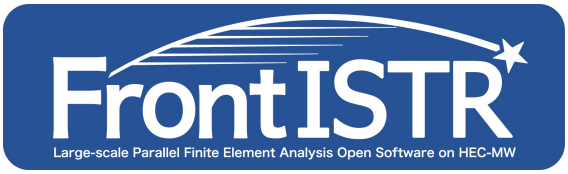
1 Introduction to FrontISTR

This software is a parallel finite element analysis program that is being continuously developed with the result of research and development “Research and development of innovation platform simulation software” project for the construction of the next generation IT infrastructure of the Ministry of Education, Culture, Sports, Science and Technology, Japan. When using this software for free or for profit use, you need to agree “MIT license”.



Item	Content
Name of Software	FrontISTR
Version	5.0
License	MIT License
	FrontISTR Commons 2-11-16 Yayoi, Bunkyo-ku, Tokyo, Japan

Correnponding Clerks
c/o Institute of Engineering Innovation,
School of Engineering
E-mail：support@frontistr.com

1.1 Manuals

- Introduction
- Installation manual
- Theory
- User’s manual
- Tutorial

1.2

Introduce you to FrontISTR.

FrontISTR is Open-Source Large-Scale Parallel FEM Program for Nonlinear Structural Analysis.

1.3 List of description on this manual

- Overview
  - Quick Start Guide
- Manuals
  - Installation manual
  - Finite Element Method Analysis Theory
  - User’s manual
  - Tutorial
- Release note
- Cheat sheet

1.4 Release Note

1.4.1 Contents updated in Ver. 5.0

The following functions were added in FrontISTR Ver. 5.0.

1.4.2 Contents updated in Ver. 3.8

The following functions were added in FrontISTR Ver. 3.8:

- Analysis function and algorithm
  - Implementation of the boundary conditions of rotational displacement
  - Implementation of the boundary conditions of torque
  - Change in the method of calculation of the reaction force of three-dimensional linear static analysis
  - Introduction of incompressible fluid analysis function (RC version)
- Elements
  - Addition of tetrahedral elements for incompressible fluid analysis (3414 elements)
  - Correction of the calculation of the stress value of (laminated) shell elements
- Materials
  - Faster loading of material definition parts inside mesh files
  - Faster loading of material definition parts inside analysis control files

    - Addition of properties of materials for incompressible fluid analysis
- Linear solvers
    - Subdivision of log output methods when using MUMPS
    - Correction of 4×4 CG solver
    - Correction of 6×6 CG solver
- Meshing and refining-related
    - Correction of errors when large-scale models are refined
- Furthermore, the following corrections were made:
    - Correction of errors of element definition with eigenvalue analysis (!STATICEIGEN) that acquires the results of non-linear static analysis
    - Correction of the memory leak of the scan_contact_state function in contact analysis
    - Correction of the warning sign displayed during compilation
    - Change in the header of the program code top
    - Change in LICENSE files

1.4.3 Contents updated in Ver. 3.7

The following functions were added in FrontISTR Ver. 3.7:

- Input/output
    - Correction of the calculation of stress value in the six degree-of-freedoms (DOFs) solver
    - Addition of calculation function for main stress and strain
    - Correction of output part of laminated shell elements
    - Addition of the INCLUDE function of the analysis files (cnt files)
    - Addition of the LINK card to the MPC input of !EQUATION
    - Change to provide Material ID (element shape ID) as output at the UCD output
    - Addition of the STEPLOG function to !SOLVER
    - Addition of the non-zero elements plot function in matrices
    - Addition of the MONITOR output function of the !SUBDIR flag
    - Addition of the output function of stimulation coefficient and valid mass (eigenvalue analysis)
    - Compatibility to large-scale meshes
    - Addition of TYPE=TIMEVALUE when !AMPLITUDE is input
    - Improvement of input function for Abaqus
    - Correction of the name search part of the MATERIAL function configured to the analysis files (cnt files)
    - Correction of the logfile output
    - Correction of global summary
- Refiner
    - Addition of the refine function to the contact problem
    - Correction of the UCD output when refining
- Analysis
    - Correction of the spring boundary conditions function
    - Faster contact pair search in contact analysis (algorithm update and OpenMP parallelism applied)
- Elements
    - Addition of the TLOAD_C3D8IC function (addition of the thermal stress load)
    - Compatibility of the truss elements (301 elements) and tetrahedral primary elements (341 elements) with parallel contact analysis
- Materials
    - Correction of errors related to the calculation of elastoplastic material when OpenMP is active
- Functions
    - Correction of the flush test

