 4 for rotational DOF.

(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]

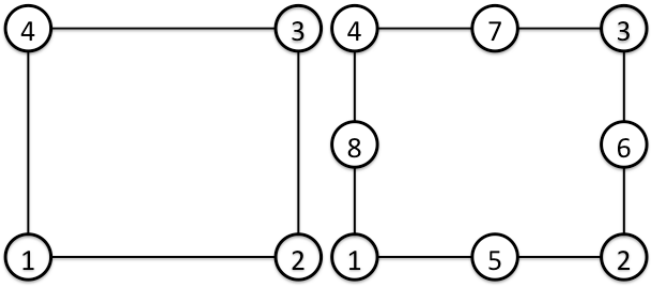
(Triangular Shell Element with 3-dof nodes)



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

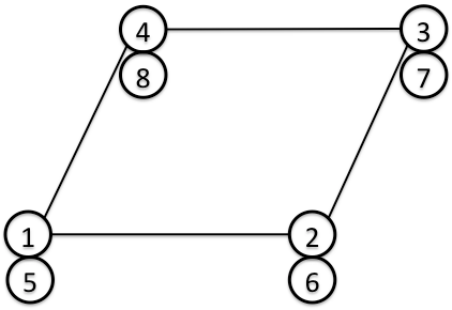
Surface No.	Linear
1	1 - 2 - 3 [front]
2	3 - 2 - 1 [back]

(Quadrilateral Shell Element)



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]

(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

4.2 Material Data

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 values of these materials is defined in the header !SECTION and !MATERIAL of the mesh data. An example of the definition is shown in the following.

(Example)

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

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)

```
!MATERIAL, NAME=M1, ITEM=3        --- Intention of defining three types of property
                                      values in the material of material name M1
!ITEM=1, SUBITEM=2                --- Young's modulus and Poisson's ratio are defined
                                      in !ITEM=1 (mandatory)
4000.,          0.3
!ITEM=2                  --- The mass density must be defined in !ITEM=2 (mandatory in
                             the case of ITEM=3)
8.0102E - 10
!ITEM=3                  --- The coefficient of linear expansion must be defined in !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.

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

Defined in the !SECTION and !MATERIAL header.

(Example)

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

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)

```
!MATERIAL, NAME=M1, ITEM=3          ---  Intention of defining three types of property
                                         values in the material of material name M1
!ITEM=1, SUBITEM=1                  ---  The density and temperature are defined
                                         in !ITEM=1 (mandatory)
7850.,   300.
7790.,   500.
      7700.,   800.
!ITEM=2                             ---  The specific heat and temperature are defined
                                         in !ITEM=2 (mandatory)
0.465,   300.
0.528,   500.
      0.622,   800.
!ITEM=3                             ---  The thermal conductivity and temperature are
                                         defined in !ITEM=3 (mandatory)
43.,     300.
38.6,    500.
      27.7,    800.
```

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

(2) In the case of an interface element

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

(Example)

```
!SECTION, TYPE=INTERFACE, EGRP=GAP          --- Definition of section
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

(3) In the case of a shell element

Defined in the !SECTION and MATERIAL header.

(Example)

```
!SECTION, TYPE=SHELL, EGRP=SH, MATERIAL=M2--- Definition of section
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 θ

Orthotropy angle is specified in [degree].

(Example) Orthotropic, double-layered shell

!MATERIAL, NAME=M1, ITEM=1
!ITEM=1, SUBITEM=17
1, 28600., 0.15, 32.3 , 28600., 12434., 12434., 12434., 0.0, 28600., 0.15, 32.3 , 28600., 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 θ (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 θ (2nd layer)

4.2.3 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
!HYPERELASTIC, TYPE=NEOHOOKE   ---  Definition of the Neo Hooke hyperelastic
                                    material
1000.0, 0.00005                ---  C10 and D are defined (mandatory)
```

(Example) Definition of the elastoplastic material

```
!MATERIAL
!ELASTIC, TYPE=ISOTROPIC           ---  Definition of the isotropic elastic material
21000.0, 0.3                       ---  The Young's modulus and the Poisson's ratio
                                        are defined (mandatory)
!PLASTIC, TYPE=DRUCKER-PRAGER      ---  Definition of the Drucker-Prager plastic
                                        material
500.0, 4.0, 10.0                   ---  Viscosity, angle of friction and hardening
                                        factor are defined (mandatory)
```

5. Overall Control Data

5.1 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.

5.2 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.
The header line can not be described in more than two lines.

<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.

5.3 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) !CONTROL

Specifies the analysis control data file.

1st Line

!CONTROL, NAME=<name>

Parameter	
NAME	Identifier (mandatory)

Parameter Name	Parameter Value	Contents
NAME	fstrCNT	Analysis control data

2nd Line or later

(2nd Line) file

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.)

Example of Use

!CONTROL, NAME=fstrCNT

myctrl.dat

(2) !MESH

Specifies the mesh data file.

1st Line

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

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

2nd Line or later

(2nd Line) fileheader

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.

Example of Use

!MESH, NAME=fstrMSH, TYPE=HECMW-DIST, REFINE=1

Mesh.in

(3) !RESTRAT

Specifies the restart data file.

1st Line

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

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

2nd Line or later

(2nd Line) fileheader

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>.

Example of Use

!RESTART, NAME=restart-in, IO=IN

restart.in

(4) !RESULT

Specifies the analysis results data file.

1st Line

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

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

2nd Line or later

(2nd Line) fileheader

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>.

Example of Use

!RESULT, NAME=fstrRES, IO=OUT, TYPE=BINARY

result.out

(5) !SUBDIR

Specifies storing input/output files into subdirectories

1st Line

!SUBDIR, ON [,optional parameter]

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

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

2nd Line or later

N/A

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 subdirectoris such as TRUNK0, TRUNK1.

Example of Use

!SUBDIR, ON, LIMIT=8000

6. Single Domain Mesh Data

6.1 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.

6.2 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.

6.3 Header List of Single Domain Mesh Data

The single domain mesh data consists of the following headers.

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
!INITIAL CONDITION	Initial conditions	M1-6
!MATERIAL	Material information	M1-7
!NGROUP	Node group	M1-8
!NODE	Node information	M1-9
!SECTION	Section information	M1-10
!SGROUP	Surface group	M1-11
!ZERO	Absolute zero-point	M1-12
!CONTACT PAIR	Contact surface pair	M1-13
!END	Read end	M1-14

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.

<Example of Mesh Data>

```
!HEADER                                                    M1-5
 TEST MODEL CTLR10

!NODE                                                      M1-9
      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                    M1-10
 1.0

!MATERIAL, NAME=M1, ITEM=2                                     M1-7
!ITEM=1, SUBITEM=2
 2.1E6,         0.3
!ITEM=2, SUBITEM=1
 7.8e-3

!NGROUP, NGRP=FIX, GENERATE                                    M1-8
              2 ,              2 ,              1
              3 ,              3 ,              1
              1 ,              1 ,              1
             69 ,             69 ,              1
             67 ,             67 ,              1

!NGROUP, NGRP=CL1                                              M1-9
 50

!END                                                           M1-14
```

(1) !AMPLITUDE (M1-1)

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

1st Line

!AMPLITUDE, NAME=<name> [, optional parameter]

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

2nd Line or later

(2nd Line or later) VAL1, T1, VAL2, T2, VAL3, T3 ... (up to four items in one line)

Hereinafter repeated

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

(2) !EGROUP (M1-2)

Definition of element group

1st Line

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

Parameter	
EGRP	Element group name (mandatory)
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 is 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" are added to
301, 309, 2                              group "NA04".
311, 313
```

(3) !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
```

(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
```

(5) !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.

(6) !INITIAL CONDITION (M1-6)

Definition of temperature initial condition

1st Line

!INITIAL CONDITION, TYPE=<type> [, optional parameter]

Parameter	
TYPE	Type (mandatory)
INPUT	External file name (omissible)

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

2nd Line or later

(2nd Line or later) nod1, VAL1 (One group for one line)

Hereinafter repeated

Parameter Name	Attributions	Contents
nod1	I/C	Node number or node group
VAL1	R	Node value

Example of Use

```
!INITIAL CONDITION, TYPE=TEMPERATURE
101,   25.0
NA01, 38.0
```

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.
- When the same node is redefined, an error will occur.

(7) !MATERIAL (M1-7)

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, and 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)

```
!MATERIAL, NAME=M1, ITEM=3        ---  Intention of defining three types of property
                                       values in the material of material name M1
!ITEM=1, SUBITEM=2                ---  The Young's modulus and Poisson's ratio is
                                       defined in !ITEM=1 (mandatory)
4000.,        0.3
!ITEM=2                           ---  The mass density must be defined in !ITEM=2 (mandatory in
                                       the case of ITEM=3)
8.0102E - 10
!ITEM=3                           ---  The coefficient of linear expansion must be defined
                                       in !ITEM=3
1.0E - 5
```

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)

```
!MATERIAL, NAME=M1, ITEM=3        --- Intention of defining three types of property
                                      values in the material of material name M1
!ITEM=1, SUBITEM= 1               --- The density and temperature are defined in !ITEM=1
                                      (mandatory)
7850.,   300.
7790.,   500.
      7700.,   800.
!ITEM=2, SUBITEM=1                --- The specific heat and temperature are defined
                                      in !ITEM=2 (mandatory)
0.465,   300.
0.528,   500.
      0.622,   800.
!ITEM=3                           --- The thermal conductivity and temperature are
                                      defined in !ITEM=3 (mandatory)
```

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)

!SECTION, TYPE=INTERFACE, EGRP=GAP --- Definition of section

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
```

(8) !NGROUP (M1-8)

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 omitted)

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 and 505" are added to group "NA01".
501, 505
!NGROUP, NGRP= NA02                    "501 and 505" are added to group "NA02".
501, 505
!NGROUP, NGRP= NA04,GENERATE           "301, 303, 305, 307, 309, 311, 312, 313" are added to
301, 309, 2                             group "NA04".
311, 313
```

(9) !NODE (M1-9)

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        Y coordinate is "0.0"
4,                 X, Y and Z coordinates are "0.0"
```

(10) !SECTION (M1-10)

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, hexadedral elements
	SHELL	Shell element
	BEAM	Beam element
	INTERFACE	Interface element
EGRP	<egrp>	Element group name
MATERIAL	<material >	Material name defined by user
SECOPT	<secopt>	= 0: Not specified, plane stress
		= 1: Plane strain
		= 2: Axial symmetry
		= 10: 0 + reduced integration
		= 11: 1 + reduced integration
		= 12: 2 + reduced integration
INPUT	<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
```

(11) !SGROUP (M1-11)

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
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.
```

(12) !ZERO (M1-12)

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
```

(13) !CONTACT PAIR (M1-13)

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
NAME	<name>	Contact pair name
TYPE	NODE-SURF	Slave surface is the node group Master surface is the surface group (default)
	SURF-SURF	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

(14) !END (M1-14)

End of mesh data

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

1st Line

!END

Parameter

N/A

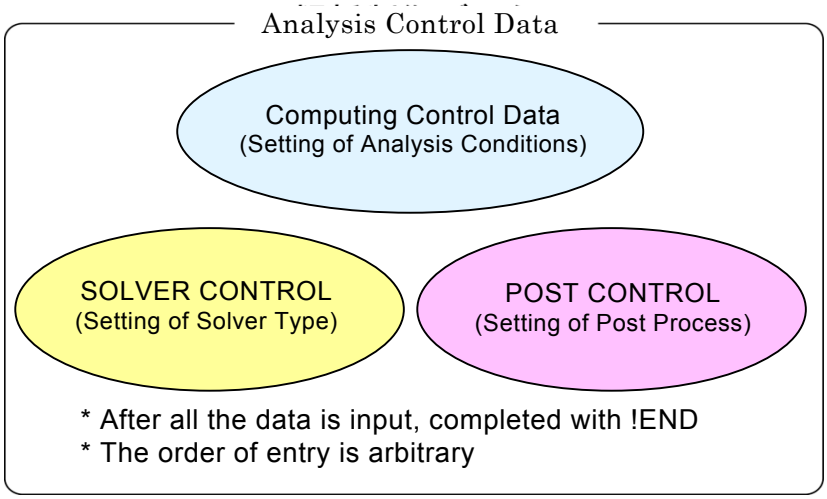
2nd Line or later

N/A

7. Analysis Control Data

7.1 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.



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>

```
### Control File for HEAT solver
!SOLUTION,TYPE=HEAT
!FIXTEMP
 XMIN,     0.0
 XMAX,  500.0                                          (1) Computing control data portion
-----------------------------------------------------------------------------
### Solver Control
!SOLVER,METHOD=1,PRECOND=2,ITERLOG=NO,TIMELOG=NO
 100,    2
```

1.0e-8,1.0,0.0 **(2) Solver control 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
```

(3) Post control (visualization) data portion

7.2 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.

7.3 Analysis Control Data

7.3.1 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 Analyses

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		1-11

	the sction correspondent to		
!END	Ending specification of control data		1-12

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
!USE_MATERIAL	User defined material		2-2-8
!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
!TRS	Tempearture dependent behaviour of viscoelastic material		2-12

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 plane		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

!COUPLE	Coupled surface definition	Required in coupled analysis	5-4
!EIGENREAD	Specification of eigenvalues and eigenvactors	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) Control data common to all analyses

<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                                                   2-3
  FIX, 1,  3,   0.0
!CLOAD                                                      2-4
  CL1,     3,  -1.0
!END                                                        1-12
```

<Description of Header>

1-1 !VERSION

Refers 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

(2) Static analysis control data

<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                                               2-3
  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
```

<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 Degree of Freedom	End No. of Restricted Degree of 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)

Convergence Value Judgment Threshold (Default: 1.0E-06)	No. of Sub Steps (When AMP exists, AMP has priority)	Max No. of Iterative Calculations	Time Function Name (Specified in !AMPLITUDE)
CONVERG	SUBSTEPS	MAXITER	AMP
1.E-5	10	30	

(3) Eigenvalue analysis control data

<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                                                     1-12
```

<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

(4) Heat conduction analysis control data

<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                                               4-9
```

RSURF, R1, 1.0E-9, 800.0
!END 1-12

<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
ELEMENT_GRP_NAME	LOAD_type	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	LOAD_type	Value	Sink
FSURF,	F1,	1.0,	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	Value	Sink
SFSURF,	1.0,	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	LOAD_type	Value	Sink
RSURF,	R1,	1.0E-9,	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	Value	Sink
SRSURF,	1.0E-9,	800.0

(5) Dynamic analysis control data

<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

<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 linear dynamic analysis

Solution of Equation of Motion	Analysis Types
idx_eqa	idx_resp
11	1

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

Parameteryof Newmark-β Method	Parameter β of Newmark-β Method
ganma	beta
0.5	0.25

Type of Mass Matrix	Type of Damping	Parameter R_m of Rayleigh Damping	Parameter Rk of Rayleigh Damping
idx_mas	idx_dmp	ray_m	ray_k
1	1	0.0	0.0

Results Output Interval	Monitoring Node ID or Node Groupe Name	Results Output Interval of Displacement Monitoring
nout	node_monit_1	nout_monit
100	55	1

Output Control Displacement	Output Control Velocity	Output Control Acceleration	Output Control Reaction Force	Output Control Strain	Output Control Stress
iout_list(1)	iout_list(2)	iout_list(3)	iout_list(4)	iout_list(5)	iout_list(6)
0	0	0	0	0	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	DOF_idS	DOF_idE	Value
FIX,	1,	3,	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
NODE_ID	DOF_id	Value
CL1,	3,	-1.0

5-4 !COUPLE, TYPE=1

◆Definition of coupled surface

Coupling Surface Group Name
COUPLING_SURFACE_ID
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	SUBSTEPS	ITMAX
1.E-10		20

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

＜Example of Dynamic analysis(Frequency Reseponse 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.
!FLOAD, LOAD CASE=2
_PickedSet6, 2, 1.

<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	f_end	n_freq	f_disp
14000	16000	20	15000.0

Analysis Start Time	Analysis End Time
t_start	t_end
0.0	6.6e-5

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

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

Output Control	Output Control	Output Control	Output Control	Output Control	Output Control
Displacement	Velocity	Acceleration	ignored	ignored	ignored
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 Name	Degree of Freedom No.	Load Value
NODE_ID	DOF_id	Value
_PickedSet5	2	1.

7.3.2 Solver Control Data

<Example of Solver Control Data>

```
### SOLVER CONTROL
!SOLVER, METHOD=CG, PRECOND=1, ITERLOG=YES, TIMELOG=YES      6-1
  10000,  2                                                  6-2
  1.0e-8,  1.0,  0.0                                         6-3
```

<Description of Header>

* Red figures are the values indicated in the example.

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

6-2

No. of Iterations	Iteration Count of Priconditioning	No. of Krylov Subspaces	No. of Colors for Multi-Color ordering
NIER	iterPREMAX	NREST	NCOLOR_IN
10000	2		

6-3

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

7.3.3 Post Process (Visualization) Control Data

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

<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

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)
			1: Boundary surface
			2: Isosurface 3: Curved surface defined by user based on the

			equation
P1-4	display_method	integer	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
P1-5	color_comp_name	character(100)	Compatible with parameter name and color map (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 vector x: x component y: y component z: 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)
P1-9	iso_number	integer	Specifies the number of isopleth lines. (Default: 5)
P1-10	specified_color	real	Specifies the color when the display_method = 4.
			0.0 <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 DISPLCEMENT)
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: Auto standard_scale = 0.1 * $\sqrt{x\_range^2 + y\_range^2 + z\_range^2}$ /max_deform user_defined: real_scale= standard_scale * deform_scale
P1-15	!initial_style	integer	Specifies the type of deformation display. (Default: 1) 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)
P1-16	!deform_style	integer	Specifies the shape display style after the initial deformation. (Default: 4) 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)
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_AVS: UCD data for AVS COMPLETE_REORDER_AVS: Rearranges the node and element ID SEPARATE_COMPLETE_AVS: For each decomposed domain COMPLETE_MICROAVS: Outputs the physical value scalar FSTR_FEMAP_NEUTRAL: Neutral file for FEMAP

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, … Ex: !position_of_lights=100.0, 200,0, 0.0
P2-5	viewpoint	real(3)	Specifies the viewpoint position by coordinates. (Default: $x = (x_{min} + x_{max})/2.0$ $y = y_{min} + 1.5 *(y_{max} - y_{min})$ $z = z_{min} + 1.5 *(z_{max} - z_{min})$)
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 the 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)
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 (Patitions 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_mapping = [minimum color], [maximum color] If the color_mapping_style = 3; !interval_mapping = [section, compatible color value] . . . repeats number specified Note: Must be described in one line.
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))
P2-21	background _color	real(3)	Specifies the background color. (Default: 0.0, 0.0, 0.0 (Black))
P2-22	isoline_color	read (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.

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_na me	character(4)	When the parameter is a vector, specifies the component to be displayed. (Default: x) norm: Norm of the vector x: x component y: y component z: 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)
P3-5	iso_value	real	Specifies the value of the isosurface.

(In the case of a curved surface (surface_sytle = 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 surface 2. Ellipse curved surface 3. Hyperboloid 4. Paraboloid 5. 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]x^2 + coef[2]y^2 + coef[3]z^2 + coef[4]xy + coef[5]xz + coef[6]yz + coef[7]x + coef[8]y + coef[9]z + coef[10]=0 Example: coef=0.0, 0.0, 0.0, 0.0, 0.0, 0.0, 0.0, 1.0, 0.0, -10.0 This means the plane surface of y=10.0.

7.4 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

7.4.1 Common Control Data

(1) !VERSION (1-1)

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

Example of Use

```
!VERSION
  3
```

(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 → Eigenvalue analysis
ELEMCHECK : Element shape check

Example of Use

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

(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

(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

(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

(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

The following parameter names can be specified.

Parameter Names	Physical Values
DISPLACEMENT	Displacement (Default output)
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
```

(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
DISPLACEMENT	Displacement (Default output)
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

Example of Use

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

(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,
```

(9)　!ECHO　(1-9)

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

Parameter

N/A

(10)　!ORIENTATION　(1-10)

Definition of　local coordinate system

Parameter

NAME　=　Name of lcoal coordinate system

DEFINITION　=　COORDINATE (Default) / NODES

2nd Line or later

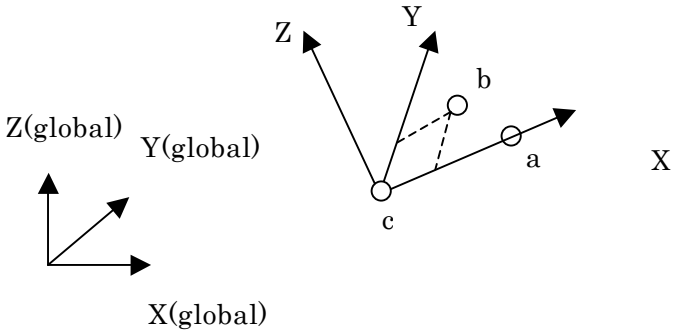
・In case of DEFINTION=COORDINATES

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

・In case of DEFINTIION=NODES

(2nd line or later)　a, b, c

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



(11) !SECTION (1-11)

Definition of local coordinate system the sction 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

(12) !END (1-12)

Displays the end of the control data.

Parameter

N/A

7.4.2 Control Data for Static Analysis

(1) !STATIC (2-1)

Performs the static analysis. (Default: omissible)

Parameter

N/A

(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, !PLASTICTY 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

(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, POISSION, 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, ν12, ν13, ν23, G12, G13, G23, Temperature

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

・In the case of TYPE = USER

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

(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)　ε0, K, n

・In case of HARDEN = RAMBERG-OSGOOD

(2nd line)　ε0, 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, and becomes the default (BILINEAR).

Parameter Name	Attributions	Contents
YIELD0	R	Initial yield stress
H	R	Hardening factor
PSTRAIN	R	Plastic strain
YIELD	R	Yield stress
ε0, K, n	R	$\bar{\sigma}=k(\varepsilon_0+\bar{\varepsilon})^n$
ε0, D, n	R	$\varepsilon=\frac{\sigma}{E}+\varepsilon_0\left(\frac{\sigma}{D}\right)^n$ $\varepsilon=\frac{\sigma}{E}+\varepsilon_0\left(\frac{\sigma}{D}\right)^n$
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 input data, regarding the specified temperature or plastic strain. It is necessary to input the same PSTRAIN array for each temperature.

