Introduction to FrontISTR

FrontISTR Commons

February 24, 2023

Contents

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	5.4
License	MIT License
Correnponding Clerks	FrontISTR Commons2-11-16 Yayoi, Bunkyo-ku, Tokyo, Japanc/o Institute of Engineering Innovation, School of EngineeringE-mail support@frontistr.org

1.1 Manuals

- Introduction
- Installation manual
- Theory
- User's manual
- Tutorial
- FAQ

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

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

1.4 Release Note

1.4.1 Contents updated in Ver. 5.4

- Improvements
  - issue 141: Keep the FLOPS value out.
  - issue 425: Add test sample for contact analysis

  - issue 459: Improve efficiency of hecmw_mat_con
- Specification changes
  - issue 101: Define Lagrange multiplier matrix in HEC-MW.
  - issue 316: Remove Document(Manual)-CI from FrontISTR's gitlab-ci.
  - issue 421: Always use conMAT for matrix structures in contact analysis.
  - issue 422: Refactoring fstr_matrix_con_contact
  - issue 423: Rename fstrMAT and move it to the middleware side
  - issue 424: Refactoring of solve_LINEQ_contact solvers on FrontISTR side
  - issue 438: Clean up cmake_minimum_required
- Bugs fixed
  - issue 205: Shell element stratified results output does not output mises stresses.
  - issue 338: Error stop when coordinate value contains a floating point number with one significant digit in the mantissa.
  - issue 371: Occurrence of division by zero in certain development environments
  - issue 378: Restart input/output problem for dynamic analysis with SURF-SURF contact
  - issue 381: Tutorial 04_hyperelastic_spring does not converge
  - issue 397: Error loading temperature (two or more temperatures) dependent property values for creep Norton law
  - issue 409: SIGSEGV during partitioning of contact model
  - issue 411: [bug] Contact analysis functionality in parallel computation (continued from inquiry 1220302050) (1562686629)
  - issue 427: Region partitioning fails for models with contact
  - issue 428: Error when hostname Rank is longer than Rank 0
  - issue 429: When Lagrange multiplier method and MPC constraint with degrees of freedom elimination method are used together in parallel analysis, their matrix resizing processes (e.g. hecmw_mpc_mat_ass) are not consistent and the analysis fails.
  - issue 439: In a multi-step analysis of unsteady heat transfer, the totaltime is not progressing properly.
  - issue 457: Bug with contact pair name resolution.
  - issue 458: Hecmw_solver_las_22 must call 2_R instead of hecmw_update_3_R
  - issue 460: Internal direct method and DIRECTmkl are not available in sequential contact analysis

1.4.2 Contents updated in Ver. 5.3

- Improvements
  - issue 103: Allow contact parameters to be specified as input data
  - issue 266: Enable MKL for windows executables.
  - issue 339: Enable trilinos(AMG) amesos in windows binary.
  - issue 364: Multi-arch CI
  - issue 388: Illegal memory access when monitoring non-existent node with global id 0
  - issue 387: Visualization of nodal and element numbers
  - issue 384: Ability to specify parameter beta in heat transfer analysis
- Specification changes
  - issue 352: Fix macro naming notation errors.
  - issue 358: Rethinking the default settings of AMG preprocessing by ML
  - issue 359: Output messages when Amesos and MUMPS are not available in ML.
  - issue 360: Simplify ML configuration log.
- Bugs fixed
  - issue 125: Eigenvalues of 0 do not appear in constraint-free eigenvalue analysis
  - issue 301: Discussion on how to handle serial version of MUMPS with FrontISTR cmake.
  - issue 329: In iterative contact analysis with friction, convergence behavior of NR method is significantly different between sequential and distributed runs.
  - issue 333: mumps solver does not work on Windows x64 executable (MPI ver.)
  - issue 336: Binary VTK output by windows binary cannot be read by paraview.
  - issue 340: Deviation of calculation results of eigenvalue analysis due to differences in usage environment.
  - issue 343: The program crashes when a large number of contact points (about several thousand) are judged to be in contact.
  - issue 350: When building serial version, if MKL is enabled, clustermkl is also enabled, resulting in link error.
  - issue 351: When building serial version with setup.sh, MKL is not enabled even if –with-mkl is specified.