- Linear solver
  - Addition of ISAINV and IRIF preprocessing
  - Addition of the Intel PARDISO interface
  - Error correction of the part related to the OpenMP atomic descriptor
  - Error correction of the USEJAD card of !SOLVER
  - Compatibility with METIS ver. 5.0
  - Correction of the hecmw_solver_direct routine
  - Error correction of SSOR preprocessing when OpenMP = 1
  - Compatibility with elements with mixed DOFs of the calculated part of rigid body mode of ML preprocessing
- Examples and tutorials
  - Addition of examples of the shell elements for mixed DOFs (elements 761 and 781)
  - Addition of examples of the spring boundary conditions function
  - Confirmation that all the examples in the attached tutorial can be executed correctly
- Partitioner
  - The partitioner now creates the dispersion mesh of a single area correctly
- Other minor corrections
  - Initialization of fstr_setup_util.f90
  - Correction of the intent sentence

1.4.4 Contents updated in Ver. 3.6

The following functions were added in FrontISTR Ver. 3.6:

- Input/output
  - Faster file loading
  - Output change when the 781 and 761 shell elements are used
  - Debug message deleted
- Refiner
  - Correction of the refinement error when the constant of the right side was configured in the definition of !EQUATION
- Analysis
  - Corrections related to frequency response analysis
  - Corrections related to thermal stress analysis when going through !SOLUTION,TYPE=STATIC
  - Corrections related to pressure normal direction update when going through !SOLUTION,TYPE=NLSTATIC
  - Correction of the getContactStiffness subroutine of contact stiffness matrix
- Elements
  - OpenMP parallelization of the element loop
  - Corrections related to the B-bar element (solid element)
  - Corrections related to the shell elements
  - Corrections related to the truss elements
  - Correction of the invalid memory when shell, beam, truss, and solid were mixed
- Material
  - Corrections related to orthotropic elastic bodies
- Linear solver
  - Correction of the matrix dump function
  - Memory saving in 3×3 ILU preprocessing
  - Addition of the 4×4 CG solver
  - Addition of the 6×6 CG solver
  - Number of conditions of the matrix after the application of preprocessing by 3×3 CG and GMRES solver (experimental)

    - Addition of divergence check to the 3×3 CG solver
    - Reutilization of preprocessing setup information when using the 3×3 solver
    - Interface to the external AMG preprocessing library (ML) for the 3×3 solver (experimental)
    - Communication concealment with matrix vector product of the 3×3 solver (experimental)
    - Multipoint constraint processing by the explicit DOF elimination method
    - Use of the iteration method solver in contact analysis (experimental)
- Updates related to partitioner
    - Faster file input
    - Expansion of the log output function
    - OpenMP parallelization of the dispersion mesh creation loop

1.4.5 Contents updated in Ver. 3.5

The following functions were added in FrontISTR Ver. 3.5:

- Analysis function-related
    - Compatibility with analysis of a mixture of shell/beam elements and solid elements (See 3.7, 4.1, 6.3 (3))
    - Compatibility with orthotropic materials in the shell elements (See 4.2.2(3))
    - Compatibility with laminated shells (See 4.2.2(3))
    - The FOLLOW function of pressure load in large deformation analysis (See 7.4.2(14))
    - Compatibility with dynamic analysis of beam elements (See 3.7)
    - Monitoring of multiple nodes in dynamic analysis (See 7.4.5(1))
    - Monitoring of nodes stress and nodes strain in dynamic analysis (See 7.4.5(1))
    - Application of the window function to the input fluid force in coupled analysis (See 7.4.5(4))
- Partitioner-related updates
    - Significant speed increase
    - Compatibility with the Metis Ver. 5 system (see installation manual)
- Mesh and refining-related updates
    - Compatibility with the refining of models with mixed element types
    - Compatibility with interpolation based on refining information of the input temperature data
- Linear solver-related updates (See 7.4.6(1))
    - Compatibility with multicolor processing and hybrid parallelism in preprocessing
    - Ordering for vector computers
    - More combinations of preprocessing and iterative solutions that can be used in problems with three DOFs
    - The dump function of the matrix data
- The following corrections were also applied:
    - Correction of the Drilling DOF of the shell elements
    - Correction of convergence judgment in non-linear analysis
    - Correction of the restart time of linear dynamic analysis
    - Correction of the node number display in the message of coupled analysis
    - Correction of ILU preprocessing
    - Error prevention during optimization with some compilers

1.5 Cheat sheet

1.5.1 Installation

```
$ ./setup.sh -p --with-tools --with-metis
$ make
$ make install
```

1.5.2 Parallel execution

```
$ hecmw_part1
$ mpirun -np <4> fistr1
```

1.5.3 Input/Output

File type	File name	I/O
Overall control data	hecmw_ctrl.dat	I
Mesh data	<ModelName>.msh	I
Analysis control file	<ModelName>.cnt	I
Mesh partitioning control file	hecmw_part_ctrl.dat	I
Log file	<0>.log	O
Results file	<ModelName>.res	O

1.5.4 Overall control data

####（hecmw_ctrl.dat）

```
!MESH, NAME=part_in, TYPE=HECMW-ENTIRE
 <ModelName>.msh
!MESH, NAME=part_out, TYPE=HECMW-DIST
 <ModelName_p4>
!MESH, NAME=fstrMSH, TYPE=HECMW-DIST
 <ModelName_p4>
!CONTROL, NAME=fstrCNT
 <ModelName>.cnt
!RESULT, NAME=fstrRES, IO=OUT
 <ModelName>.res
```

1.5.5 Mesh partitioning control data

####(hecmw_part_ctrl.dat)

```
!PARTITION, TYPE=NODE-BASED, METHOD=PMETIS, DOMAIN=<4>
```

1.5.6 Mesh data

```
!HEADER
<TITLE>
!NODE
 NODE_ID, x, y, z
!ELEMENT, TYPE=<341>
 ELEM_ID, node1, node2, node3, ...
!SECTION, TYPE=<SOLID>, EGRP=<EG1>, MATERIAL=<MAT1>
!NGROUP, NGRP=<NG1>
 node1, node2, ...
!SGROUP, SGRP=<SG1>
 elem1, localsurf1, elem2, localsurf2, ...
!EGROUP, EGRP=<EG1>
 elem1, elem2, ...
!CONTACT PAIR, NAME=<CP1>
 <Slave_NodeGroup>, <Master_SurfaceGroup>
```

```
!AMPLITUDE, NAME=<AMP1>, VALUE=<RELATIVE|ABSOLUTE>
 value1, time1, value2, time2, ...
!INITIAL CONDITION, TYPE=TEMPERATURE
 NODE_ID, value
!EQUATION
 <number of terms>, <right-hand value>
 NODE_ID, <dof>, <coefficient>, ...
!ZERO
!END
```

1.5.7 Analysis control file (common)

```
!VERSION
 3.7
!WRITE, VISUAL, FREQUENCY=<output interval>
!WRITE, RESULT, FREQUENCY=<output interval>
!OUTPUT_VIS
 <Name of Output variable>, <ON|OFF>
!OUTPUT_RES
 <Name of Output variable>, <ON|OFF>
!RESTART, FREQUENCY=<output interval>
!END
```