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

Definition of hyperelastic material

Parameter

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

2nd Line or later

・In case of TYPE = NEOHOOKE

(2nd line)　C_{10}, D

Parameter Name	Attributions	Contents
C_{10}	R	Material constant
D	R	Material constant

・In case of TYPE = MOONEY-RIVLIN

(2nd line)　C_{10}, C_{01}, D

Parameter Name	Attributions	Contents
C_{10}	R	Material constant
C_{01}	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

(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

(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)

(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

(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=PRTHPTROPIC
(2nd line) α11, α22, α33, Temperature

Parameter Name	Attributions	Contents
expansion	R	Coefficient of thermo expansion
α11, α22, α33	R	Coefficient of thermo expansion

Temperature	R	Temperature (required when DEPENDENCIES = 1)

(10) !USER_MATERIAL (2-2-8)

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

(11) !BOUNDARY (2-3)

Definition of displacement boundary conditions

Parameter

GRPID = Group ID

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

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,
```

※Restricted value is 0.0

(12) !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
```

(13)　!CLOAD　(2-4)

Definition of concentrated load

Parameter

GRPID　= Group ID

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

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	Degree of freedom No.
Value	R	Load value

Example of Use

```
!CLOAD, GRPID=1
1, 1, 1.0e3
ALL, 3, 10.0
```

(14)　!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
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, 0.0, 1.0
```

(15) !ULOAD (2-6)

Input of user definition load

Parameter

FILE = file name (Mandatory)

(16) !CONTACT_ALGO (2-7)

Specification of the contact analysis algorithm

Parameter

TYPE = SLAGRANGE : Lagrange multiplier method
ALAGRANGE : Extended Lagrange multiplier method

(17) !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 x1.e3)
TPENALTY = Contact tangential direction Penalty (Default: 1.e3)

2nd Line or later

(2nd line) PAIR_NAME, fcoef, factor

Parameter Name	Attributions	Contents
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
```

(18) !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
```

(19) !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)

(20)　!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	Attributions	Contents
DTIME	R	Time increment value (Default: 1)
ETIME	R	End value of time increment in this step (Default: 1)

(3rd line and later)

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

Example of Use

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

(21) !TRS (2-12)

Thermorheological Simplicity description on temperature behavior of viscoelastic materials

Parameter

DEFINITION = WLF(Default) /ARRHENUS

2nd Line or later

(2nd line) θ_0, C_1, C_2

Parameter Name	Attributions	Contents
θ_0	R	Reference temperature
C_1, C_2	R	Material constants

7.4.3 Control Data for Eigenvalue Analysis

(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
```

7.4.4 Control Data for Heat Conduction Analysis

(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
```

(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
```

(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

Example of Use

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

(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

Parameter 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

Example of Use

```
!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
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

(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

(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.
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

(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
```

(8) !RADIATE (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	Rdiation factor
Sink	R	Ambient temperature

Example of Use

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

Load Parameters

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

(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
```

(10) !WELD_LINE (4-10)

Definition of weld line (Linear)

Parameter

N/A

2nd Line

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

7.4.5 Control Data for Dynamic Analysis

(1) !DYNAMIC (5-1)

Dynamic analysis control

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

Parameter

TYPE = LINEAR / NONLINEAR (Linear dynamic analysis/Nonlinear dynamic analysis)

2nd Line or later

(2nd line) idx_eqa, idx_resp

Parameter Name	Attributions	Contents
idx_eqa	I	Solution of equation of motion (Direct time integration) (Default: 1) 1: Implicit method (Newmark-β method) 11: Explicit method (Center difference method)
idx_resp	I	Analysis type (Default: 1) 1: Time history response analysis

2: 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
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) ganma , beta

Parameter Name	Attributions	Contents
ganma	R	Parameterγof Newmark-β method (Default: 0.5)
beta	R	Parameterβof Newmark-β 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 matrix 2: 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) 0: Not output (Element base and node base), 1: Output 2: Output (Node base) 3: Output (Element base)
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	Attributions	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

(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 (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

!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.

(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		Restricted value (Default: 0)

Example of Use

```
!ACCELERATION, TYPE=TRANSIT, AMP=AMP1
1, 1, 3, 0.0
ALL, 3, 3
※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.

(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)

2nd Line or later

(2nd Line) COUPLING_SURFACE_ID

Parameter Name	Attributions	Contents
SURFACE_ID	C	Surface group name

Example of Use

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

(5) !EIGENREAD(5-5)

Controlling the input file for frequency response analysis

Parameter

N/A

2nd Line

(2nd line) eigenlog_filename

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
```

(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
```

7.4.6 Solver Control Data

(1) !SOLVER (6-1)

Control of solver

Mandatory control data

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, 10, 11, 12）
1, 2: (Block) SSOR
3: (Block) Diagonal Scaling
10: Block ILU(0)
11: Block ILU(1)
12: Block ILU(2)
10, 11, 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.

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

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)

2nd Line or later

(2nd Line) NIER, iterPREmax, NREST

Parameter Name	Attributions	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 and Diagonal scaling, 2 for parallel computation with other preconditioner than Diagonal scaling)
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	Attributions	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, ITER=YES, TIME=YES
10000, 2
1.0e-8, 1.0, 0.0
```

7.4.7 Post Process (Visualization) Control Data

(1) !VISUAL (P1-0)

Specifies the visualization method.

METHOD = PSR : Surface rendering

visual_start_step : Specification of time step number which starts the visualization process (Default: 1)

visual_end_step : Specification of time step number which ends the visualization process (Default: All)

visual_interval_step : Specification of time step interval which performs the visualization

process (Default: 1)

(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.

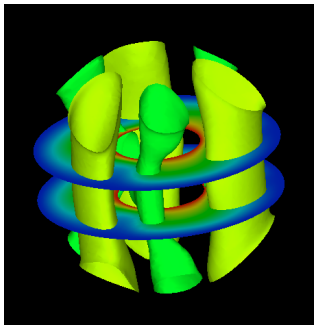


Figure 7.4.1: Example of surface_num Setting

!surface (P1-2)

Sets the contents of the surface.

Ex.: The contents of the four surfaces in Figure 4.1.2 are as follows.

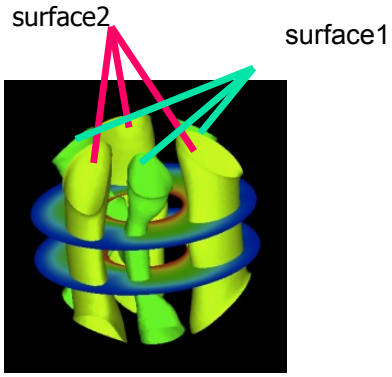


Figure 7.4.2: Example of Surface Setting

```
!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 surfaces

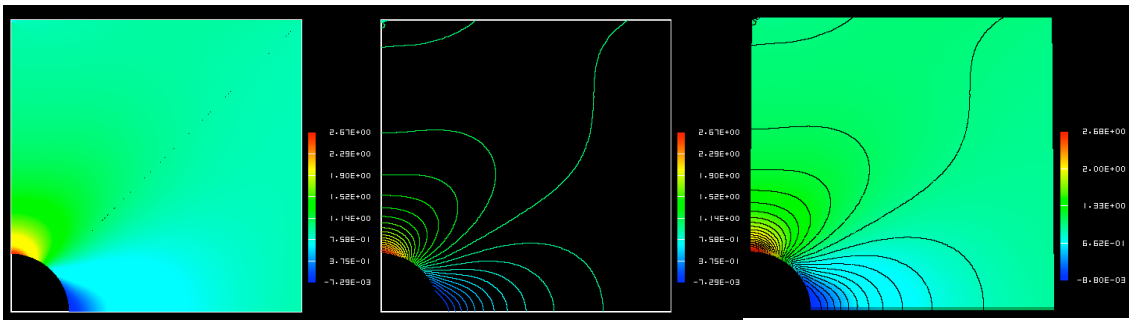
$$coef[1]x2 + coef[2]y2 + coef[3]z2 + coef[4]xy + coef[5]xz + coef[6]yz + coef[7]x + coef[8]y + coef[9]z + coef[10]=0$$

Figure 7.4.3: Example of surface_style Setting

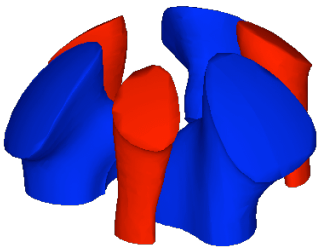
(3) !display_method (P1-4)

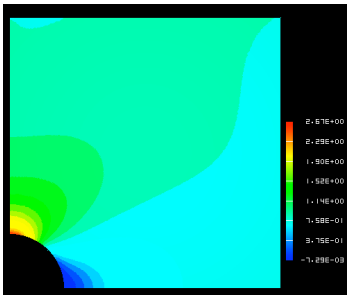
Display method (Default: 1)

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



!display_method=1 !display_method=2 !display_method=3





!display_method=4 !display_method=5

Figure 7.4.4: Example of display_method Setting

(4) !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

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 : εx 2 : εy 3 : εz

4 : εxy 5 : εyz 6 : εzx

When !color_comp_name=STRESS is specified

1 : σx 2 : σy 3 : σz

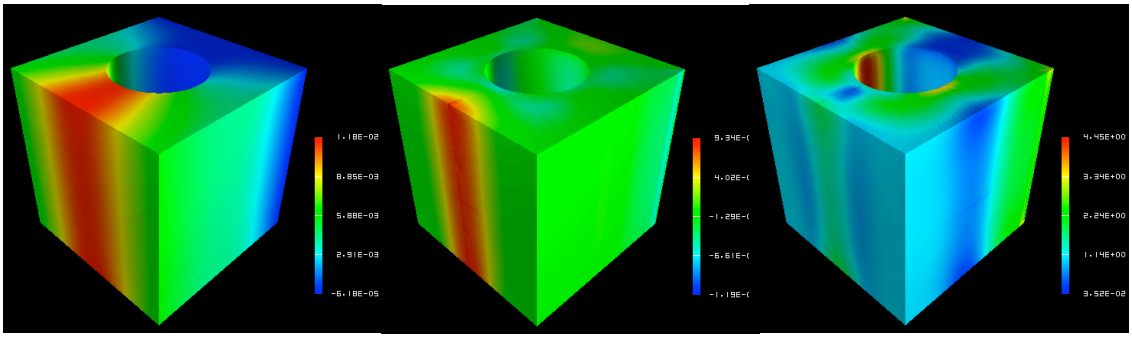
4 : τxy 5 : τyz 6 : τzx

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



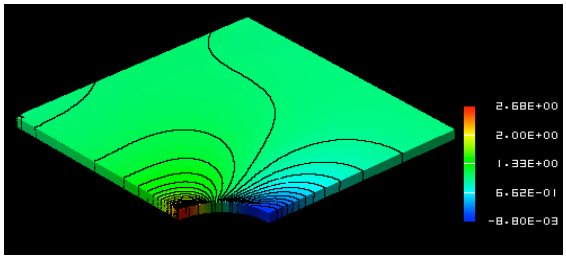
!color_comp_name=displacement !color_comp_name=strain !color_comp = 3

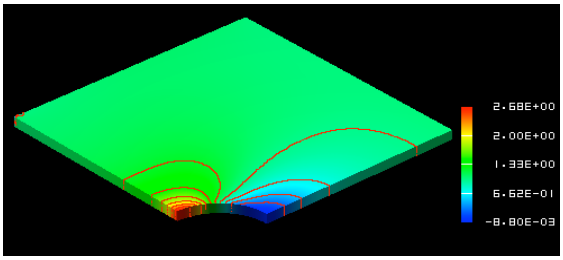
!color_subcomp = 1 !color_subcomp_name = 1 !color_subcomp = 7

Figure 7.4.5: Example of color_comp, color_subcomp and color_comp_name Setting

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

When display_method=2, 3 or 5





!isoline_number = 30
!isoline_color = 0.0, 0.0, 0.0

!isoline_number = 10
!isoline_color = 1.0, 0.0, 0.0

Figure 7.4.6: Example of isoline_number and isoline_color Setting

(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)

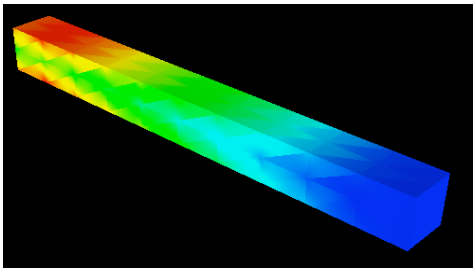
4: Dotted line mesh (Displayed in blue if not specified)

(7) !deform_scale (P1-14)

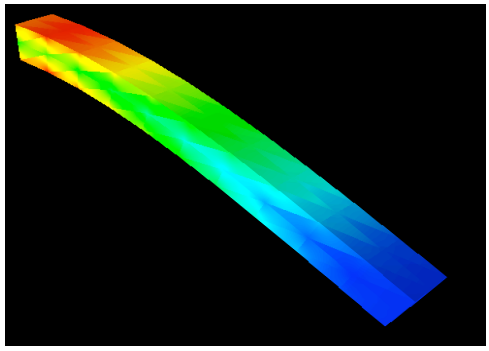
Specifies the displacement scale when displaying deformation.

Default: Auto

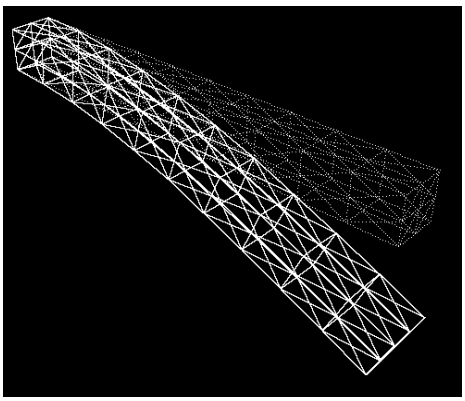
standard_scale = 0.1 * $\sqrt{x\_range^2 + y\_range^2 + z\_range^2}$ /max_deform



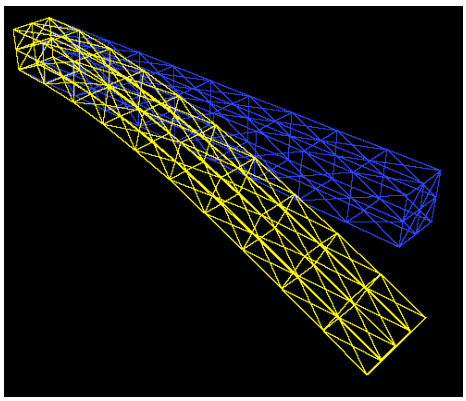
!initial_style=2
!deform_style = 0



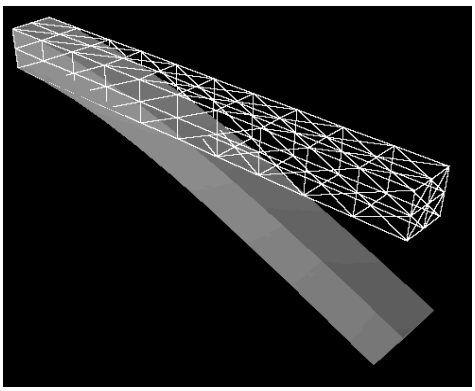
!initial_style=0
!deform_style = 2



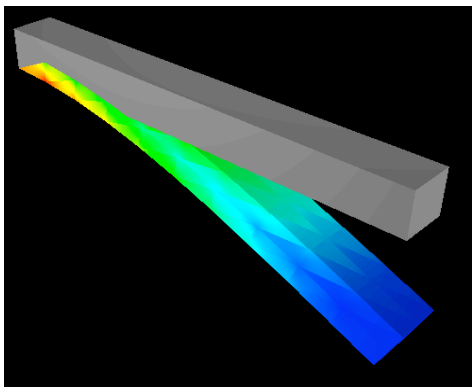
!initial_style=4
!deform_style = 1
!initial_line_color = 1.0, 1.0, 1.0



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

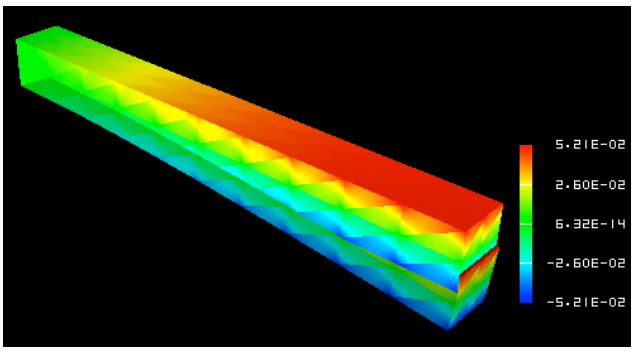


!initial_style=1
!deform_style = 2

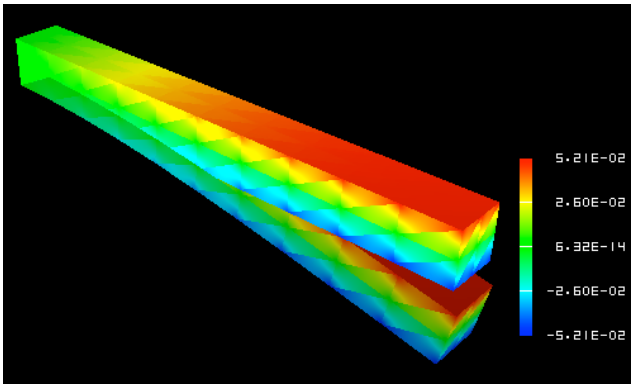


!initial_style=2
!deform_style = 3

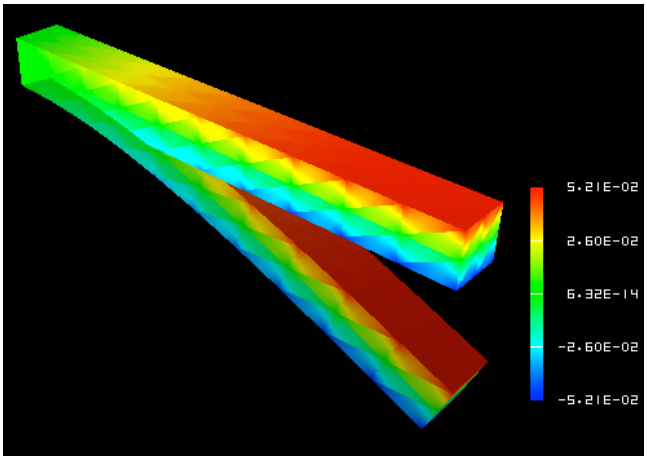
Figure 7.4.7: Example of Display Styles Setting



!deform_scale=1.0



!deform_scale=2.0



!deform_scale=4.0

Figure 7.4.8: 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

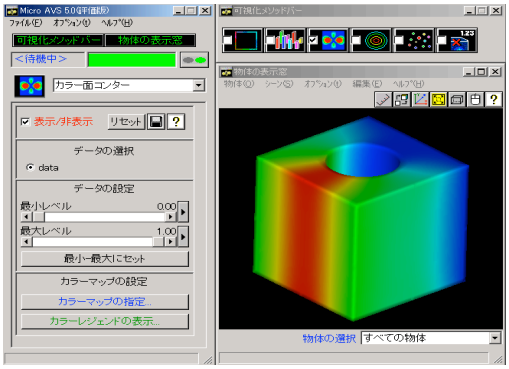
COMPLETE_REORDER_AVS: Rearranges the node and element ID in the UCD data for AVS

SEPARATE_COMPLETE_AVS: UCD data for AVS for each decomposed domain

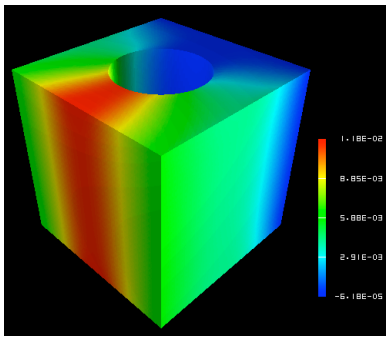
COMPLETE_MICROAVS: Outputs the physical values in the scalar in the UCD data for AVS

BIN_COMPLETE_AVS: Outputs COMPLETE_AVS in binary format

FSTR_FEMAP_NEUTRAL: Neutral file for FEMAP



!output_type = AVS

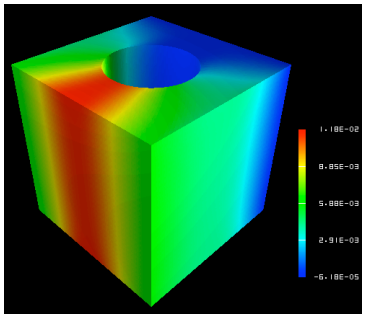


!output_type=BMP

Figure 7.4.9: Example of output_type

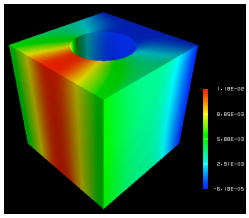
(9)　!x_resolution　!y_resolution (P2-1　P2-2)

Specifies the resolution when output_type=BMP.



!x_resolution=500

!y_resolution=500



!x_resolution=300

!y_resolution=300

Figure 7.4.10: Example of x_resolution and y_resolution Setting

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

viewpoint: Specifies the viewpoint position by coordinates.