  - issue 363: Ctrl+x does not work when starting FrontISTR with docker run.
  - issue 365: Refiner in binary distribution (Docker/Windows binary) has a bug.
  - issue 366: Error when specifying contact pressure visualization file output for models with no contact.
  - issue 368: Enable parallel make when using setup.sh
  - issue 369: Avoid error when element output is specified for !OUTPUT_VIS
  - issue 376: Name resolution bug in setting up orthotropic anisotropic material
  - issue 377: The default setting of DWITH_MPI option in cmake should be ON, but it is OFF.
  - issue 380: Regarding the creation of hecmw_vis.ini, errors due to timing inconsistencies between MPI processes

1.4.3 Contents updated in Ver. 5.2

- Improvements
  - issue 079: Output of contact normal force and friction force per unit area
  - issue 142: Specify num of parts in hecmw_part1 command line argument
  - issue 218: Describe the result (res) file format in the documentation.
  - issue 298: Cluster Paradiso support for Docker containers.
  - issue 303: Make Metis ordering available in internal direct method
  - issue 305: Enable RCM ordering with internal direct method.
  - issue 313: JAD SpMV is not OpenMP parallelized
  - issue 314: BLAS Lv.1 operations are not OpenMP parallelized
  - issue 315: Apply parallelized BLAS Lv.1 functions to CG methods
  - issue 324: Anisotropic hyperelasticity
- Specification changes
  - issue 239: Different format of build date and execute date in standard output.
  - issue 262: Exclude result output for internal patch elements created by contact and MPC.
  - issue 274: Fix INFINITE to INFINITESIMAL.
  - issue 302: Refactoring the internal direct method solver
  - issue 307: Update mingw build image for Windows binary creation
  - issue 309: FATAL error is too severe when you have MUMPS and no scalapack
  - issue 326: Remove unnecessary or unimportant version notations
- Bugs fixed
  - issue 045: Weird behavior of internal direct method?
  - issue 111: AMPLITUDE boundary condition can be specified with other than AMP=.
  - issue 280: Can't compile on FrontISTR-5.1 or master branch with Trilinos-13.0.0.
  - issue 290: The version number at runtime is different from the actual version.
  - issue 292: The link to the manual in README.md has not been updated.
  - issue 300: Bugin DLOAD when multiple face groups are set.
  - issue 304: Fix for RCM ordering.
  - issue 306: msmpi.dll is not shipped.
  - issue 327: Bug fix for DEBUG build not going through.
  - issue 328: Error when specifying ISTRESS/ISTRAIN output when defining SURF-SURF contact.
  - issue 332: Couldn't build with setup.sh anymore.

1.4.4 Contents updated in Ver. 5.1.1

- Improvements
  - issue 108: Output a time to VTK
- Specification changes
  - issue 156: Make test error detection relative to the reference.
  - issue 237: add cmake rules for new MUMPS version
  - issue 247: Fix autoinc example
  - issue 282: Massive error messages when running MPI in parallel with Docker container-provided FrontISTR.
  - issue 284: Support DAG feature in Gitlab CI/CD.
  - issue 288: Find Intel MKL with cmake.
- Bug fixes
  - issue 026: ELEMCHECK Sparsity Overflow
  - issue 052: Delete compile warnings (ongoing)
  - issue 244: Compiling error by gfortran-10

  - issue 245: bug : unable to compile in setup.sh
  - issue 259: fix a bug in Positive DOF elimination with MPC
  - issue 278: Minor bug fixes
  - issue 279: mkl not found in cmake
  - issue 281: Spelling mistakes
  - issue 283: Minor fixes to the Fbar element
  - issue 285: The 741 Shell eigenvalue parsing may fail.
  - issue 287: Unable to run tests in bash on the mac
  - issue 289: heat/exU2 sometimes fails on test_hybrid due to numerical error