Variable name	Physical value	Target
DISP	displacement	VIS,RES
ROT	rotation	VIS,RES
REACTION	nodal reactive force	VIS,RES
NSTRAIN	nodal strain	VIS,RES
NSTRESS	nodal stress	VIS,RES
NMISES	nodal mises stress	VIS,RES
ESTRAIN	elemental strain	RES
ESTRESS	elemental stress	RES
EMISES	elemental mises stress	RES
ISTRAIN	integration point strain	RES
ISTRESS	integration point stress	RES
PL_ISTRAIN	integration point plastic strain	RES
VEL	velocity	VIS,RES
ACC	acceleration	VIS,RES

1.5.8 Analysis control file (Static analysis)

```
!SOLUTION, TYPE=<STATIC|NLSTATIC>
!STATIC
!BOUNDARY,GRPID=<1>
 NODE_ID, <Start of DOF>, <End of DOF>, <Constraint value>
!CLOAD,GRPID=<1>
 NODE_ID, <DOF>, <Load value>
!DLOAD,GRPID=<1>
 SGRP, <Load type>, <Load parameter>
!SPRING,GRPID=<1>
 NODE_ID, <Constraint DOF>, <Spring constant>
```

1.5.9 Analysis control file (Contact)

```
!CONTACT_ALGO, TYPE=<SLAGRANGE|ALAGRANGE>
!CONTACT, GRPID=<1>, NTOL=<法線方向閾値>, TTOL=<接線方向閾値>, NPENALTY=<法線方向ペナルティ>,
TPENALTY=<接線方向ペナルティ>
 <Name of contact pair>, <friction coefficient>, <摩擦のペナルティ剛性>
```

1.5.10 Analysis control file (Heat stress)

```
!REFTEMP
 <temperature>
!TEMPERATURE, READRESULT=<Total number of steps>, SSTEP=<Start step>, INTERVAL=<step
interval>
```

1.5.11 Analysis control file (Eigenvalue)

```
!EIGEN
 <Nuber of eigenvalue>, <allowable value>, <maximum number of iteration>
!BOUNDARY
```

1.5.12 Analysis control file (Heat conduction)

```
!HEAT
 <DT>, <計算時間>, <時間増分>, <許容変化>, <最大反復>, <判定値>
!FIXTEMP
 NODE_ID, <温度>
!CFLUX
 NODE_ID, <熱流束>
!DFLUX
 ELEMENT_ID, <荷重タイプ>, <熱流束>
!SFLUX
 SGRP, <熱流束>
!FILM
 ELEMENT_ID, <荷重タイプ>, <熱伝達係数>, <雰囲気温度>
!SFLIM
 SGRP, <熱伝達係数>, <雰囲気温度>
!RADIATE
 ELEMENT_ID, <荷重タイプ>, <輻射係数>, <雰囲気温度>
!SRADIATE
 SGRP, <輻射係数>, <雰囲気温度>
```

1.5.13 解析制御ファイル（動解析共通）

```
!BOUNDARY
!CLOAD
!VELOCITY, TYPE=<INITIAL|TRANSIT>, AMP=<NAME>
 Node_ID, <自由度>, <自由度>, <拘束値>
!ACCELERATION, TYPE=<INITIAL|TRANSIT>, AMP=<NAME>
 Node_ID, <自由度>, <自由度>, <拘束値>
```

1.5.14 解析制御ファイル（時刻歴応答）

```
!DYNAMIC, TYPE=NONLINEAR
 <陰解法1|陽解法11>, <時刻歴1>
 <開始時刻>, <終了時刻>, <全ステップ数>, <時間増分>
 <γ>, <β>
 <集中質量|consistent質量2>, 1, <Rm>, <Rk>
 1, <モニタリング節点>, <モニタリング出力間隔>
 <変位>, <速度>, <加速度>, <反力>, <ひずみ>, <応力>
```