Default: x = (xmin + xmax)/2.0,

y = ymin + 1.5 *(ymax – ymin),

z = zmin + 1.5 *(zmax – 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 × z axis

Y-axis: z axis × x axis

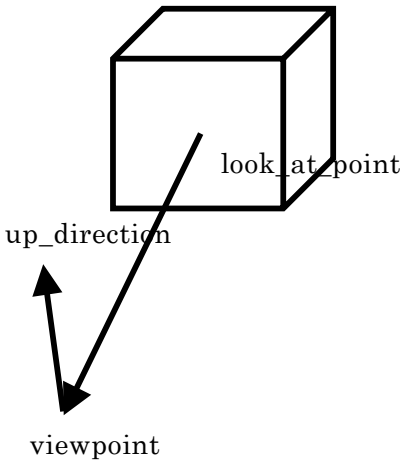


Figure 7.4.11: View Frame Determination Method

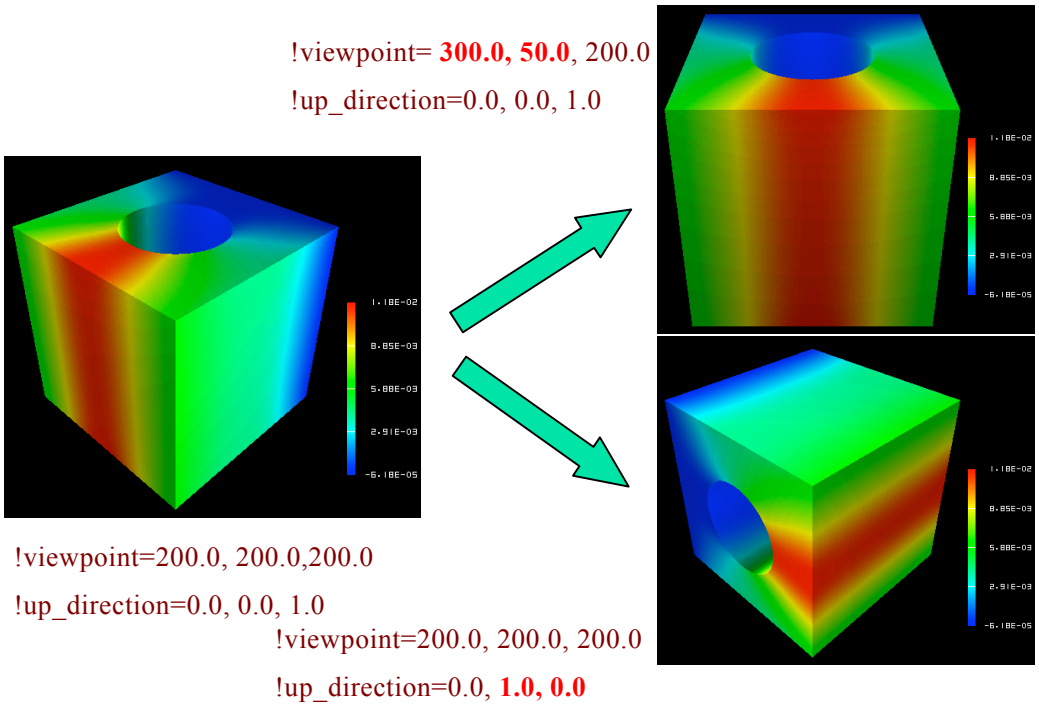


Figure 7.4.12: Example of !viewpoint, look_at_point and up_direction Setting

(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.

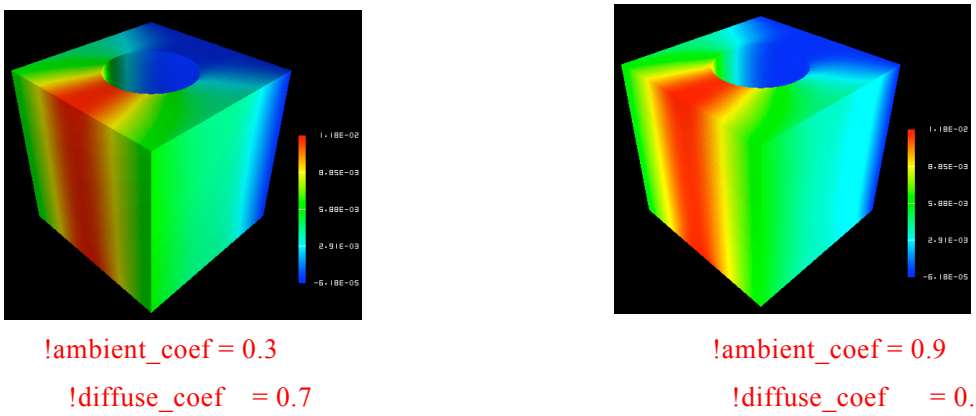


Figure 7.4.13: Example of Lighting Model Parameter Setting

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

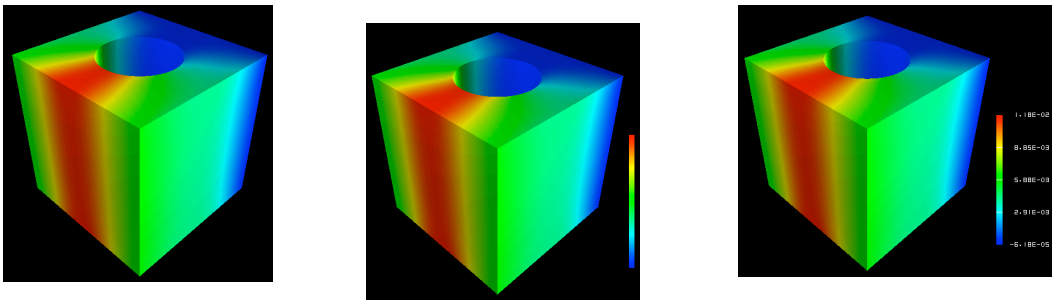
!color_mapping_bar_on: Specifies whether to display the color mapping bar.

0: off 1: on (Default: 0)

!scale_marking_on: Specifies the existence of memory of the color mapping bar.

0: off 1: on (Default : 0)

!num_of_scales: Specifies the number of memory. (Default: 3)



!color_mapping_bar_on=0
!scale_marking_on =0

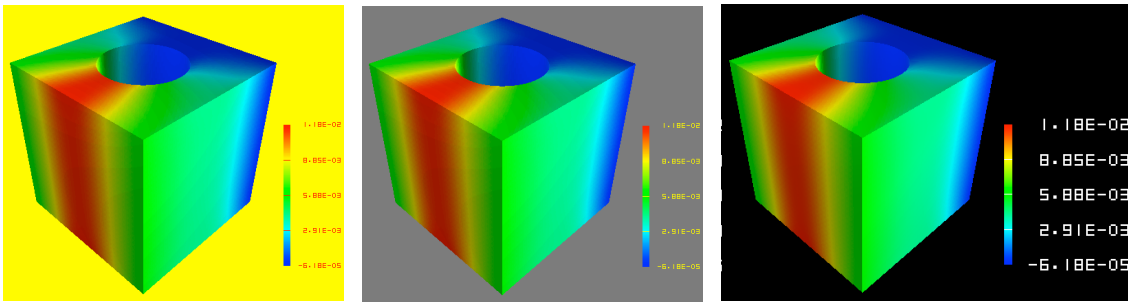
!color_mapping_bar_on =1
!scale_markig_on =0

!color_mapping_bar_on =1
!scale_marking_on=1
!num_of_scale = 5

Figure 7.4.14: Example of Color Mapping Bar Display

(13) !font_size !font_color !backgroud_color (P2-19 P2-20 P2-21)

Specifies the background color and character font.



!background_color =1.0,1.0,0.0
!font_color=1.0, 0.0, 0.0
!font_size=1.5

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

!background_color =0.0, 0.0,0.0
!font_color=1.0, 1.0, 1.0
!font_size=2.5

Figure 7.4.15: Example of Background and Font Setting

(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.

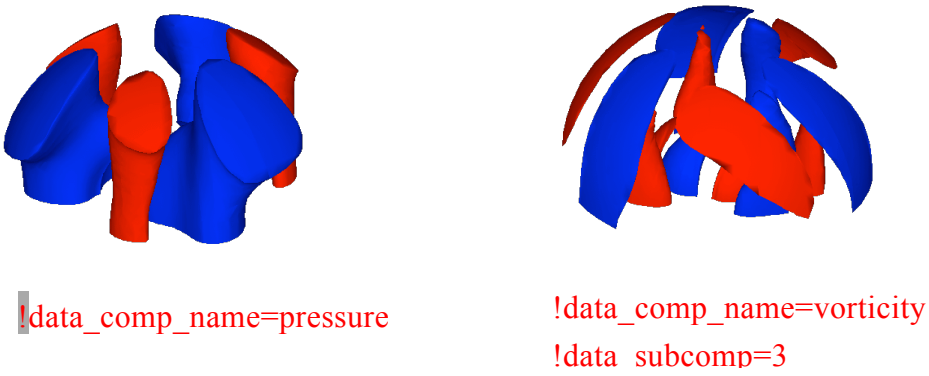


Figure 7.4.16: Example of data_comp, data_subcomp and data_comp_name Setting

(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
!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.

8. 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.

8.1 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 maximum

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

.....

8.2 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) 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

(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)

8.3 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) 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

(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

8.4 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) 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

(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
dtime: Time increment

8.5 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) 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.

(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

(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

9. Example Verification

9.1 Verification by Simple Geometric Model

(1) Elastic static analysis

The subject in this verification was a cantilever beam with mesh partitioning as shown in Figure 9.1.1. Regarding the verification conditions, the analysis was performed with 7 conditions from exA - exG where the load conditions were changed as shown in Figure 9.1.2. In addition, this is a verification case where a direct method solver was used, with the same load conditions as exA being applied to exG.

The verification results for each case are shown in Table 9.1.1 ~ Table 9.1.7.

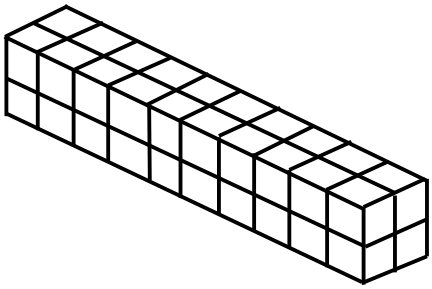
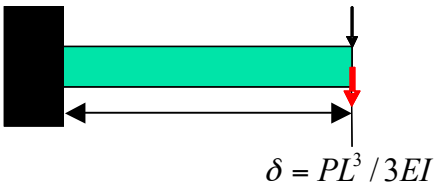
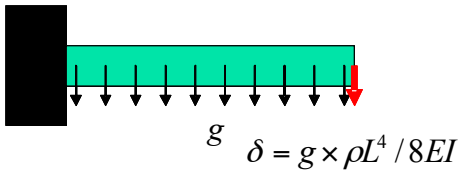


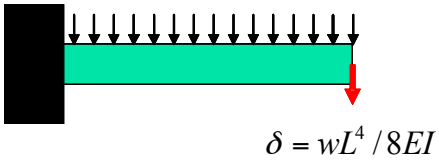
Figure 9.1.1: Example of Mesh Partitioned Cantilever Beam (Hexahedral Element)



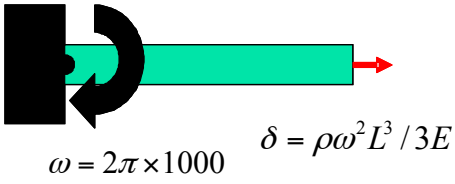
(a) exA, G: Concentrated load



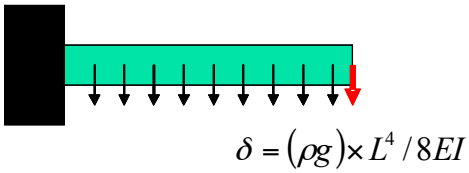
(b) exD: Gravity



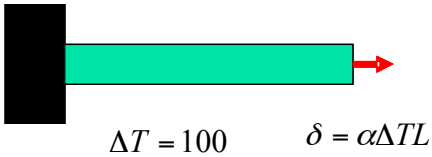
(c) exB: Surface distributed load



(d) exE: Centrifugal force



(e) exC: Volumetric load



(f) exF: Heat load

Young's Modulus: E = 4000.0 kgf/mm^2

Poisson's Ratio: v = 0.3

Mass density: ρ=8.0102x10^{-10} kgs^2/mm^4

Geometrical moment of inertia: I = 1.0/12.0mm^4

Gravitational acceleration: g = 9800.0 mm/s^2

Linear thermal expansion coefficient: α= 1.0x10^{-5}

Length: L = 10.0 mm

Cross-sectional area: A = 1.0 mm^2

Figure 9.1.2: Verification Conditions of Cantilever Beam Model

Table 9.1.1: exA: Verification Results of Concentrated Load Problem

Case Name	No. of Elements	Predicted Value: δmax = -1.000			Remarks
		NASTRAN	ABAQUS	FrontISTR	
A231	40	-0.338	-0.371	-0.371	33 nodes / plane stress problem
A232	40	-0.942	-1.002	-1.002	105 nodes / plane stress problem
A241	20	-0.720	-0.711	-0.711	33 nodes / plane stress problem
A242	20	-0.910	-1.002	-1.002	85 nodes / plane stress problem
A341	240	-0.384	-0.384	-0.386	99 nodes
A342	240	-0.990	-0.990	-0.999	525 nodes
A351	80	-0.353	-0.355	-0.351	99 nodes
A352	80	-0.993	-0.993	-0.992	381 nodes
A361	40	-0.954	-0.985	-0.984	99 nodes
A362	40	-0.994	-0.993	-0.993	330 nodes
A731	40	-	-	-0.991	33 nodes / direct method
A741	20	-	-	-0.996	33 nodes / direct method

Table 9.1.2: exB: Verification Results of Surface Distributed Load Problem

Case Name	No. of Elements	Predicted Value:δmax=-3.750			Remarks
		NASTRAN	ABAQUS	FrontISTR	
B231	40	-1.281	-1.403	-1.403	33 nodes / plane stress problem
B232	40	-3.579	-3.763	-3.763	105 nodes / plane stress problem
B241	20	-3.198	-2.680	-2.680	33 nodes / plane stress problem
B242	20	-3.426	-3.765	-3.765	85 nodes / plane stress problem
B341	240	-1.088	-1.449	-1.454	99 nodes
B342	240	-3.704	-3.704	-3.748	525 nodes
B351	80	-3.547	-1.338	-1.325	99 nodes
B352	80	-.3717	-3.716	-3.713	381 nodes
B361	40	-3.557	-3.691	-3.688	99 nodes
B362	40	-3.726	-3.717	-3.717	330 nodes
B731	40	-	-	-3.722	33 nodes / direct method
B741	20	-	-	-3.743	33 nodes / direct method

Table 9.1.3: exC: Verification Results of Volumetric Load Problem

Case Name	No. of Elements	Predicted Value: δmax= -2.944e-5			Remarks
		NASTRAN	ABAQUS	FrontISTR	
C231	40	-	-1.101e-5	-1.101e-5	33 nodes / plane stress problem
C232	40	-	-2.951e-5	-2.951e-5	105 nodes / plane stress problem
C241	20	-	-2.102e-5	-2.102e-5	33 nodes / plane stress problem
C242	20	-	-2.953e-5	-2.953e-5	85 nodes / plane stress problem
C341	240	-	-1.136e-5	-1.140e-5	99 nodes
C342	240	-	-2.905e-5	-2.937e-5	525 nodes
C351	80	-	-1.050e-5	-1.039e-5	99 nodes

C352	80	-	-2.914e-5	-2.911e-5	381 nodes
C361	40	-	-2.895e-5	-2.893e-5	99 nodes
C362	40	-	-2.915e-5	-2.915e-5	330 nodes
C731	40	-	-	-2.922e-5	33 nodes / direct method
C741	20	-	-	-2.938e-5	33 nodes / direct method

Table 9.1.4: exD: Verification Results of Gravity Problem

Case Name	No. of Elements	Predicted Value: δmax= -2.944e-5			Remarks
		NASTRAN	ABAQUS	FrontISTR	
D231	40	-1.101e-5	-1.101e-5	-1.101e-5	33 nodes / plane stress problem
D232	40	-2.805e-5	-2.951e-5	-2.951e-5	105 nodes / plane stress problem
D241	20	-2.508e-5	-2.102e-5	-2.102e-5	33 nodes / plane stress problem
D242	20	-2.684e-5	-2.953e-5	-2.953e-5	85 nodes / plane stress problem
D341	240	-1.172e-5	-1.136e-5	-1.140e-5	99 nodes
D342	240	-2.906e-5	-2.905e-5	-2.937e-5	525 nodes
D351	80	-1.046e-5	-1.050e-5	-1.039e-5	99 nodes
D352	80	-2.917e-5	-2.914e-5	-2.911e-5	381 nodes
D361	40	-2.800e-5	-2.895e-5	-2.893e-5	99 nodes
D362	40	-2.919e-5	-2.915e-5	-2.915e-5	330 nodes
D731	40	-	-	-2.922e-5	33 nodes / direct method
D741	20	-	-	-2.938e-5	33 nodes / direct method

Table 9.1.5: exE: Verification Results of Centrifugal Force Problem

Case Name	No. of Elements	Predicted Value: δmax= 2.635e-3			Remarks
		NASTRAN	ABAQUS	FrontISTR	
E231	40	2.410e-3	2.616e-3	2.650e-3	33 nodes / plane stress problem
E232	40	2.447e-3	2.627e-3	2.628e-3	105 nodes / plane

					stress problem
E241	20	2.386e-3	2.622e-3	2.624e-3	33 nodes / plane stress problem
E242	20	2.387e-3	2.627e-3	2.629e-3	85 nodes / plane stress problem
E341	240	2.708e-3	2.579e-3	2.625e-3	99 nodes
E342	240	2.639e-3	2.614e-3	2.638e-3	525 nodes
E351	80	2.642e-3	2.598e-3	2.625e-3	99 nodes
E352	80	2.664e-3	2.617e-3	2.616e-3	381 nodes
E361	40	2.611e-3	2.603e-3	2.603e-3	99 nodes
E362	40	2.623e-3	2.616e-3	2.616e-3	330 nodes
E731	40	-	-	2.619e-3	33 nodes / direct method
E741	20	-	-	2.622e-3	33 nodes / direct method

Table 9.1.6: exF: Verification Results of Thermal Stress Load Problem

Case Name	No. of Elements	Predicted Value: δmax= 1.000e-2			Remarks
		NASTRAN	ABAQUS	FrontISTR	
F231	40	-	1.016e-2	1.007e-2	33 nodes / plane stress problem
F232	40	-	1.007e-2	1.007e-2	105 nodes / plane stress problem
F241	20	-	1.010e-2	1.010e-2	33 nodes / plane stress problem
F242	20	-	1.006e-2	1.006e-2	85 nodes / plane stress problem
F341	240	-	1.047e-2	1.083e-2	99 nodes
F342	240	-	1.018e-2	1.022e-2	525 nodes
F351	80	-	1.031e-2	1.062e-2	99 nodes
F352	80	-	1.015e-2	1.017e-2	381 nodes
F361	40	-	1.026e-2	1.026e-2	99 nodes
F362	40	-	1.016e-2	1.016e-2	330 nodes

Table 9.1.7: exG: Verification Results of Direct Method (Concentrated Load Problem)

Case Name	No. of Elements	Predicted Value: δmax=-1.000			Remarks
		NASTRAN	ABAQUS	FrontISTR	
G231	40	-0.338	-0.371	-0.371	33 nodes / plane stress problem
G232	40	-0.942	-1.002	-1.002	105 nodes / plane stress problem
G241	20	-0.720	-0.711	-0.711	33 nodes / plane stress problem
G242	20	-0.910	-1.002	-1.002	85 nodes / plane stress problem
G341	240	-0.384	-0.384	-0.386	99 nodes
G342	240	-0.990	-0.990	-0.999	525 nodes
G351	80	-0.353	-0.355	-0.351	99 nodes
G352	80	-0.993	-0.993	-0.992	381 nodes
G361	40	-0.954	-0.985	-0.984	99 nodes
G362	40	-0.994	-0.993	-0.993	330 nodes
G731	40	-	-	-0.991	33 nodes
G741	20	-	-	-0.996	33 nodes

(2) Nonlinear static analysis

(2-1) exnl1: Geometrical nonlinear analysis

The same model of verification case exA - G was used for the verification model of verification case exI. The conceptual diagram of the verification model is shown in Figure 9.1.3. A geometric nonlinear analysis is implemented in this model. The verification results are shown in Table 9.1.8. A nonlinear calculation of the load increment value of 0.1P for 10 steps is implemented for the final load of 1.0P.

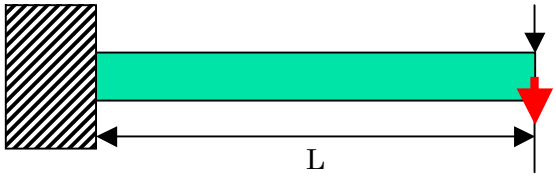


Figure 9.1.3: Verification Model

Table 9.1.8: exI: Verification Results (Maximum Deflection Amount History)

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

(2-2) exnl2: Elastoplasticity deformation analysis

The National Agency for Finite Element Methods and Standards (U.K.): Test NL1 from NAFEMS was referred to in this verification problem, and incorporated the geometrical non-linearity and multiple hardening rules in order to implement the elastoplasticity deformation analysis. The analysis model is shown in Figure 9.1.4.

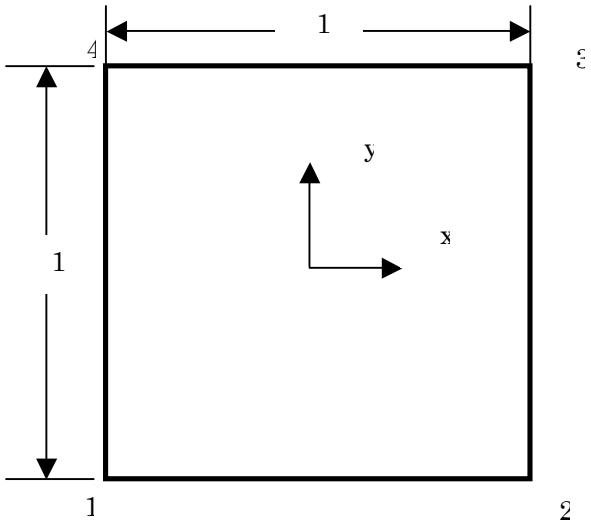


Figure 9.1.4: Elastoplasticity Deformation Analysis Model

(1) Verification conditions:

Material: Mises elastoplastic material

Young's Modulus	E	250	GPa
Poisson's Ratio	v	0.25	
Initial yield stress		5	MPa
Initial yield strain		0.25×10^{-4}	
Isotropic hardening factor H_i		0 or 62.5 GPa	

(2) Boundary conditions

Step 1: Forced displacement in nodes 2 and 3 $u_x = 0.2500031251 * 10^{-4}$

Step 2: Forced displacement in nodes 2 and 3 $u_x = 0.25000937518 * 10^{-4}$

Step 3: Forced displacement in nodes 3 and 4 $u_y = 0.2500031251 * 10^{-4}$

Step 4: Forced displacement in nodes 3 and 4 $u_y = 0.25000937518 * 10^{-4}$

Step 5: Forced displacement in nodes 2 and 3 $u_x = -0.25000937518 * 10^{-4}$

Step 6: Forced displacement in nodes 2 and 3 $u_x = -0.2500031251 * 10^{-4}$

Step 7: Forced displacement in nodes 3 and 4 $u_y = -0.25000937518 * 10^{-4}$

Step 8: Forced displacement in nodes 3 and 4 $u_y = -0.2500031251 * 10^{-4}$

All the nodes not shown here will be completely restrained.