1.4.5 Contents updated in Ver. 5.1

- Improvement
  - issue 207: Add build date output
  - issue 204: Add samples for unsteady heat transfer analysis
  - issue 195: Refiner-related debug output
  - issue 194: Positive DOF elimination is now available when refining models with MPCs
  - issue 183: Clean up and extend the ML interface
  - issue 169: linux binary distribution
  - issue 143: Add a conrod model to tutorial
  - issue 140: Manuals Generation by CI
  - issue 136: Test support for OpenMP thread and MPI parallelism
  - issue 132: Outputting the number of cores used when using OpenMP thread and MPI in parallel.
  - issue 067: Elastoplastic consistent tangential stiffness implementation (Mises only)
  - issue 036: Interpolation using time information in reading heat transfer result files for automatic time increments
- External specification changes
  - issue 131: Removing the FrontISTR starter
- Internal Specifications Change
  - issue 158: Changed internally generated nodal group naming convention when entering a contact surface-surface pair
  - issue 122: Enhanced Testing
  - issue 121: gitlab-ci fix (extended testing)
  - issue 107: Implementing FILM and RADIATE functions using HCEMW
  - issue 056: Duplicate code removed: fistr1/src/lib/physics/ElasticNeoHooke.f90
  - issue 046: Check programmer information
  - issue 044: Output integration (static and dynamic)
  - issue 037: Change the res_bin_io.inc and res_txt_io.inc extensions.
- Bug Fixes
  - issue 203: 611 Fixed a bug in a beam element and added a test example
  - issue 185: Fix bug in eigenvalue analysis result file (.res)
  - issue 184: Linear Dynamic Analysis (Implicit Method) for Linear Analysis
  - issue 181: 541 Fixing the mass sequence when using interface elements
  - issue 159: Improve readability of internal variables (e.g. DISTCLR_CONT)
  - issue 123: Fixed a bug in applying ML preprocessing to heat transfer analysis.
  - issue 114: Fixed a bug when setting a subdirectory to output VTK files.
  - issue 113: Fixed some output issues
  - issue 055: Initializing the variable iexit in subroutine fstr_solve_dynamic_nlimplicit
  - issue 054: Fix the fstr_ctrl_get_data_array_ex argument used in the read_user_matl function
  - issue 053: Dynamic nonlinear parallelism with memory leak correction (combined with SOR preprocessing)
  - issue 026: Fix ELEMCHECK sparsity value

1.4.6 Contents updated in Ver. 5.0

The following functions were added in FrontISTR Ver. 5.0.

1.4.7 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

1.4.8 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

1.4.9 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=NL-STATIC
  - 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

1.4.10 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 Install

```
$ tar xzf FrontISTR-v5.1.tar.gz
$ cd FrontISTR-v5.1
$ mkdir build; cd build
$ cmake .. -DCMAKE_INSTALL_PREFIX=$HOME/local
$ make -j2; make install
```

1.5.2 Parallel Execution

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

1.5.3 Input

File Type	File Name
Overall Control Data	hecmw_ctrl.dat
Mesh Data	.msh
Analysis Control Data	.cnt
Partitioning Control Data	hecmw_part_ctrl.dat

1.5.4 Output

File Type	File Name
Log File	<0>.log
Analysis Result File	.res.<0>.
Visualization File	_vis_psf..pvtu

1.5.5 Overall Control Data (hecmw_ctrl.dat)

```
!MESH, NAME=part_in, TYPE=HECMW-ENTIRE
 <ModelName>.msh
!MESH, NAME=part_out, TYPE=HECMW-DIST
 <ModelName>.p
!MESH, NAME=fstrMSH, TYPE=HECMW-DIST, REFINE=<1>
 <ModelName>.p
!CONTROL, NAME=fstrCNT
 <ModelName>.cnt
!RESTART, NAME=restart_in, IO=INOUT
 <ModelName>.restart
!RESULT, NAME=fstrTEMP, IO=IN
 <ModelName>.res
!RESULT, NAME=fstrRES, IO=OUT, TYPE=BINARY
 <ModelName>.res
!RESULT, NAME=vis_out, IO=OUT
 <ModelName>_vis
!SUBDIR, ON
```

1.5.6 Partitioning Control Data (hecmw_part_ctrl.dat)

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

1.5.7 MeshData

```
!HEADER
 <TITLE>
!NODE
 <NODE_ID>, <x>, <y>, <z>
!ELEMENT, TYPE=<341>, EGRP=<E1>
 <ELEM_ID>, <node1>, <node2>, <node3>, ...
!MATERIAL, NAME=<STRMAT>, ITEM=<3>
!ITEM=1, SUBITEM=2
 <YoungModulus>, <PoissonRatio>
!ITEM=2
 <Density>
!ITEM=3
 <ExpansionCoeff>
!MATERIAL, NAME=<HEATMAT>, ITEM=<3>
!ITEM=1, SUBITEM=2
 <Density>, <Temperature>
!ITEM=2, SUBITEM=2
 <SpecificHeat>, <Temperature>
!ITEM=3, SUBITEM=2
 <Conductivity>, <Temperature>
!SECTION, TYPE=<SOLID>, EGRP=<E1>, MATERIAL=<STRMAT>
!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>, ...
!EQUATION
 <Num_terms>, <RHS>
 <NODE_ID>, <dof>, <coeff>, ...
!ZERO
 <AbsoluteZero>
!END
```