1.5.15 解析制御ファイル（周波数応答）

```
!DYNAMIC, TYPE=NONLINEAR
 <陰解法1|陽解法11>, <周波数2>
 <下限周波数>, <上限周波数>, <応答計算点数>, <変位測定周波数>
 <振動開始時刻>, <振動終了時刻>
 <集中質量1>, 1, <Rm>, <Rk>
 <サンプリング数>, <モード空間1|物理空間2>, <モニタリング節点>
 <変位>, <速度>, <加速度>, 0, 0, 0
```

```
!EIGENREAD
 <固有値解析のログファイル>
 <モード始点>, <モード終点>
!FLOAD
 NODE_ID, <自由度>, <荷重値>
```

1.5.16 解析ステップ

```
!STEP, TYPE=<STATIC|VISCO>, SUBSTEPS=<分割数>, CONVERG=<判定値>
 <時間増分値>, <時間増分終値>
 BOUNDARY, <GRPID>
 LOAD, <GRPID>
 CONTACT, <GRPID>
```

境界条件種類 属するカード

BOUNDARY	!BOUNDARY, !SPRING
LOAD	!CLOAD, !DLOAD, !TEMPERATURE
CONTACT	!CONTACT

1.5.17 材料物性値

```
!MATERIAL, NAME=<材料名>
!ELASTIC, TYPE=<ISOTROPIC|ORTHOTROPIC>, DEPENDENCIES=<0>
 <ヤング率>, <ポアソン比>
!DENSITY
 <質量密度>
!EXPANSION_COEFF, TYPE=<ISOTROPIC|ORTHOTROPIC>, DEPENDENCIES=<0>
 <線膨張係数>

!PLASTIC, YIELD=MISES, HARDEN=BILINEAR, DEPENDENCIES=<0>
 <初期降伏応力>, <硬化係数>

!PLASTIC, YIELD=MISES, HARDEN=MULTILINEAR, DEPENDENCIES=<0>
 <降伏応力>, <塑性ひずみ>
 <降伏応力>, <塑性ひずみ>
 ...

!PLASTIC, YIELD=MISES, HARDEN=SWIFT, DEPENDENCIES=<0>
 <ε0>, <K>, <n>

!PLASTIC, YIELD=<Mohr-Coulomb|Drucker-Prager>, HARDEN=BILIENAR, DEPENDENCIES=<0>
 <粘着力>, <内部摩擦角>, <硬化係数>

!HYPERELASTIC, TYPE=NEOHOOKE
 <C10>, <D>

!VISCOELASTIC
 <せん断緩和弾性率>, <緩和時間>

!CREEP, TYPE=Norton, DEPENDENCIES=<0>
 <A>, <n>, <m>
```

1.5.18 ソルバー制御

```
!SOLVER, METHOD=<CG>, PRECOND=<1>, MPCMETHOD=<3>
 <反復回数>, <前処理繰り返し数>, <クリロフ>, <目標色数>
 <打切り誤差>, <対角成分倍率>, 0.0
```

解法	**備考**
CG	
BiCGSTAB	
GMRES	クリロフ部分空間数を設定すること

GPBiCG
DIRECT
DIRECTmkl 接触解析で使う
MUMPS

値　前処理

1,2 SSOR
3　Diagonal Scaling
5　AMG
10　Block ILU(0)
11　Block ILU(1)
12　Block ILU(2)

値 MPC手法

1　ペナルティ法
2　MPC-CG法
3　陽的自由度消去法

1.5.19 Post-process (Output of AVS format)

```
!VISUAL
!output_type=COMPLETE_REORDER_AVS
```

1.5.20 Post-process (Output of BMP format)

```
!VISUAL, method=PSR
!surface_num=1
!surface
!surface_style=1
!display_method=1
!color_comp_name=STRESS
!color_comp=7
!x_resolution=800
!y_resolution=600
!output_type=BMP
```

1.5.21 Non-linear analysis

Type of analysis	Related card
Static analysis	!SOLUTION, TYPE=NLSTATIC !STEP
Dynamic analysis	!DYNAMIC, TYPE=NONLINEAR !STEP
Material nonlinearlity	!MATERIAL !PLASTIC !HYPERELASTIC !VISCOELASTIC !CREEP