The theoretical solution of this problem is as follows.

Strain (x10^{-4})			Equivalent Stress (MPa)	
ε_x	ε_y	ε_z	H_i =0 H_k =0	H_i =62.5 H_k =0
0.25	0	0	5.0	5.0
0.50	0	0	5.0	5.862
0.50	0.25	0	5.0	5.482
0.50	0.50	0	5.0	6.362
0.25	0.50	0	5.0	6.640
0	0.50	0	5.0	7.322
0	0.25	0	3.917	4.230
0	0	0	5.0	5.673

The calculation results for the above are as follows.

Strain （×10^{-4}）			Equivalent Stress (MPa)	
ε_x	ε_y	ε_z	H_i =0 H_k =0	H_i =62.5 H_k =0
0.25	0	0	5.0(0.0%)	5.0(0.0%)
0.50	0	0	5.0(0.0%)	5.862(0.0%)
0.50	0.25	0	5.0(0.0%)	5.482(0.0%)
0.50	0.50	0	5.0(0.0%)	6.362(-0.05%)
0.25	0.50	0	5.0(0.0%)	6.640(-0.21%)
0	0.50	0	5.0(0.0%)	7.322(-0.34%)
0	0.25	0	3.824(-2.4%)	4.230(-2.70%)
0	0	0	5.0(0.0%)	5.673(-2.50%)

(2-3) Contact analysis (1)

The National Agency for Finite Element Methods and Standards (U.K.): Contact Patch Test Problem CGS-4 was referred to in this verification problem, to test the limited sliding contact problem function with friction. The analysis model is shown in Figure 9.1.5.

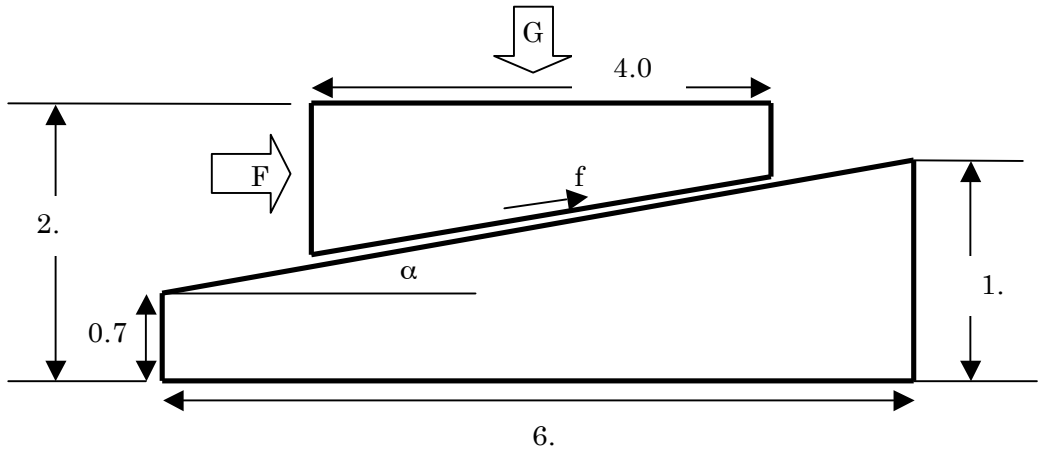


Figure 9.1.5: Contact Analysis Model

The equilibrium conditions of this problem are as follows.

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

The frictional force in the viscous friction stage was $f_c = E_t \Delta u$, and becomes $f_c = \mu(G\cos\alpha + F\sin\alpha \quad f_c = \mu(G\cos\alpha + F\sin\alpha)$ in the sliding friction stage.

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

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

(2-4) Contact analysis (2): Hertz contact problem

The Hertz contact problem of a cylinder of infinite length and an infinite plane surface was analyzed in this verification.

The radius of the cylinder was set to R = 8 mm, and Young's modulus E and Poisson's ratio μ of a deformable body was 1,100 Mpa and 0.0 respectively. Assuming that the contact area was sufficiently smaller than the radius of the cyclinder, a 1/4 cylindrical model was used to perform the analysis in consideration of the symmetry of the problem.

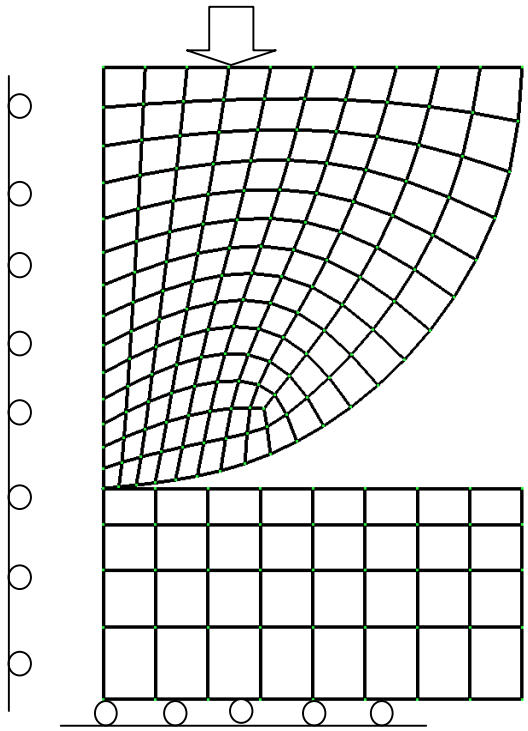


Figure 9.1.6: Hertz Contact Problem Analysis Model

(1) Verification results of contact radius

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

$a = \sqrt{\frac{4FR}{\pi E^*}}$ Herein, $E^* = E/2(1-\mu^2)$. In this calculation, the contact radius becomes a = 1.36 when pressure F = 100.

The equivalent nodal force of the point of contact is shown in Figure 9.1.7. This nodal force distribution is extrapolated to acquire the contact radius.

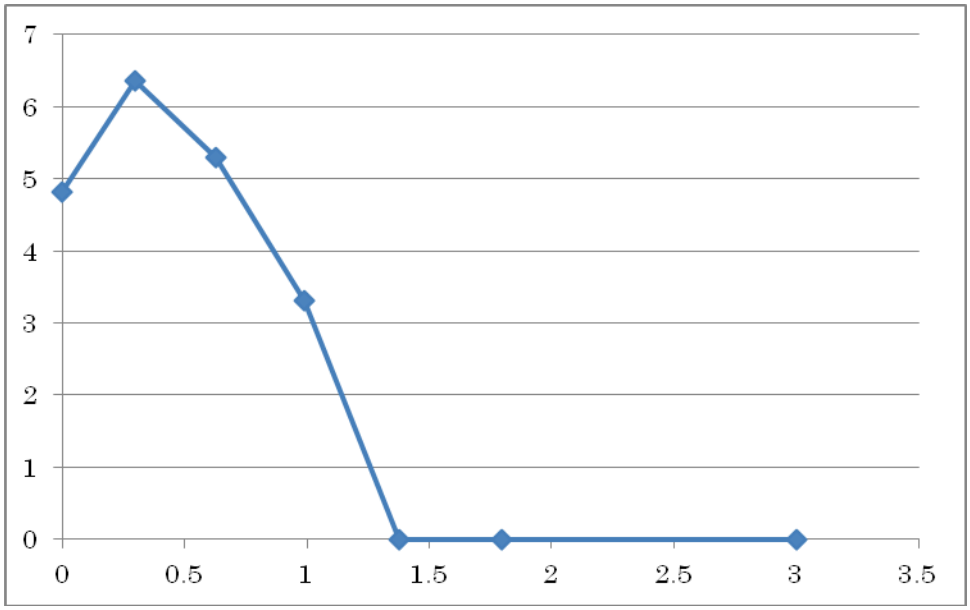


Figure 9.1.7: Equivalent Nodal Force Distribution of Contact Point

(2) Verification results of maximum shear stress

In this theoretical solution, the maximum shear stress is $\tau_{max} = 0.30\sqrt{\frac{FE^*}{\pi R}}$ in contact position z = 0.78a. In this calculation condition it becomes $\tau_{max} = 14.2\,\tau_{max} = 14.2$. In contrast to this, the calculation result of $\tau_{max} = 15.6$ was acquired.

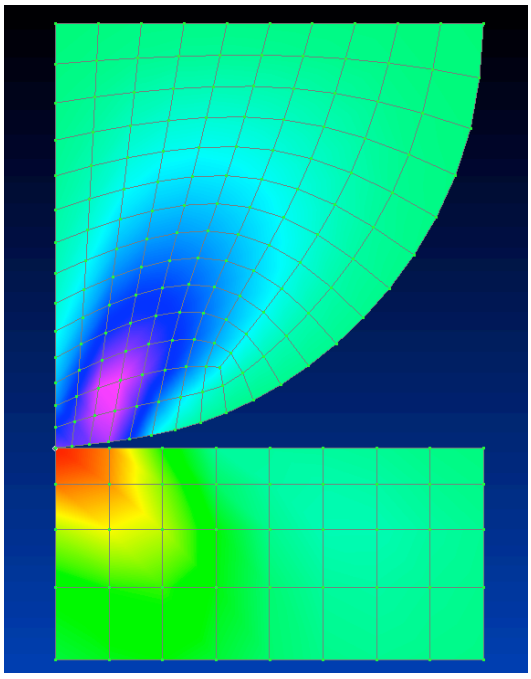


Figure 9.1.8: Shear Stress Distribution (Maximum Value = 15.6)

(3) Eigenvalue analysis

The verification model of verification case exJ ~ K is the same model as in verification case exA ~ G. The conceptual diagram of the verification model is shown in Figure 9.1.9. An eigenvalue analysis for this model was implemented. The eigenvalues to be acquired are the linear - cubic eigenvalues. In addition, the iterative method solver is used in exJ, and the direct method solver is used in exK. The verifications results are shown in Table 9.1.9 ~ Table 9.1.12.

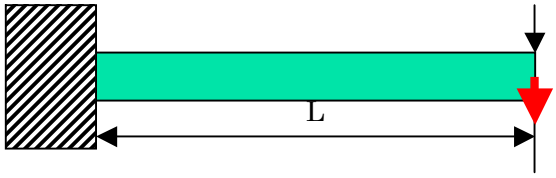


Figure 9.1.9: Verification Model

The vibration eigenvalue of the cantilever beam can be acquired by the following equations.

Linear : $$n_1 = \frac{(1.875)^2}{2\pi l^2}\sqrt{\frac{gEI}{\omega}} \tag{9.1}$$

Quadratic : $$n_2 = \frac{(4.694)^2}{2\pi l^2}\sqrt{\frac{gEI}{\omega}} \tag{9.2}$$

Cubic : $$n_3 = \frac{(7.855)^2}{2\pi l^2}\sqrt{\frac{gEI}{\omega}} \tag{9.3}$$

The characteristics of the verification model are as follows.

$$\begin{aligned} l &= 10.mm \\ E &= 4000.kgf/mm^2 \\ I &= 1./12.mm^4 \\ \omega &= 7.85x10^{-6}\,kgf/mm^3 \\ g &= 9800.mm/\sec^2 \end{aligned} \tag{9.4}$$

Therefore, up to a cubic eigenvalue becomes as follows.

n_1 = 3.609e3

n_2 = 2.262e4

$n_3 = 6.335e4$

Table 9.1.9: exJ: Verification Results with Iterative Method (Linear Eigenvalue)

Case Name	No. of Elements	Predicted Value: n1=3.609e3		Remarks
		NASTRAN	FrontISTR	
J231	40	5.861e3	5.861e3	33 nodes / plane stress problem
J232	40	3.596e3	3.593e3	105 nodes / plane stress problem
J241	20	3.586e3	4.245e3	33 nodes / plane stress problem
J242	20	3.590e3	3.587e3	85 nodes / plane stress problem
J341	240	5.442e3	5.429e3	99 nodes
J342	240	3.621e3	3.595e3	525 nodes
J351	80	3.695e3	4.298e3	99 nodes
J352	80	3.610e3	3.609e3	381 nodes
J361	40	3.679e3	3.619e3	99 nodes
J362	40	3.611e3	3.606e3	330 nodes

Table 9.1.10: exJ: Verification Results with Iterative Method (Quadratic Eigenvalue)

Case Name	No. of Elements	Predicted Value: n1=2.262e4		Remarks
		NASTRAN	FrontISTR	
J231	40	3.350e4	3.351e4	33 nodes / plane stress problem
J232	40	2.163e4	2.156e4	105 nodes / plane stress problem
J241	20	2.149e4	2.516e4	33 nodes / plane stress problem
J242	20	2.149e4	2.143e4	85 nodes / plane stress problem
J341	240	3.145e4	3.138e4	99 nodes
J342	240	2.171e4	2.155e4	525 nodes
J351	80	2.208e4	2.546e4	99 nodes
J352	80	2.156e4	2.149e4	381 nodes
J361	40	2.202e4	2.168e4	99 nodes

J362	40	2.154e4	2.144e4	330 nodes

Note: Since the linear and quadratic eigenvalues of the 3D model have multiple roots, the value based on the cubic equation is used to describe the quadratic value in the table.

Table 9.1.11: exK: Verification Results with Direct Method (Linear Eigenvalue)

Case Name	No. of Elements	Predicted Value: n1=3.609e3		Remarks
		NASTRAN	FrontISTR	
J231	40	5.861e3	5.861e3	33 nodes / plane stress problem
J232	40	3.596e3	3.593e3	105 nodes / plane stress problem
J241	20	3.586e3	4.245e3	33 nodes / plane stress problem
J242	20	3.590e3	3.587e3	85 nodes / plane stress problem
J341	240	5.442e3	5.429e3	99 nodes
J342	240	3.621e3	3.595e3	525 nodes
J351	80	3.695e3	4.298e3	99 nodes
J352	80	3.610e3	3.609e3	381 nodes
J361	40	3.679e3	3.619e3	99 nodes
J362	40	3.611e3	3.606e3	330 nodes
J731	40	-	3.606e3	33 nodes
J741	20	-	3.594e3	33 nodes

Table 9.1.12: exK: Verification Results with Direct Method (Quadratic Eigenvalue)

Case Name	No. of Elements	Predicted Value: n1=2.262e4		Remarks
		NASTRAN	FrontISTR	
J231	40	3.350e4	3.351e4	33 nodes / plane stress problem
J232	40	2.163e4	2.156e4	105 nodes / plane stress problem
J241	20	2.149e4	2.516e4	33 nodes / plane stress problem
J242	20	2.149e4	2.143e4	85 nodes / plane stress problem

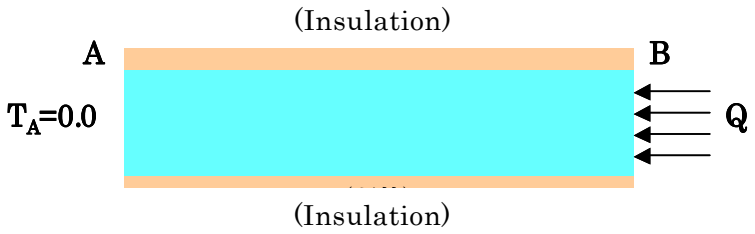
J341	240	3.145e4	3.138e4	99 nodes
J342	240	2.171e4	2.155e4	525 nodes
J351	80	2.208e4	2.546e4	99 nodes
J352	80	2.156e4	2.149e4	381 nodes
J361	40	2.202e4	2.168e4	99 nodes
J362	40	2.154e4	2.144e4	330 nodes
J731	40	-	2.156e4	33 nodes
J741	20	-	2.153e4	33 nodes

Note: Since the linear and quadratic eigenvalues of the 3D model have multiple roots, the value based on the cubic equation is used to describe the quadratic value in the table.

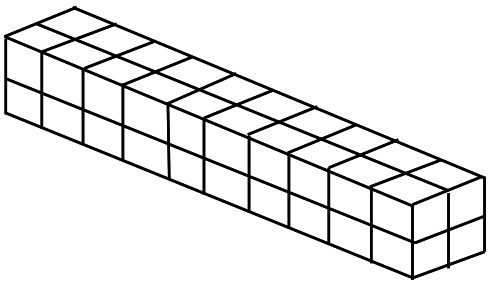
(4) Heat conduction analysis

Conditions common to a steady heat conduction analysis are shown in Figure 9.1.10. The individual conditions of verification case exM - exT are shown in Figure 9.1.11. Mesh partitioning equivalent to that of exA was used.

Temperature distribution tables for each case of the verification results are shown in Table 9.1.13 ~ Table 9.1.20.



(1) Model Conceptual Diagram



(2) Mesh Partition

Length between AB: L = 10.0m

Cross-sectional area: A = 1.0 mm^2

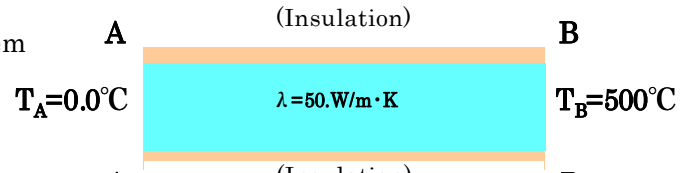
Temperature dependency of thermal conductivity

Thermal conductivity λ(W/mK)	Temperature (°C)
50.0	0.
35.0	500.
20.0	1000.

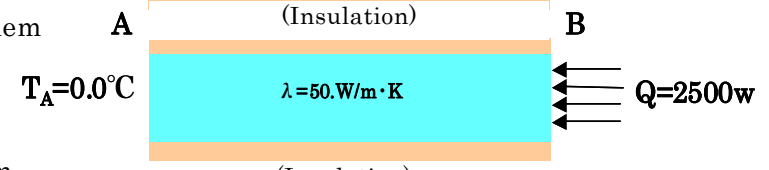
Figure 9.1.10: Verification Conditions of Steady Heat Conduction Analysis

exM: Linear material

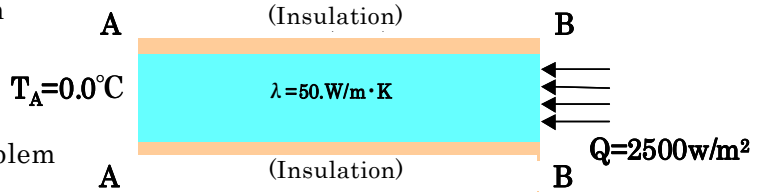
exN: Specified temperature problem



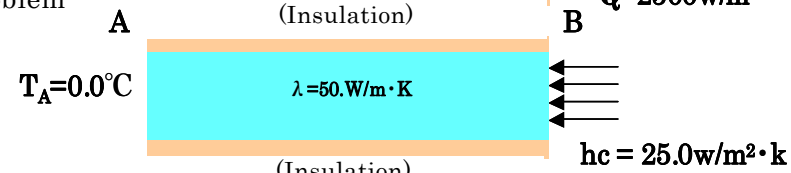
exO: Concentrated heat flux problem



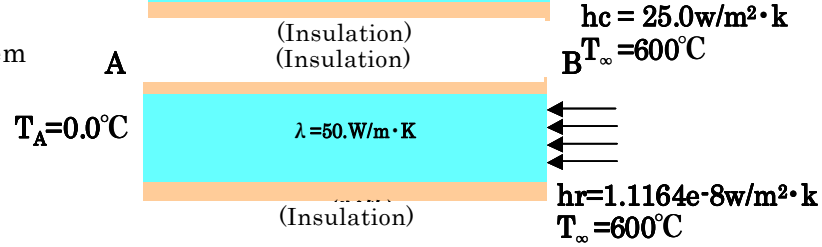
exP: Distributed heat flux problem



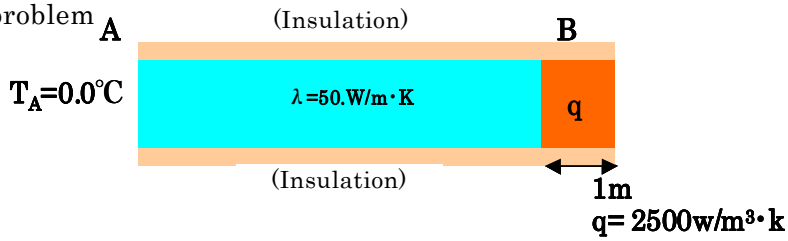
exQ: Convective heat transfer problem



exR: Radiant heat transfer problem



exS: Volumetric heat generation problem



exT: Internal gap problem

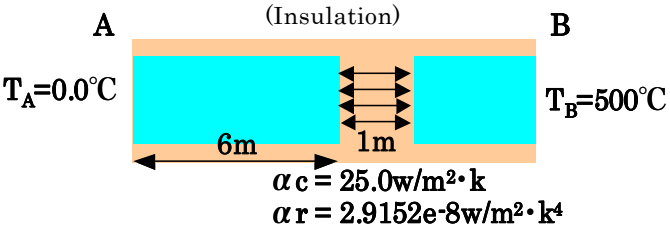


Figure 9.1.11: Analysis Conditions for each Verification Case

Table 9.1.13: exM: Verification Results of Steady Calculation by Linear Material

Case Name	Element Type	No. of Elements / Nodes	Distance from End A (m)					
			End A	2.0	4.0	6.0	8.0	End B
M361A	361	40／33	0.0	100.	200.	300.	400.	500.
M361B	361	40／105	0.0	100.	200.	300.	400.	500.
M361C	361	20／33	0.0	100.	200.	300.	400.	500.
M361D	361	20／85	0.0	100.	200.	300.	400.	500.
M361E	361	240／99	0.0	100.	200.	300.	400.	500.
M361F	361	24／525	0.0	100.	200.	300.	400.	500.
M361G	361	80／99	0.0	100.	200.	300.	400.	500.

Table 9.1.14: exN: Verification Results of Specified Temperature Problem

Case Name	Element Type	No. of Elements / Nodes	Distance from End A (m)					
			End A	2.0	4.0	6.0	8.0	End B
ABAQUS	361	40／99	0.0	87.3	179.7	278.2	384.3	500.
N231	231	40／33	0.0	87.2	179.5	278.0	384.1	500.
N232	232	40／105	0.0	86.0	178.3	276.8	382.9	500.
N241	241	20／33	0.0	87.3	179.7	278.2	384.3	500.
N242	242	20／85	0.0	87.3	179.7	278.2	384.3	500.
N341	341	240／99	0.0	87.3	179.7	278.2	384.3	500.
N342	342	24／525	0.0	87.9	179.9	278.0	383.6	500.
N351	351	80／99	0.0	87.3	179.7	278.2	384.3	500.
N352	352	80／381	0.0	87.3	179.7	278.2	384.3	500.
N361	361	40／99	0.0	87.3	179.7	278.2	384.3	500.
N362	362	40／330	0.0	87.3	179.7	278.2	384.3	500.
N731	731	40／33	0.0	87.3	179.7	278.2	384.3	500.
N741	741	20／33	0.0	87.3	179.7	278.2	384.3	500.

Table 9.1.15: exO: Verification Results of Concentrated Heat Flux Problem

Case Name	Element Type	No. of Elements / Nodes	Distance from End A (m)					
			End A	2.0	4.0	6.0	8.0	End B
ABAQUS	361	40／99	0.0	103.2	213.7	333.3	464.8	612.6
O231	231	40／33	0.0	103.2	213.7	333.3	464.8	612.6
O232	232	40／105	0.0	103.2	213.7	333.3	464.8	612.6
O241	241	20／33	0.0	103.2	213.7	333.3	464.8	612.6
O242	242	20／85	0.0	103.2	213.7	333.4	465.2	618.0
O341	341	240／99	-	-	-	-	-	-
O342	342	24／525	0.0	104.4	214.9	334.7	466.3	614.6
O351	351	80／99	-	-	-	-	-	-
O352	352	80／381	0.0	103.2	213.7	333.3	465.0	624.2
O361	361	40／99	0.0	103.2	213.7	333.3	464.8	612.6
O362	362	40／330	0.0	103.2	213.7	333.4	465.5	623.5
O731	731	40／33	0.0	103.2	213.7	333.3	464.8	612.5
O741	741	20／33	0.0	103.2	213.7	333.3	464.8	612.6

Table 9.1.16: exP: Verification Results of Distributed Heat Flux Problem

Case Name	Element Type	No. of Elements / Nodes	Distance from End A (m)					
			End A	2.0	4.0	6.0	8.0	End B
ABAQUS	361	40／99	0.0	103.2	213.7	333.3	464.8	612.6
P231	231	40／33	0.0	103.2	213.7	333.3	464.8	612.6
P232	232	40／105	0.0	103.2	213.7	333.3	464.8	612.6
P241	241	20／33	0.0	103.2	213.7	333.3	464.8	612.6
P242	242	20／85	0.0	103.2	213.7	333.3	464.8	612.6
P341	341	240／99	-	-	-	-	-	-
P342	342	24／525	0.0	103.2	213.7	333.3	464.8	612.6
P351	351	80／99	-	-	-	-	-	-
P352	352	80／381	0.0	103.2	213.7	333.3	464.8	612.6
P361	361	40／99	0.0	103.2	213.7	333.3	464.8	612.6
P362	362	40／330	0.0	103.2	213.7	333.4	465.5	612.6
P731	731	40／33	0.0	103.2	213.7	333.3	464.8	612.5
P741	741	20／33	0.0	103.2	213.7	333.3	464.8	612.6

Table 9.1.17: exQ: Verification Results of Convective Heat Transfer Problem

Case Name	Element Type	No. of Elements / Nodes	Distance from End A (m)					
			End A	2.0	4.0	6.0	8.0	End B
ABAQUS	361	40／99	0.0	89.2	183.8	284.8	393.9	513.2
Q231	231	40／33	0.0	89.2	183.8	284.8	393.9	513.2
Q232	232	40／105	0.0	89.2	183.8	284.8	393.9	513.2
Q241	241	20／33	0.0	89.2	183.8	284.8	393.9	513.2
Q242	242	20／85	0.0	89.2	183.8	284.8	393.9	513.2
Q341	341	240／99	-	-	-	-	-	-
Q342	342	240／525	0.0	89.2	183.8	284.8	393.9	513.2
Q351	351	80／99	-	-	-	-	-	-
Q352	352	80／381	0.0	89.2	183.8	284.8	393.9	513.2
Q361	361	40／99	0.0	89.2	183.8	284.8	393.9	513.2
Q362	362	40／330	0.0	89.2	183.8	284.8	393.9	513.2
Q731	731	40／33	0.0	89.2	183.8	284.8	393.9	513.2
Q741	741	20／33	0.0	89.2	183.8	284.8	393.9	513.2

Table 9.1.18: exR: Verification Results of Radiant Heat Transfer Problem

Case Name	Element Type	No. of Elements / Nodes	Distance from End A (m)					
			End A	2.0	4.0	6.0	8.0	End B
ABAQUS	361	40／99	0.0	89.5	184.4	285.8	395.3	515.2
R231	231	40／33	0.0	89.5	184.4	285.8	395.3	515.2
R232	232	40／105	0.0	89.5	184.4	285.8	395.3	515.2
R241	241	20／33	0.0	89.5	184.4	285.8	395.3	515.2
R242	242	20／85	0.0	89.5	184.4	285.8	395.3	515.2
R341	341	240／99	-	-	-	-	-	-
R342	342	240／525	0.0	89.5	184.4	285.8	395.3	515.2
R351	351	80／99	-	-	-	-	-	-
R352	352	80／381	0.0	89.5	184.4	285.8	395.3	515.2
R361	361	40／99	0.0	89.5	184.4	285.8	395.3	515.2
R362	362	40／330	0.0	89.5	184.4	285.8	395.3	515.2
R731	731	40／33	0.0	89.5	184.4	285.8	395.3	515.2
R741	741	20／33	0.0	89.5	184.4	285.8	395.3	515.2

Table 9.1.19: exS: Verification Results of Volumetric Heat Generation Problem

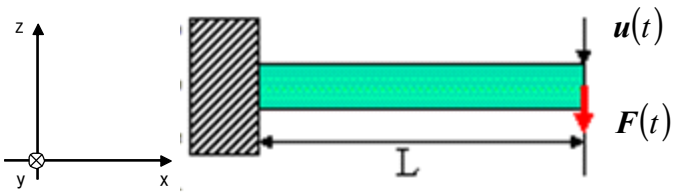
Case Name	Element Type	No. of Elements / Nodes	Distance from End A (m)					
			End A	2.0	4.0	6.0	8.0	End B
ABAQUS	361	40／99	0.0	103.2	213.7	333.3	464.8	612.6
S231	231	40／33	0.0	103.2	213.7	333.3	464.8	612.6
S232	232	40／105	0.0	103.2	213.7	333.3	464.8	612.6
S241	241	20／33	0.0	103.2	213.7	333.3	464.8	612.6
S242	242	20／85	0.0	103.2	213.7	333.3	464.8	612.6
S341	341	240／99	-	-	-	-	-	-
S342	342	24／525	0.0	103.2	213.7	333.3	464.8	612.6
S351	351	80／99	-	-	-	-	-	-
S352	352	80／381	0.0	103.2	213.7	333.3	464.8	612.6
S361	361	40／99	0.0	103.2	213.7	333.3	464.8	612.6
S362	362	40／330	0.0	103.2	213.7	333.3	464.8	612.6
S731	731	40／33	0.0	103.2	213.7	333.3	464.8	612.6
S741	741	20／33	0.0	103.2	213.7	333.3	464.8	612.6

Table 9.1.20: exT: Verification Results of Internal Gap Problem

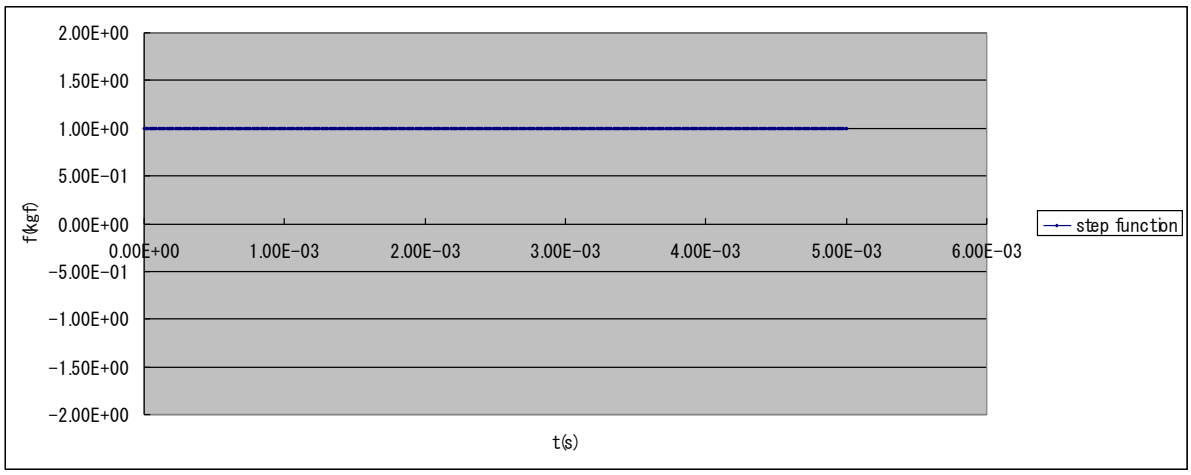
Case Name	Element Type	No. of Elements / Nodes	Distance from End A (m)					
			End A	2.0	4.0	6.0	8.0	End B
ABAQUS	361	40／99	0.0	88.6	182.4	282.6	387.7	500.0
T231	231	40／33	0.0	88.6	182.4	282.6	387.7	500.0
T232	232	40／105	0.0	88.6	182.4	282.6	387.7	500.0
T241	241	20／33	0.0	88.6	182.4	282.6	387.7	500.0
T242	242	20／85	0.0	88.6	182.4	282.6	387.7	500.0
T341	341	240／99	-	-	-	-	-	-
T342	342	24／525	0.0	88.6	182.4	282.6	387.7	500.0
T351	351	80／99	-	-	-	-	-	-
T352	352	80／381	0.0	88.6	182.4	282.6	387.7	500.0
T361	361	40／99	0.0	88.6	182.4	282.6	387.7	500.0
T362	362	40／330	0.0	88.6	182.4	282.6	387.7	500.0
T731	731	40／33	0.0	88.6	182.4	282.6	387.7	500.0
T741	741	20／33	0.0	88.6	182.4	282.6	387.7	500.0

(5) Linear dynamic analysis

In exW, the same cantilever beam as in item (1) was used as the subject to perform the linear dynamic analysis. The verification conditions are shown in Figure 9.1.12. In this verification, the affects in which the time increments may have on the results for the same mesh partition was verified. As a dynamic analysis method, both the implicit method and the explicit method were used, and element types 361 and 342 were used. The verification results are shown in Table 9.1.22 and Figure 9.1.13 ~ Figure 9.1.15.



Analysis Model



Time History of External Force F

Theoretical solution of vibration point displacement:

$$\boldsymbol{F}(t) = \boldsymbol{F}_0 I(t)$$

Where,

$\boldsymbol{F}_0$: Constant vector

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

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

Figure 9.1.12: Verification Conditions of Linear Dynamic Analysis

Verification conditions:

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

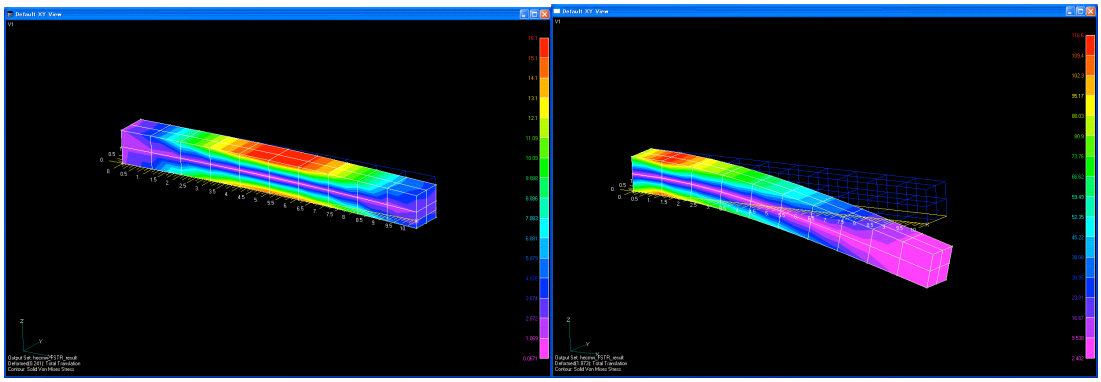
Element Hexahedral linear element
Tetrahedral quadratic element
Solution Implicit method
Parameterγof Newmark-βmethod 1/2
Parameterβof Newmark-βmethod 1/4
Explicit method
Damping N/A

Table 9.1.21: Verification Conditions of Linear Dynamic Analysis (Continued)

Case Name	Element Type	No. of Nodes	No. of Elements	Solution	Time Increment Δt(s)
W361_c0_im_m2_t1	361	99	40	Implicit method	1.0E-06
W361_c0_im_m2_t2	361	99	40	Implicit method	1.0E-05
W361_c0_im_m2_t3	361	99	40	Implicit method	1.0E-04
W361_c0_ex_m2_t1	361	99	40	Explicit method	1.0E-08
W361_c0_ex_m2_t2	361	99	40	Explicit method	1.0E-07
W361_c0_ex_m2_t3	361	99	40	Explicit method	1.0E-06
W342_c0_im_m2_t1	342	525	240	Implicit method	1.0E-06
W342_c0_im_m2_t2	342	525	240	Implicit method	1.0E-05
W342_c0_im_m2_t3	342	525	240	Implicit method	1.0E-04
W342_c0_ex_m2_t1	342	525	240	Explicit method	1.0E-08
W342_c0_ex_m2_t2	342	525	240	Explicit method	5.0E-08
W342_c0_ex_m2_t3	342	525	240	Explicit method	1.0E-07

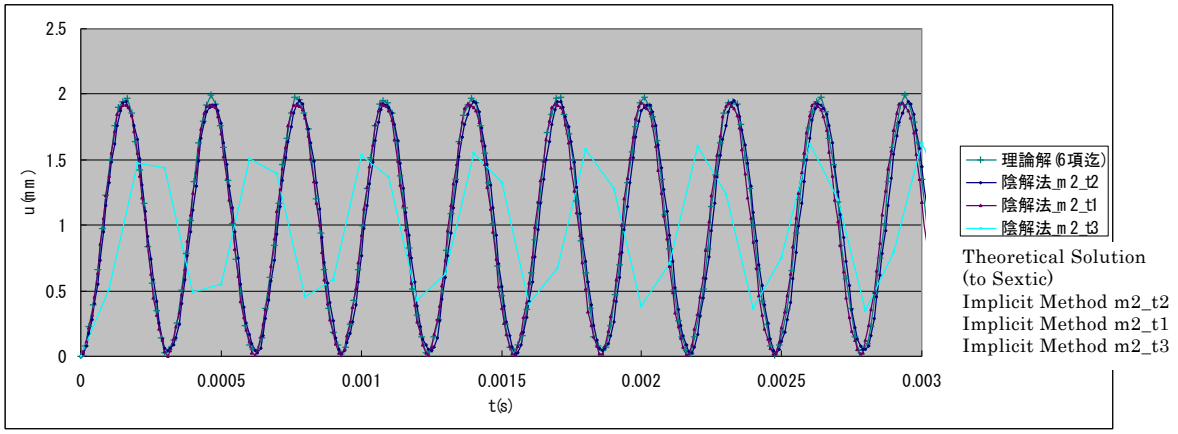
Table 9.1.22: exW: Verification Results of Linear Dynamic Analysis for Cantilever Beam

Case Name	Element Type	No. of Nodes	No. of Elements	Solution	z Direction Displacement: uz (mm) when Time t = 0.002 (s)	
					Theoretical Solution Repeated to Sextic Equation	FrontISTR
W361_c0_im_m2_t1	361	99	40	Implicit method	1.9753	1.9302
W361_c0_im_m2_t2	361	99	40	Implicit method	1.9753	1.8686
W361_c0_im_m2_t3	361	99	40	Implicit method	1.9753	0.3794
W361_c0_ex_m2_t1	361	99	40	Explicit method	1.9753	1.9302
W361_c0_ex_m2_t2	361	99	40	Explicit method	1.9753	1.9247
W361_c0_ex_m2_t3	361	99	40	Explicit method	1.9753	Divergence
W342_c0_im_m2_t1	342	525	240	Implicit method	1.9753	1.9431
W342_c0_im_m2_t2	342	525	240	Implicit method	1.9753	1.8719
W342_c0_im_m2_t3	342	525	240	Implicit method	1.9753	0.3873
W342_c0_ex_m2_t1	342	525	240	Explicit method	1.9753	1.9359
W342_c0_ex_m2_t2	342	525	240	Explicit method	1.9753	1.9358
W342_c0_ex_m2_t3	342	525	240	Explicit method	1.9753	Divergence

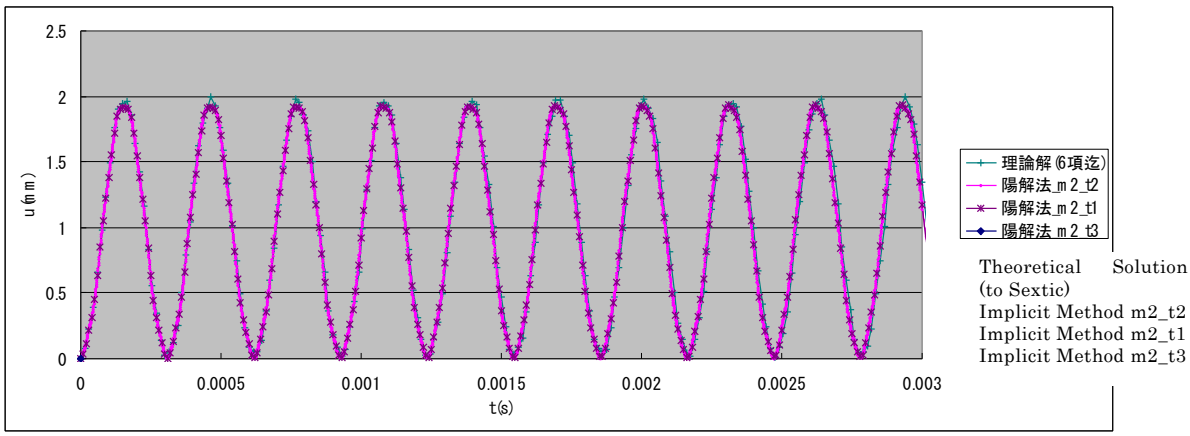


(a) t=2.0E-03(s)　　(b) t=4.0E-03(s)

Figure 9.1.13: Deformed Figure and Equivalent Stress Distribution of Cantilever Beam (W361_c0_im_m2_t2)

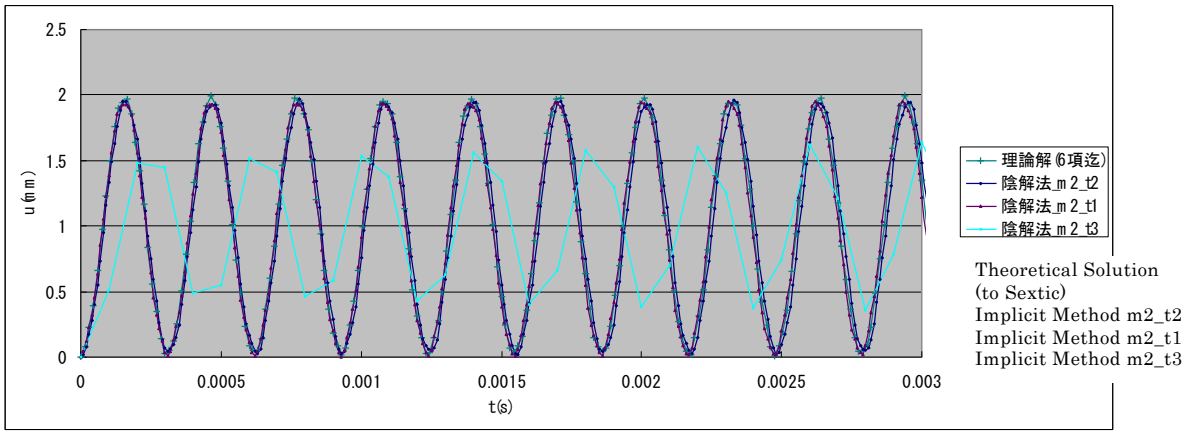


(a)　Element Type 361: Implicit method

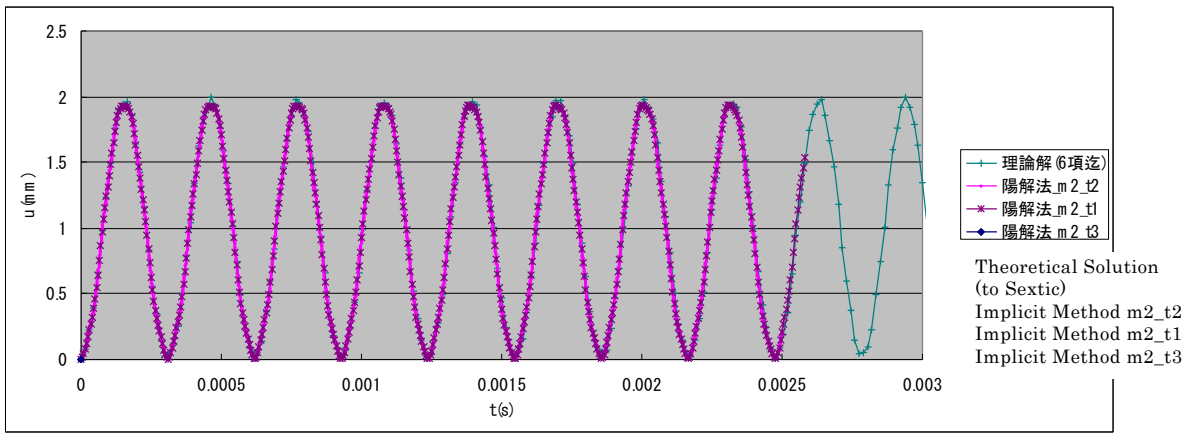


(b)　Element Type 361: Explicit method

Figure 9.1.14: Time History of Vibration Point Displacement u_z



(a) Element Type 342: Implicit method

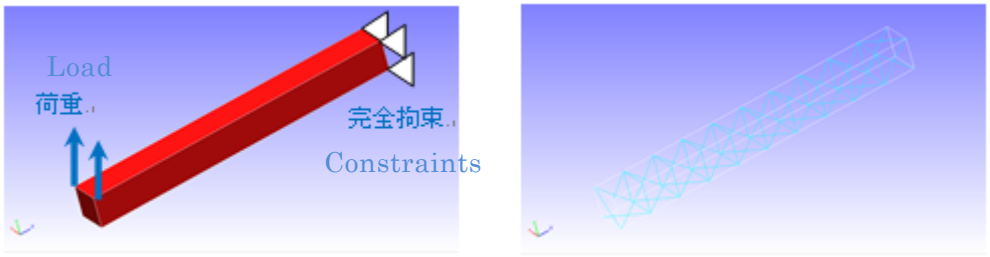


(b) Element Type 342: Explicit method

Figure 9.1.15: Time History of Vibration Point Displacement u_z

(6) Frequency Response Analysis

A cantilever beam was used for the verification and validation of frequency response analayis.The results were compared with those of ABAQUS. The analysis model and boundary conditions are shown below.



Analysis model (Element type= 341, Number of Elements:126, Number of Nodes: 55)

Analysis conditions :

Young's modulus	E	210000 N/mm^2
Poisson's ratio	ν	0.3
Density	ρ	7.89E-09t/mm^3
Gravity	g	9800.0 mm/s^2
Applied force	F_0	1.0 N
Rayleigh damping parameter Rm		0.0
Rayleigh damping parameter Rk		7.2E-07

Eigenvalue upto 5th mode obtained by eigenvalue analysis are listed below.

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

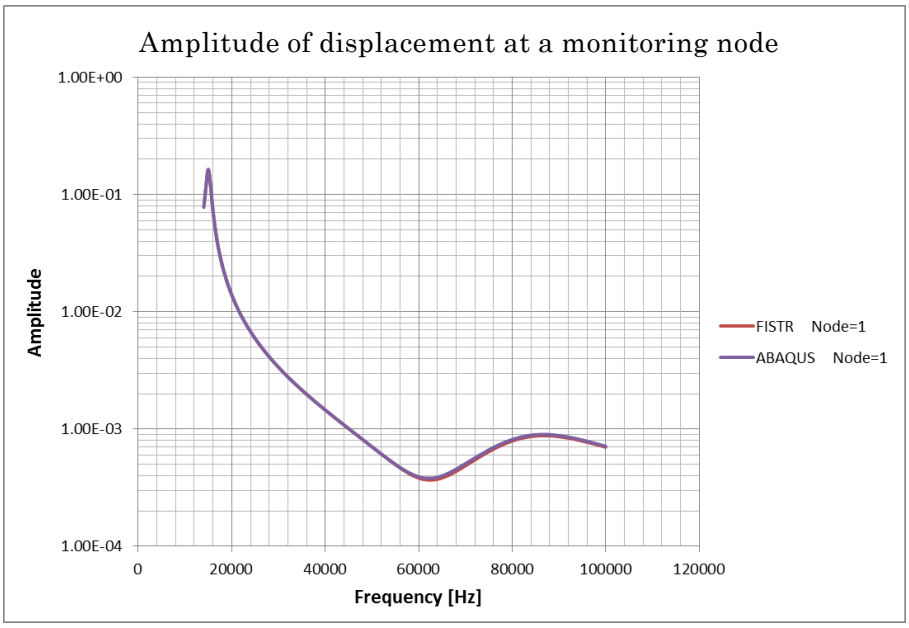


Fig. 9.1.16 : Frequency dependency of the displacement amplitude at loaded point

9.2 Example of Actual Model for Elastic Static Analysis

9.2.1 Analysis Model

A list of an actual model verification example for the elastic static analysis is shown in Table 9.2.1. The shape of the model (portions excluded) is shown in Figure 9.2.1 ~ Figure 9.2.5. In order to execute an example of element type 731 and 741, a separate direct method solver is required.

Table 9.2.1: Example of Actual Model Verification for Elastic Static Analysis

Case Name	Element Type	Verification Model	No. of Nodes	No. of Degrees of Freedom
EX01A	342	Connecting rod (100,000 nodes)	94,074	282,222
EX01B	342	Connecting rod (330,000 nodes)	331,142	993,426
EX02	361	Block with hole	37,386	112,158
EX03	342	Turbine blade	10,095	30,285
EX04	741	Cylindrical shell	10,100	60,600
EX05A	731	Wine glass (coarse)	7,240	43,440
EX05B	731	Wine glass (medium)	48,803	292,818
EX05C	731	Wine glass (fine)	100,602	603,612

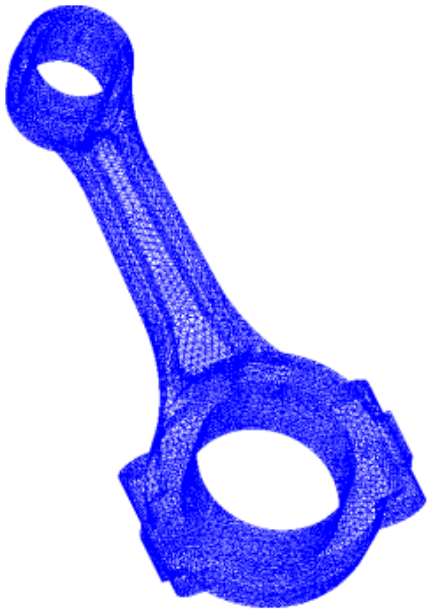


Figure 9.2.1: Connecting Rod (EX01A)

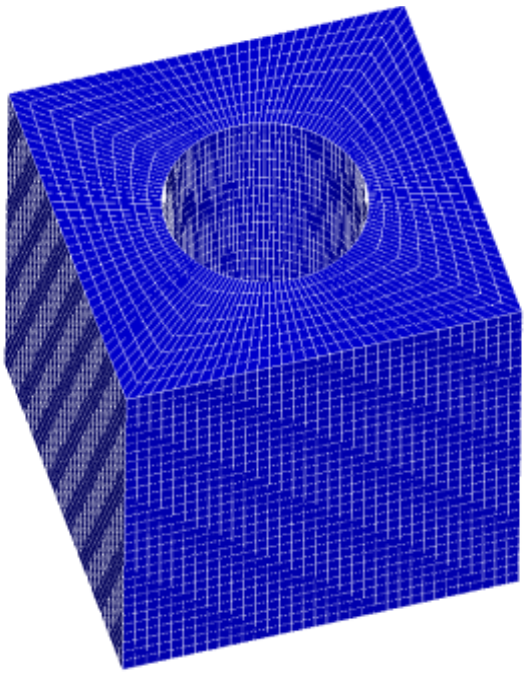


Figure 9.2.2: Block with Hole (EX02)

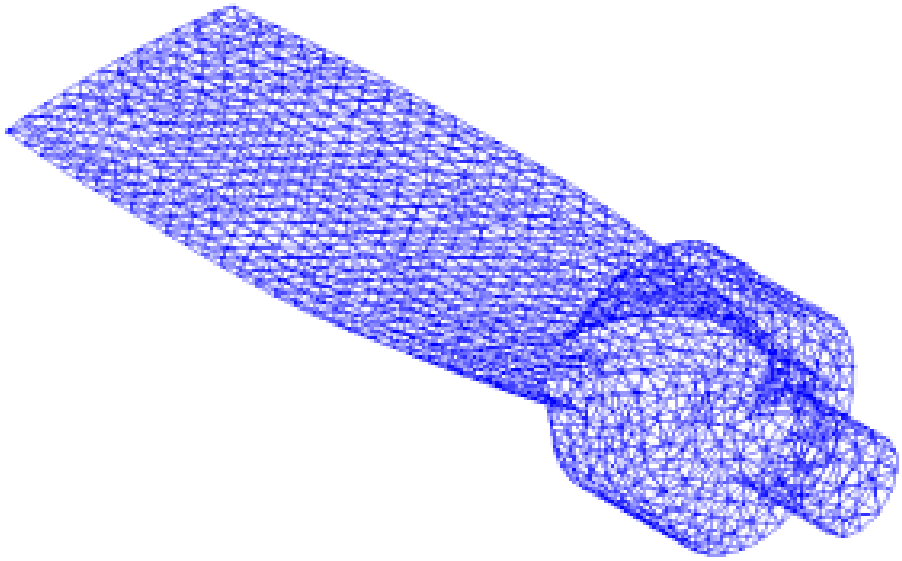


Figure 9.2.3: Turbine Blade (EX03, EX06)

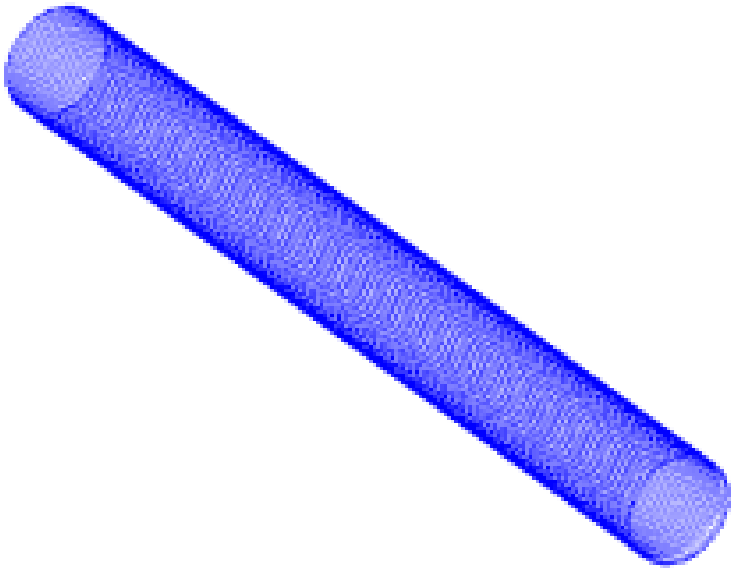


Figure 9.2.4: Cylindrical Shell (EX04, EX09)

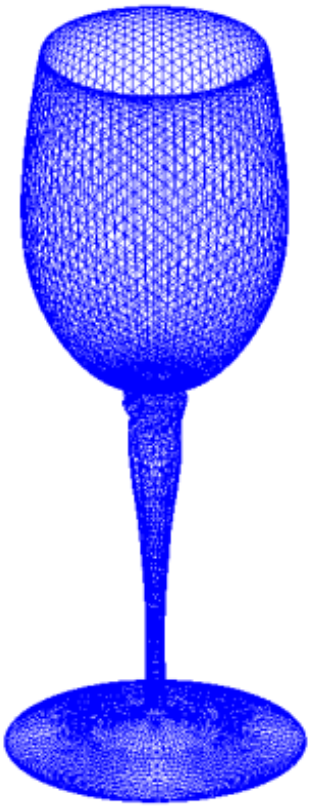


Figure 9.2.5: Wine Glass (EX05, EX10A)

9.2.2 Analysis Results

9.2.2.1 Example of Analysis Results

An example of the analysis results is shown in Figure 9.2.6 ~ Figure 9.2.9.

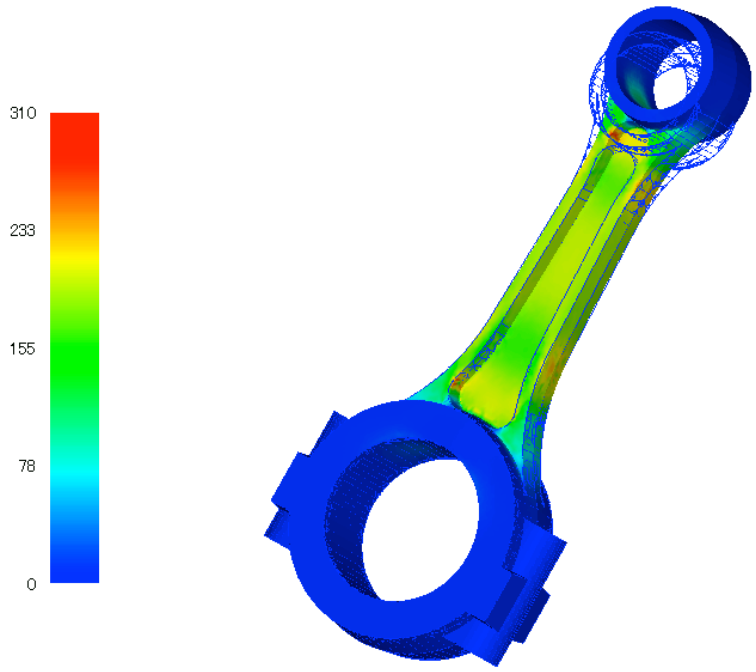


Figure 9.2.6: EX01A Analysis Results (Mises Stress and Deformed Figure (10 times))

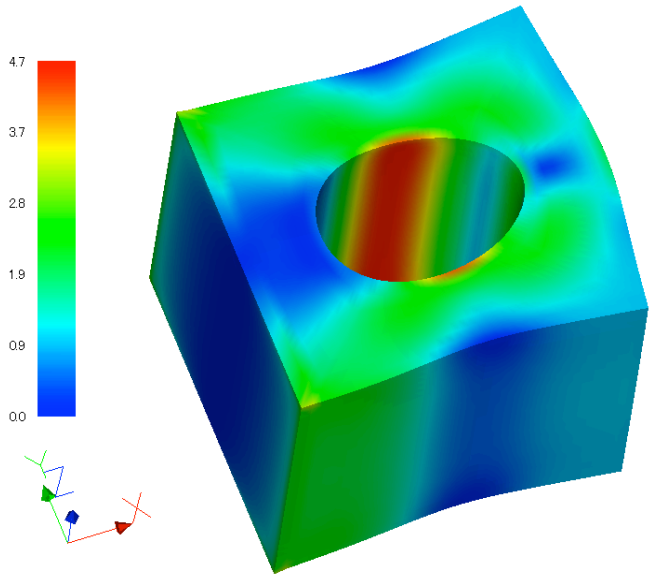


Figure 9.2.7: EX02 Analysis Results (Mises Stress and Deformed Figure (100 times))

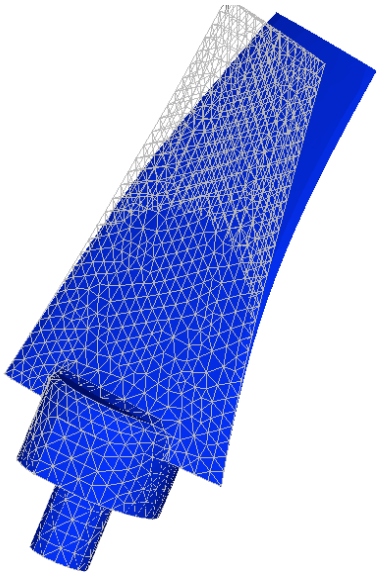


Figure 9.2.8: EX03 Analysis Results (Deformed Figure (10 times))

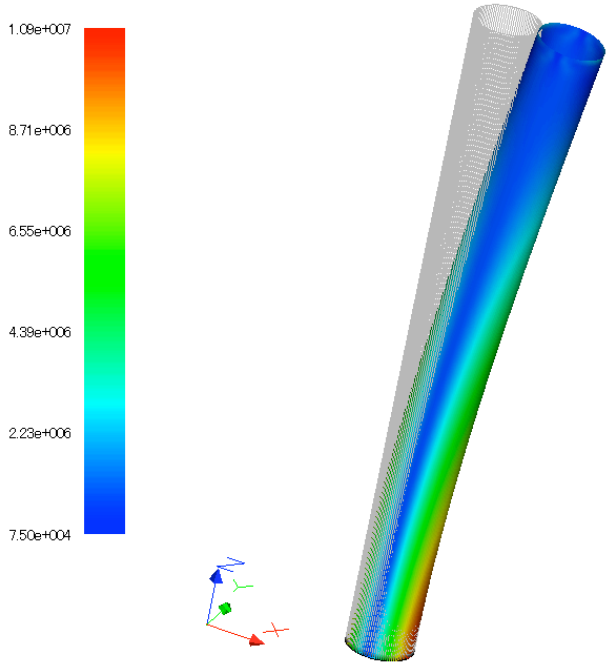


Figure 9.2.9: EX04 Analysis Results (Deformed Figure (100 times))

9.2.2.2 Verification Results of Analysis Performance by Verification Example EX02

An equivalent block model with a hole as in verification example EX02 was used to perform the analysis by general purpose commercial software ABAQUS. The maximum and minimum comparison results of FrontISTR and the stress component are shown in Figure 9.2.10. This figure shows that the stress component is extremely coherent.

Next, the results of investigating the affects of domain partitioning in the stress distribution are shown. The domain partitioning was performed according to the RCB method, and X, Y and Z were partitioned into two in each axial direction, and the overall model was partitioned into 8 domains. The partitioned state is shown in Figure 9.2.11. The stress distribution in the analysis results of a single domain model and a model partitioned into eight domains is shown in Figure 9.2.12.

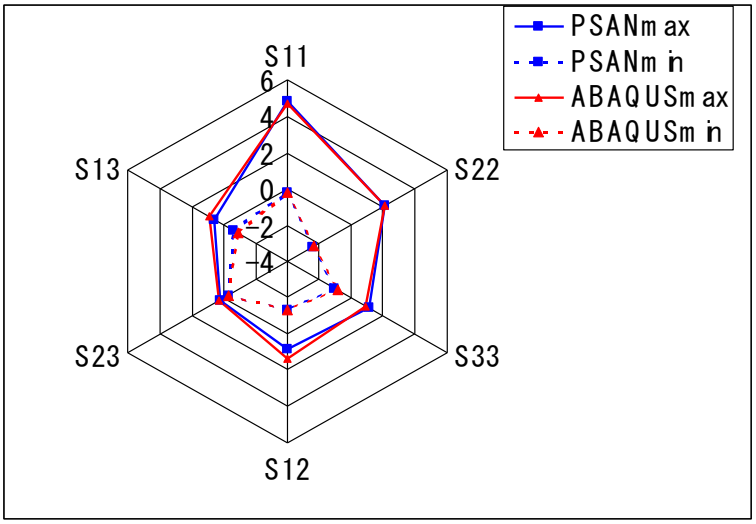


Figure 9.2.10: Comparison of Stress Component with General Purpose Software in EX02

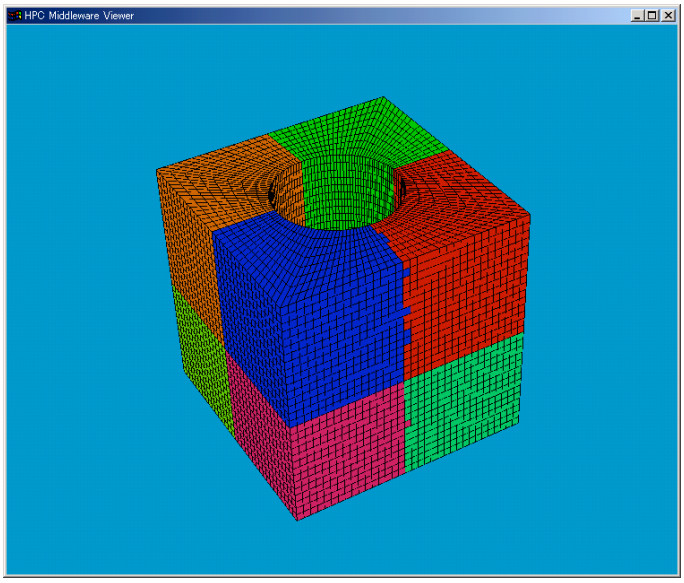
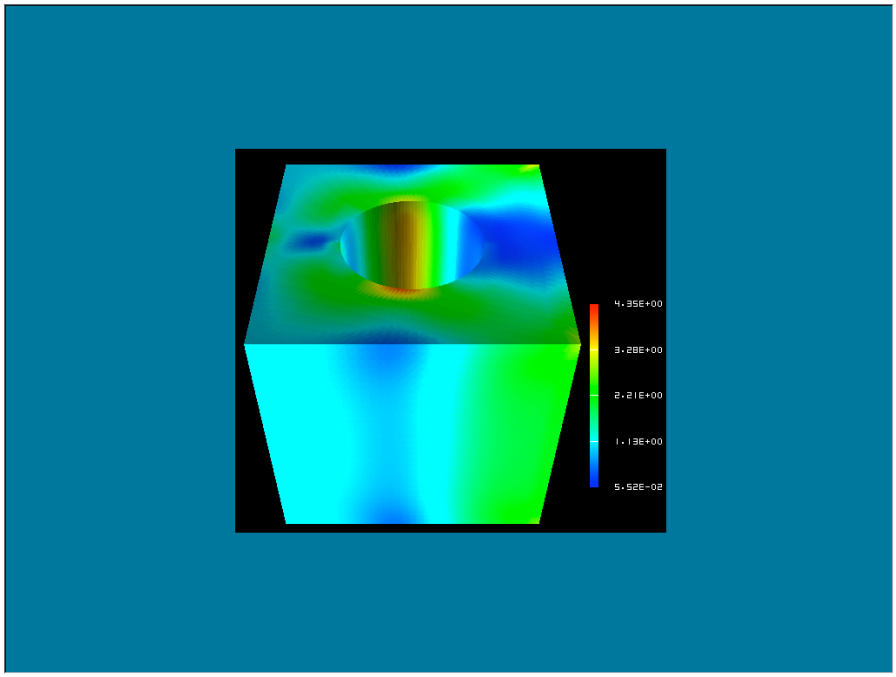
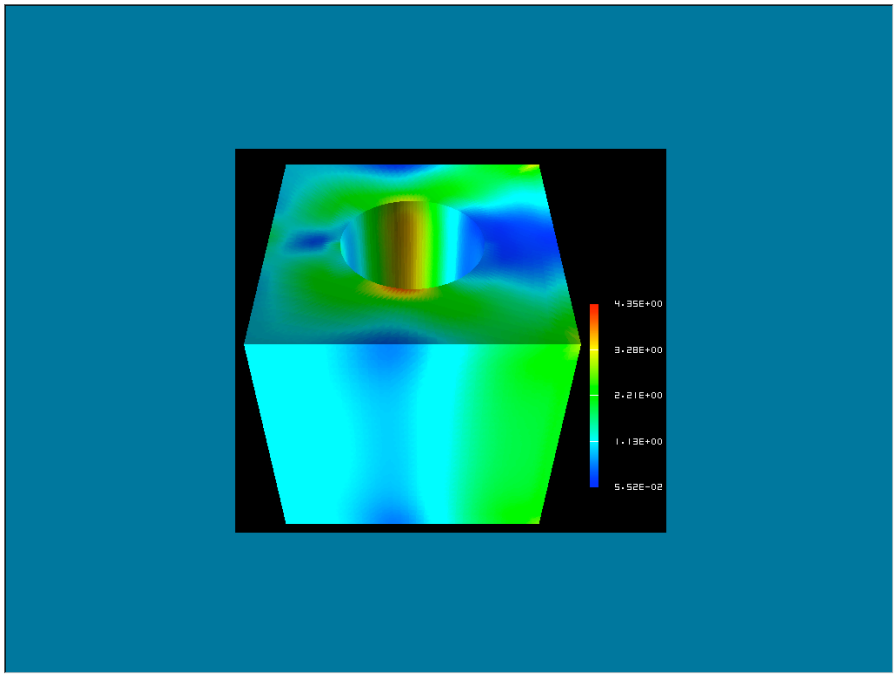


Figure 9.2.11: Eight Domain Partitioned Results of EX02 by RCB Method



(a) Single Domain Model



(b) Eight Domain Partitioned Model

Figure 9.2.12: Difference of Mises Stress Distribution by Domain Partitioning

Figure 9.2.12 shows no difference between both models, and it is clear that both models are in complete agreement.

Next, the comparison results of the execution time by setting the HEC-MW solver is shown in Table 9.2.2. The convergence history to the solution is shown in Figure 9.2.13.

Table 9.2.2: Comparison of Execution Time by HEC-MW Solver

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

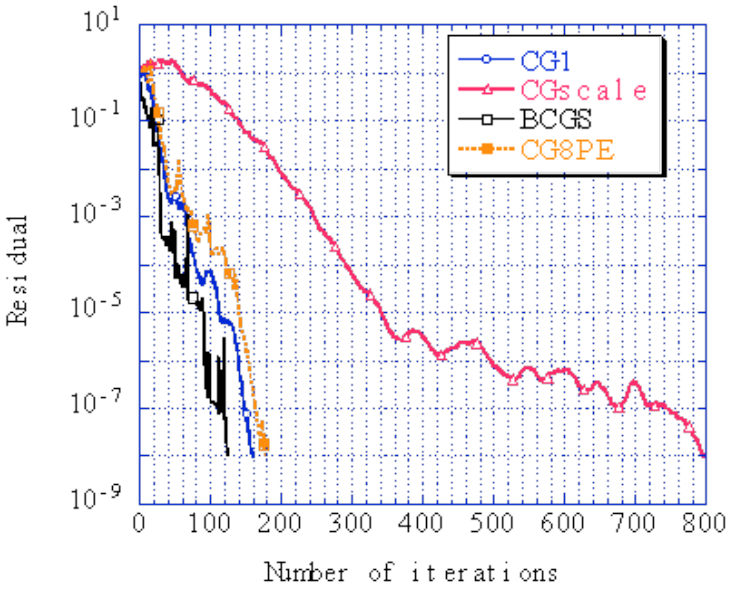


Figure 9.2.13: Comparison of Convergence History by HEC-MW Solver

(Convergence judgment threshold: 1.0×10^{-8})

9.2.2.3 Comparison of Computing Time by Verification Example EX01A

Verification example EX01A (connecting rod) was used to verify the acceleration rate of the calculating speed by domain partitioning. A Xeon 2.8 GHz 24 node cluster computing system was used for the calculations. The results are shown in Figure 9.2.14. In this figure, it is clear that the calculating speed accelerated in proportion to the number of domains.

The difference in the computing time by the environment of the computing system was also examined. The results are shown in Table 9.2.3.

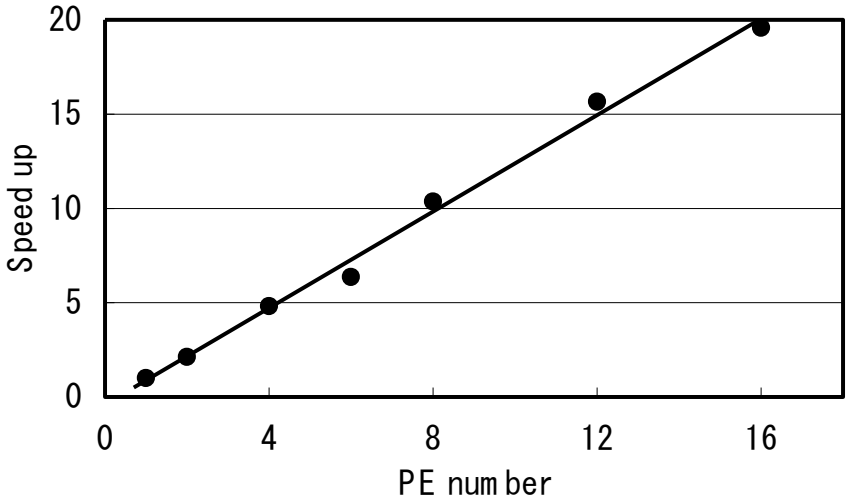


Figure 9.2.14: Accelerating Effectiveness by Domain Partitioning

Table 9.2.3: Comparison of Computing Time by Computing System (1 CPU)

CPU	(GHz)	OS	CPU Time(s)	(solver)
Xeon	2.8	Linux	850	(817)
Pentium Ⅲ	0.866	Win2000	2008	(1980)
〃 M	0.760	WinXP	1096	(1070)
〃 4	2.0	WinXP	802	(785)
〃 4	2.8	WinXP	738	(718)
Celeron	0.700	Win2000	2252	(2215)
Pentium 4	2.4	WinXP	830	(804)

9.3 Example of Actual Model for Eigenvalue Analysis

9.3.1 Analysis Model

A list of the verification examples of an actual model for the eigenvalue analysis is shown in Table 9.3.1. Among these models, the shapes of the model of EX07 (turbine rotor) and EX08 (spring) are shown in Figure 9.3.1 and Figure 9.3.2. The same shapes as in the verification example for the elastic static analysis where the verification contents are the same is used for the shape of the other models, and is already shown in the above. In order to execute an example of element type 731 and 741, a separate direct method solver is required.

Table 9.3.1: Verification Example of Actual Model for Eigenvalue Analysis

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

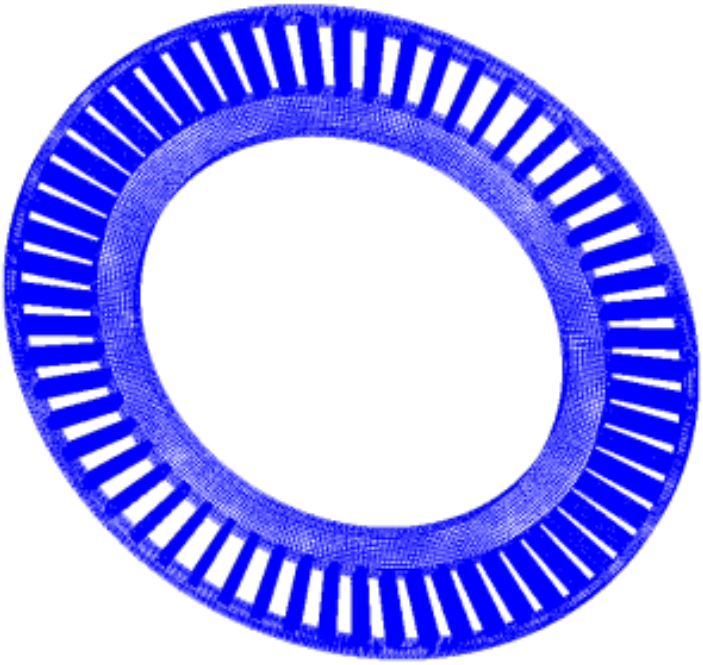


Figure 9.3.1: Turbine Rotor (EX07)

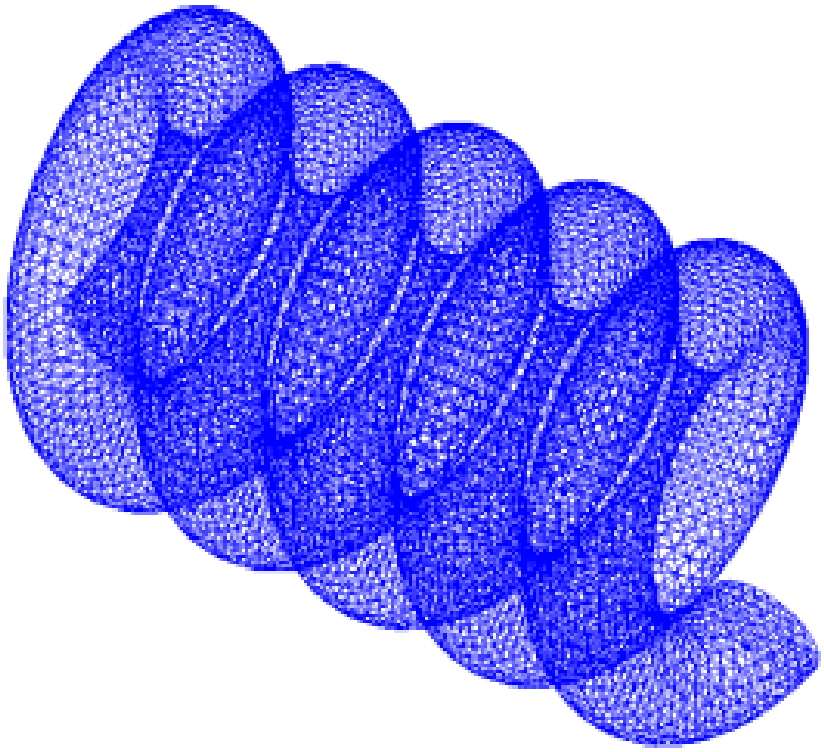
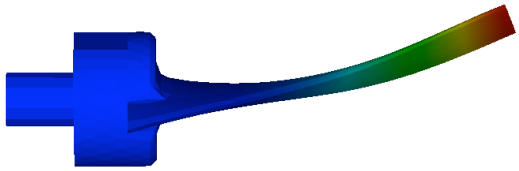


Figure 9.3.2: Spring (EX08)

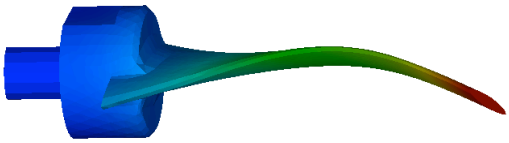
9.3.2 Analysis Results

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

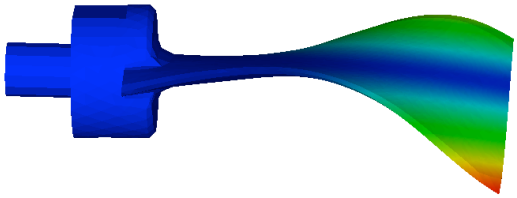
(1) EX06 Turbine blade



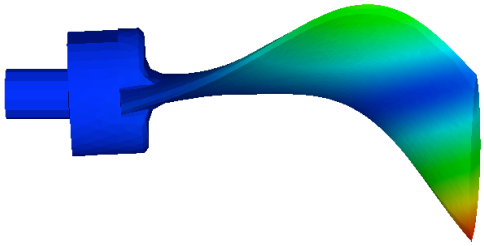
(a) Mode 1 (1170 kHz)



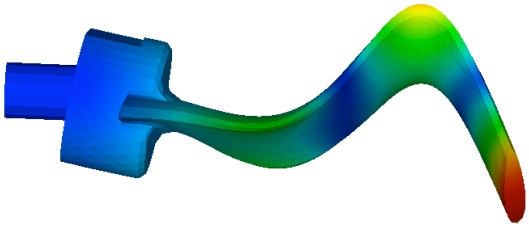
(b) Mode 2 (3250kHz)



(c) Mode 3 (4130kHz)



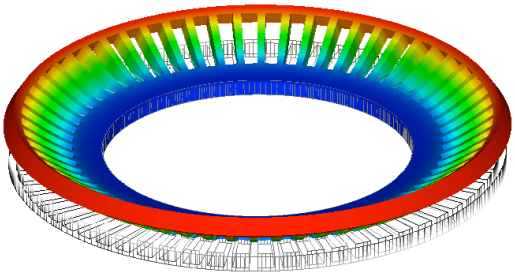
(d) Mode 4 (4140kHz)



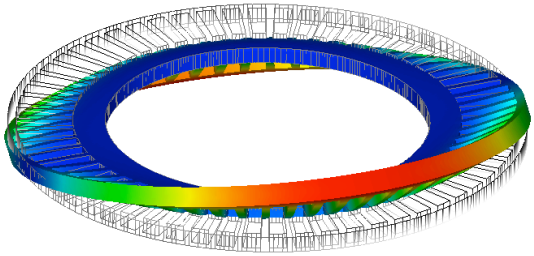
(e) Mode 5 (8210kHz)

Figure 9.3.3: EX06 Turbine Blade Vibration Mode

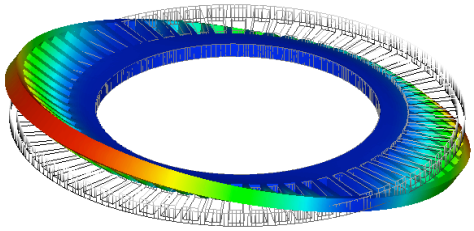
(2) EX07 Turbine rotor



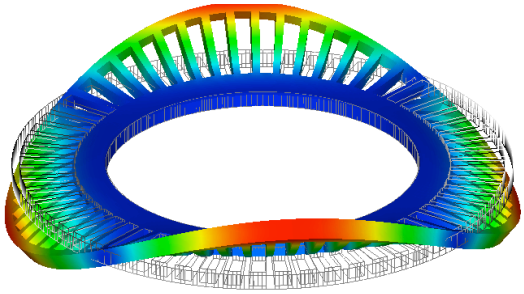
(a) Mode 1（1067Hz）



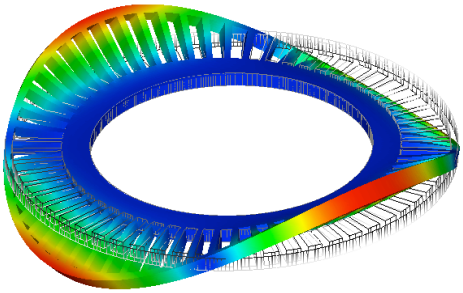
(b) Mode 2（1072Hz）



(c) Mode 3（1072Hz）



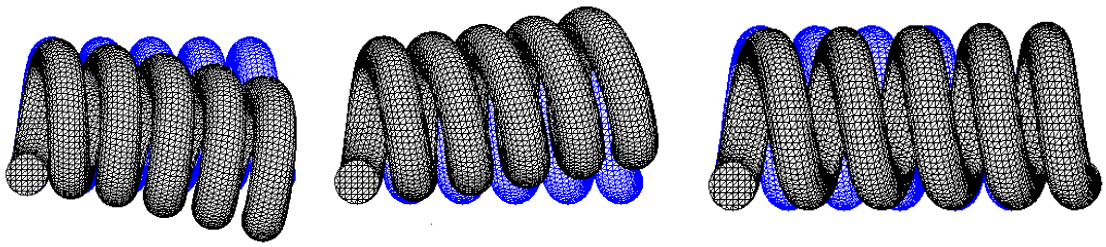
(d) Mode 4（1093Hz）



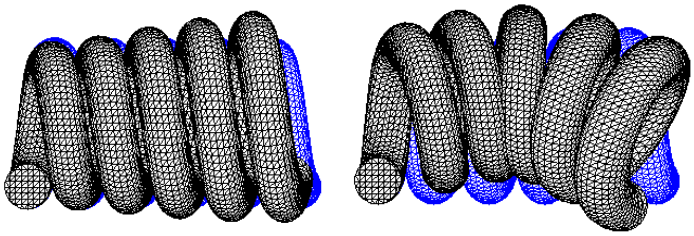
(e) Mode 5（1093Hz）

Figure 9.3.4: EX07 Turbine Rotor Vibration Mode

(3) EX08 Spring



Mode 1 (445 Hz) Mode 2 (446 Hz) Mode 3 (908 Hz)



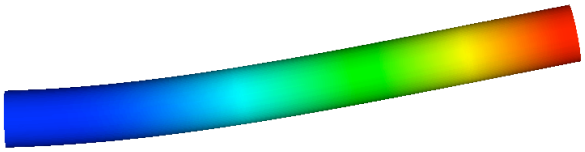
Mode 4 (985 Hz) Mode 5 (1810 Hz)

Vibration Eigenvalue

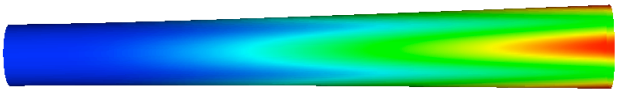
	FrontSTR	NASTRAN
1	445 Hz	445 Hz
2	446 Hz	446 Hz
3	908 Hz	908 Hz
4	985 Hz	986 Hz
5	1810 Hz	1810 Hz

Figure 9.3.5: EX08 Spring Vibration Mode

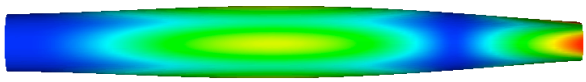
(4) EX09 Cylindrical shell



(a) Mode 1,2（109Hz）



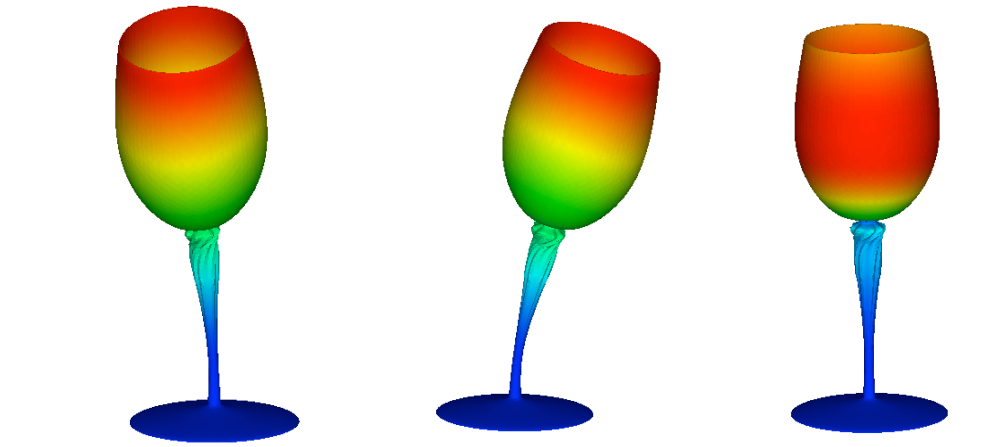
(b) Mode 3,4（570Hz）



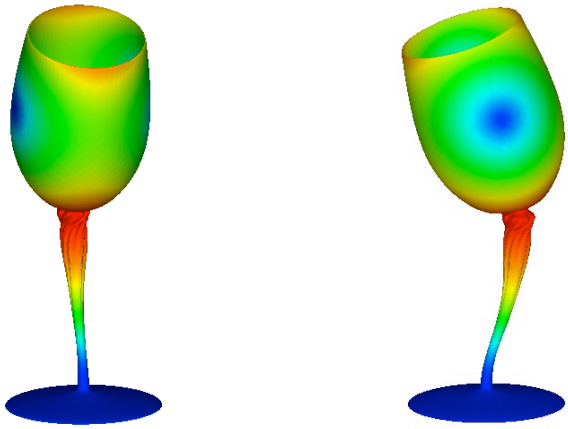
(c) Mode 5（615Hz）

Figure 9.3.6: EX09 Cylindrical Shell Vibration Mode

(5) EX10A Wine glass



(a) Mode 1（101Hz） (b) Mode 2（102Hz） (c) Mode 3（354Hz）



(d) Mode 4（1335Hz） (e) Mode 5（1336Hz）

Figure 9.3.7: EX10A Wine Glass Vibration Mode

9.4 Example of Actual Model for Heat Conduction Analysis

9.4.1 Analysis Model

In heat conduction analysis, a spent nuclear fuel shipping container was used as the actual model. A verification example of three types of actual models where the roughness of the mesh was changed was used for verification. A list is shown in Table 9.4.1. The shape of the models is shown in Figure 9.4.1 ~ Figure 9.4.4.

Table 9.4.1: Verification Example of Actual Model for Heat Conduction Analysis

Case Name	Element Type	Verification Model	No. of Nodes	No. of Degrees of Freedom
EX21A	361	Spent nuclear fuel shipping container	88,938	79,920
EX21B	361		309,941	289,800
EX21C	361		1,205,765	1,159,200

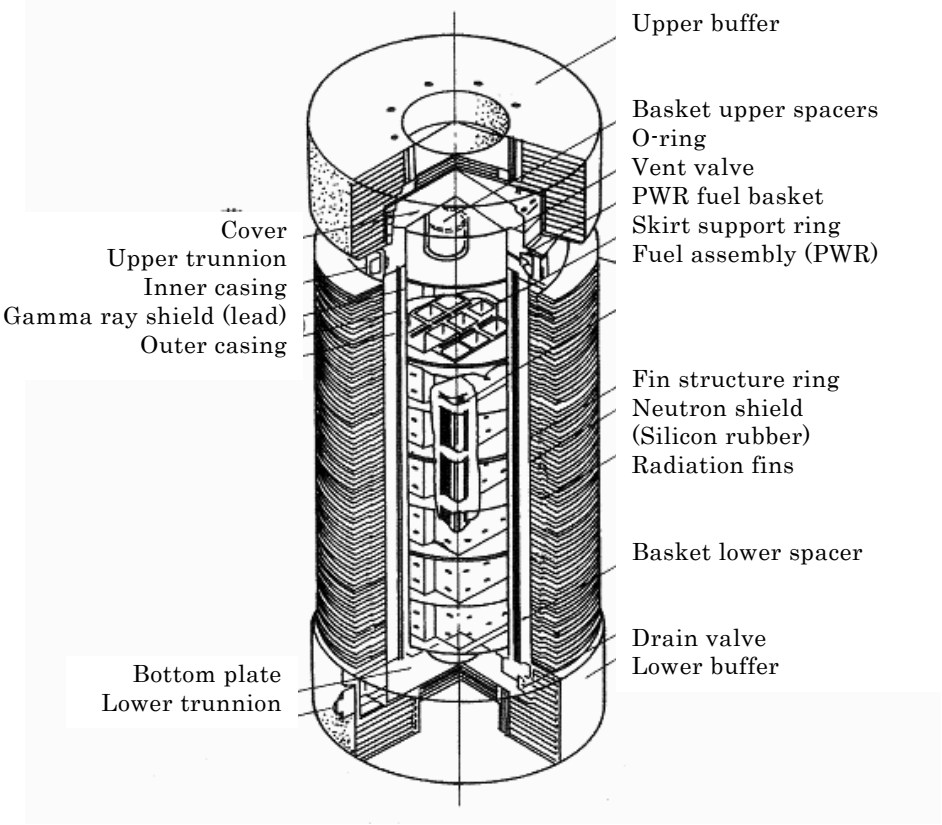
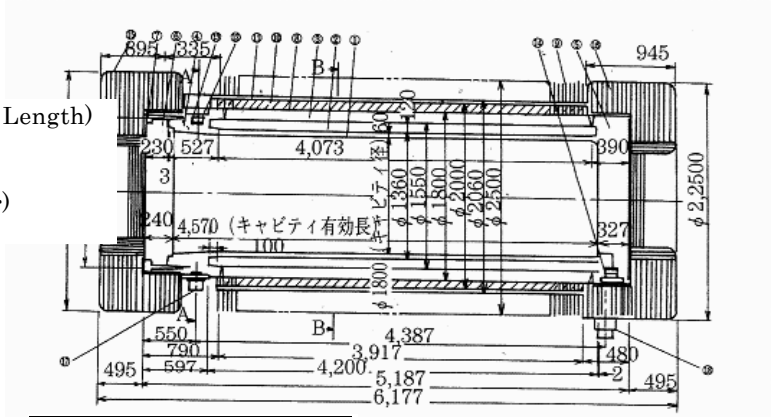


Figure 9.4.1: Spent Nuclear Fuel Shipping Container

Reference: *Transport Engineering of Nuclear Fuel Materials* (Nikkan Kogyo Shimbun) 1998

(Cavity Effective Length)

(Cavity Diameter)



No.	Name
1	Inner Casing
2	Gamma Ray Shield
3	Outer Casing
4	Top End
5	Bottom Plate
6	Cover
7	Cover Clamping Bolts
8	Fin Structure Ring
9	Radiation Fans
10	Neutron Shield
11	Neutron Shield Heat Insulating Material
12	Upper Heat Insulating Material
13	Vent Valve
14	Drain Valve
15	Upper Buffer
16	Lower Buffer
17	Upper Trunnion
18	Lower Trunnion

Name	Weight (ton)
Cover	4.4
Main Body	75.2
Basket	7.1
Fuel Assembly	6.6
Cooling Water	3.1
Upper/Lower Buffers	8.0
Total	104.4

Figure 9.4.2: Dimensions of Spent Nuclear Fuel Shipping Container

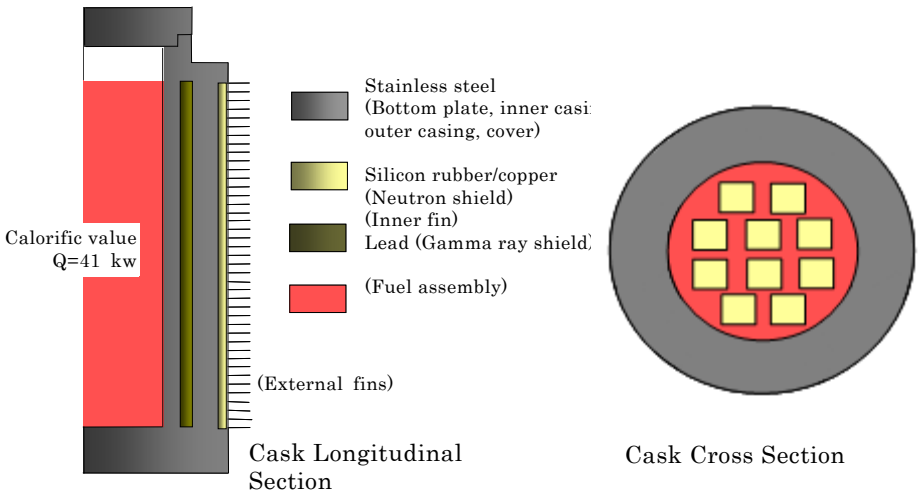


Figure 9.4.3: Conceptual Diagram of Model

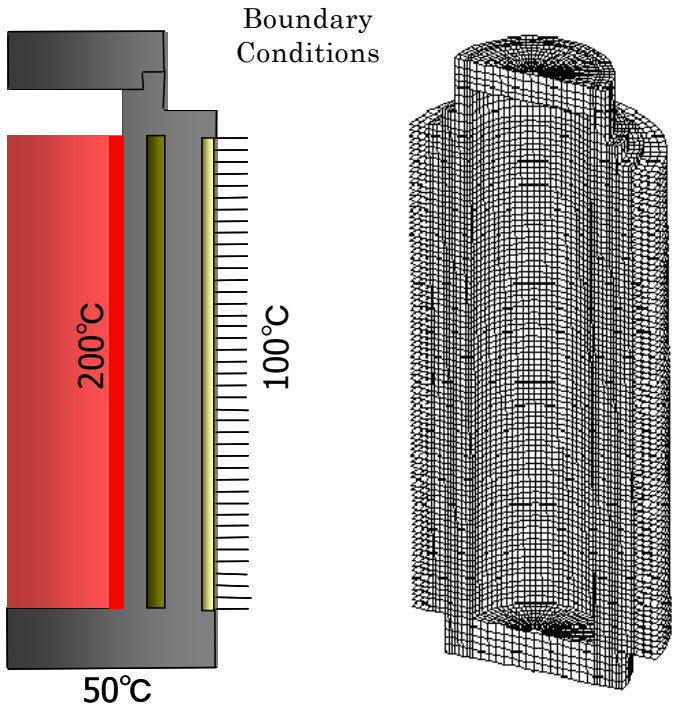
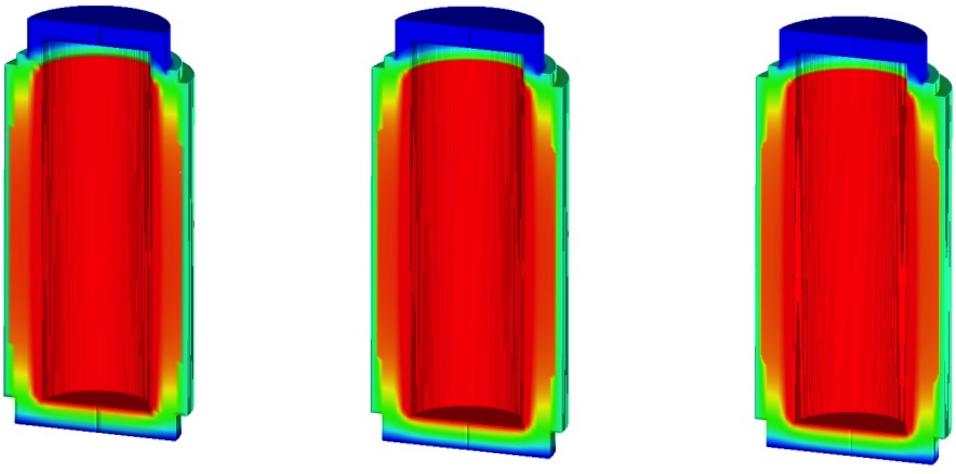


Figure 9.4.4: Model Boundary Conditions and Mesh Partitioning Figure (EX21A)

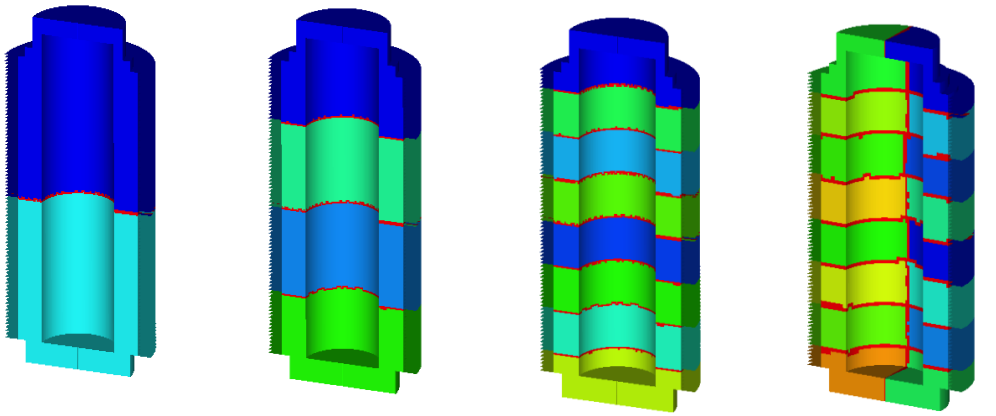
9.4.2 Analysis Results

An example of the analysis results is shown in Figure 9.4.5 ~ Figure 9.4.7.



EX21A EX21B EX21C

Figure 9.4.5: Temperature Distribution Figure



2 Partitions 4 Partitions 8 Partitions 16 Partitions

Figure 9.4.6: Distributed Model Figure

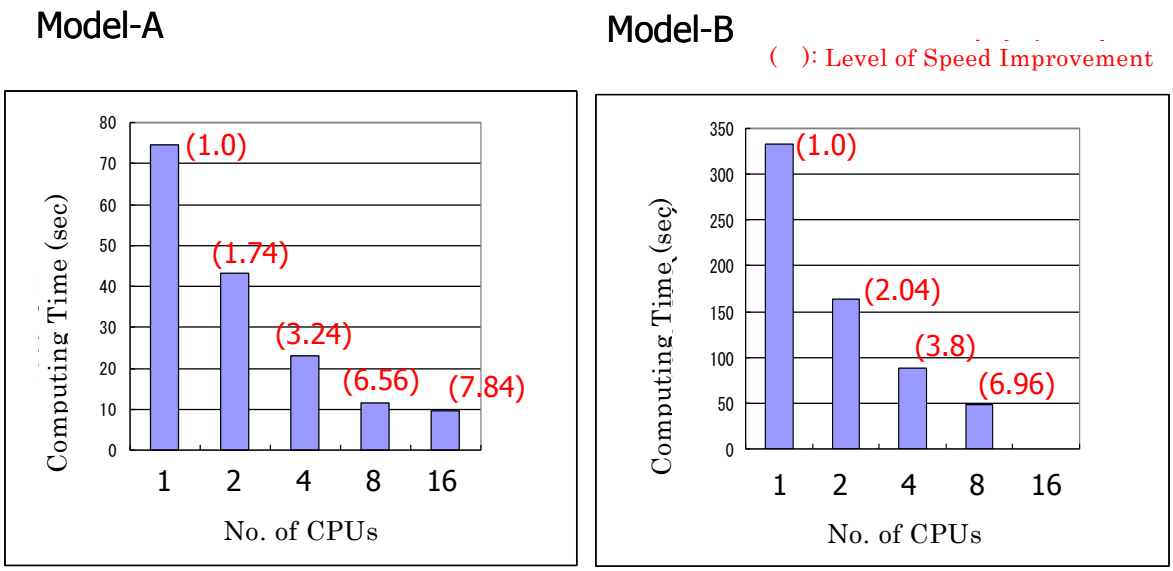


Figure 9.4.7: Level of Speed Improvement by Distribution Process

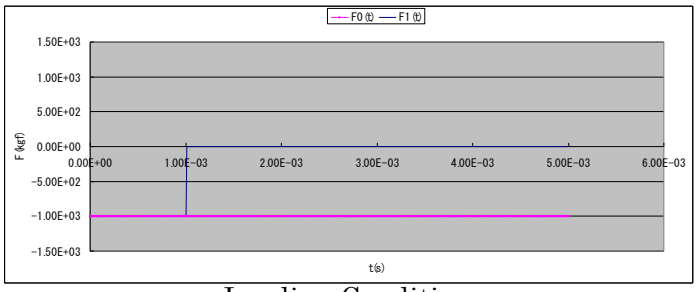
9.5 Example of Actual Model for Linear Dynamic Analysis

9.5.1 Analysis Model

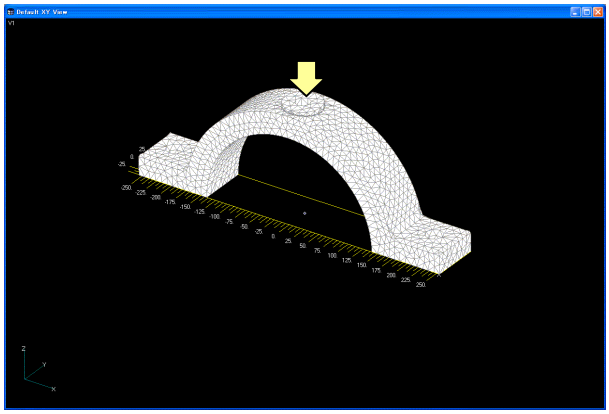
In the linear dynamic analysis, the machine parts shown in Figure 9.5.1 were used as the actual model. In this actual model, a verification example was set for four cases with different load conditions and damping coefficients. A list is shown in Table 9.5.1.

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

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



Loading Conditions

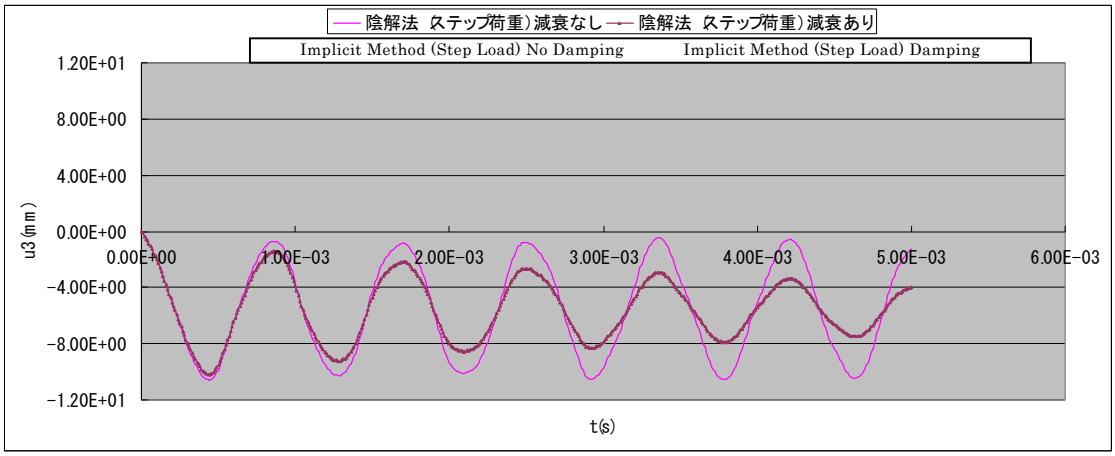


Mesh Figure

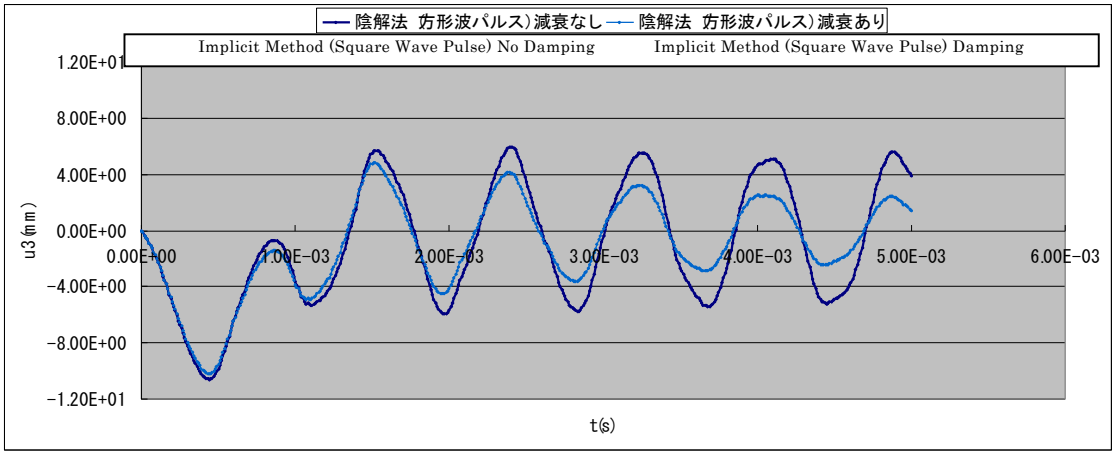
Figure 9.5.1: Mesh Model

9.5.2 Analysis Results

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

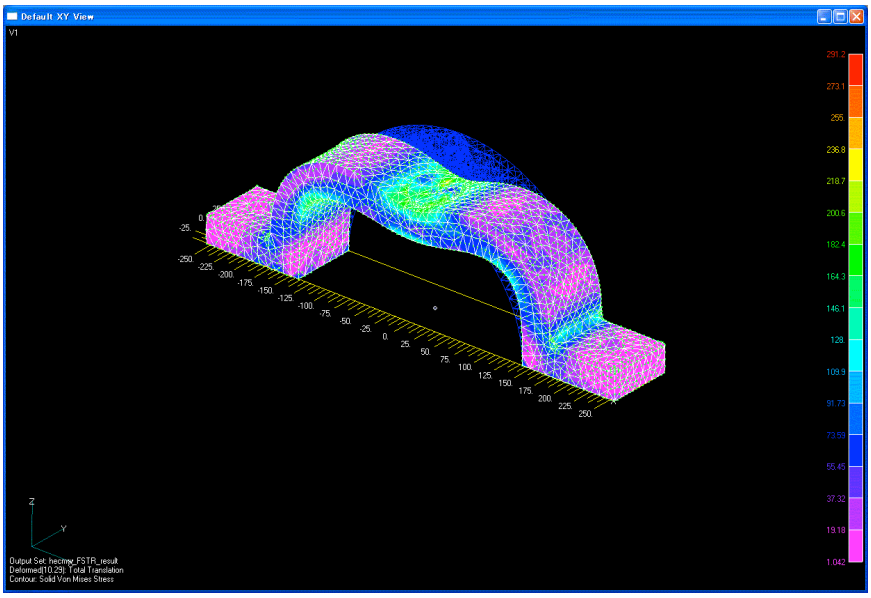


(a) In the case of Step Load

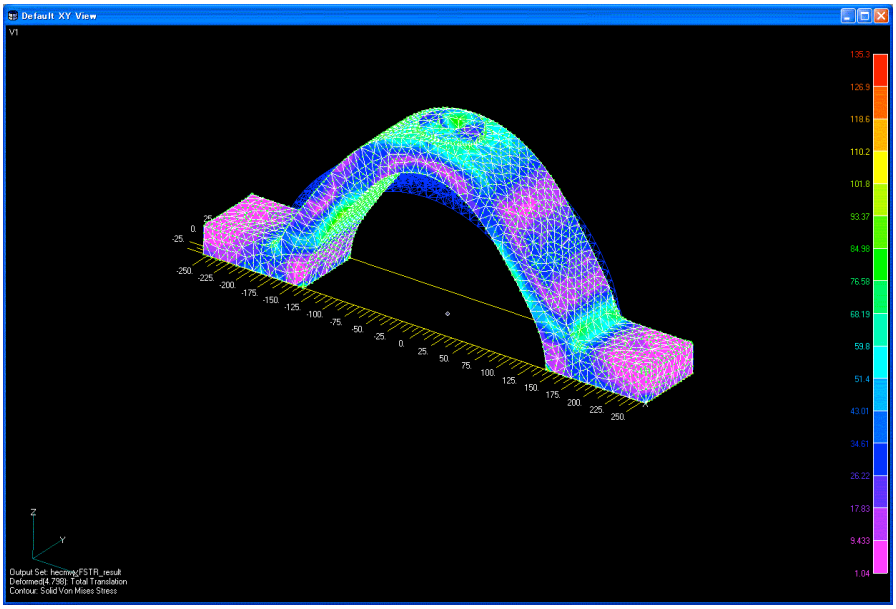


(b) In the case of Square Wave Pulse Load

Figure 9.5.2: Time History of Vibration Point Displacement u_z



(a) t=5.0E-04(s)



(b) t=4.0E-03(s)

Figure 9.5.3: Deformed Figure and Equivalent Stress Distribution

(Deformed Magnification 5.0): EX31C