1.5.8 Version

```
!VERSION
 5
```

1.5.9 Static Analysis

```
!SOLUTION, TYPE=STATIC
!STATIC
!BOUNDARY,GRPID=<1>
 <NODE_ID>, <StartDOF>, <EndDOF>, <Value>
!CLOAD,GRPID=<1>
 <NODE_ID>, <DOF>, <LoadValue>
!DLOAD,GRPID=<1>
 <SGRP>, <LoadType>, <LoadParameter>
!SPRING,GRPID=<1>
 <NODE_ID>, <DOF>, <SpringConstant>
```

1.5.10 Contact

```
!CONTACT_ALGO, TYPE=<SLAGRANGE|ALAGRANGE>
!CONTACT, GRPID=<1>, NTOL=<NormalThreshold>, TTOL=<TangentThreshold>, NPENALTY=<NormalPenalty
 <ContactPair>, <FrictionCoeff>, <FrictionPenalty>
```

1.5.11 Thermal Stress

```
!REFTEMP
 <Temperature>
!TEMPERATURE, READRESULT=<ResultStep>, SSTEP=<FirstStep>, INTERVAL=<StepInterval>
```

1.5.12 Eigen

```
!SOLUTION, TYPE=EIGEN
!EIGEN
 <NumOfEigenvalues>, <Allowance>, <MaxIterations>
!BOUNDARY
```

1.5.13 Heat Conduction

```
!SOLUTION, TYPE=HEAT
!HEAT
 <DT>, <CalcTime>, <TimeIncrement>, <Allowable>, <MaxIteration>, <Allowance>
!INITIAL_CONDITION, TYPE=<TEMPERATURE>
 <NODE_ID>, <Temperature>
!FIXTEMP
 <NODE_ID>, <Temperature>
!CFLUX
 <NODE_ID>, <HeatFlux>
!DFLUX
 <ELEMENT_ID>, <LoadType>, <HeatFlux>
```

```
!SFLUX
 <SGRP>, <HeatFlux>
!FILM
 <ELEMENT_ID>, <LoadType>, <HeatTransferCoeff>, <AmbientTemp>
!SFLIM
 SGRP, <HeatTransferCoeff>, <AmbientTemp>
!RADIATE
 ELEMENT_ID, <LoadType>, <RadiationFactor>, <AmbientTemp>
!SRADIATE
 SGRP, <RadiationFactor>, <AmbientTemp>
!WELD_LINE
 <Current>, <Voltage>, <HeatInput>, <TorchSpeed>
 EGRP, <DOF>, <StartPoint>, <EndPoint>, <TorchWidth>, <StartTime>
```

1.5.14 Dynamic Analysis

```
!SOLUTION, TYPE=DYNAMIC
!BOUNDARY
!CLOAD
!DLOAD
!SPRING
!VELOCITY, TYPE=<INITIAL|TRANSIT>, AMP=<NAME>
 NODE_ID, <DOF>, <DOF>, <RestrictedValue>
!ACCELERATION, TYPE=<INITIAL|TRANSIT>, AMP=<NAME>
 NODE_ID, <DOF>, <DOF>, <RestrictedValue>
!INITIAL_CONDITION, TYPE=<VELOCITY|ACCELERATION>
 NODE_ID, <DOF>, value
```

1.5.15 Time History Response Analysis

```
!DYNAMIC, TYPE=<LINEAR|NONLINEAR>
 <ImplicitMethod1|ExplicitMethod11>, 1
 <StartTime>, <EndTime>, <NumberOfSteps>, <TimeIncrement>
 <gamma>, <beta>
 <LumpedMass1|ConsistentMass2>, 1, <Rm>, <Rk>
 1, <MonitoringNode>, <OutputInterval>
 <Displacement>, <Velocity>, <Acceleration>, <Reaction>, <Strain>, <Stress>
```

1.5.16 Frequency Response Analysis

```
!DYNAMIC, TYPE=NONLINEAR
 <ImplicitMethod1|ExplicitMethod11>, 2
 <MinFrequency>, <MaxFrequency>, <NumOfDivisions>, <MeasurementFrequency>
 <StartTime>, <EndTime>
 <LumpedMass1>, 1, <Rm>, <Rk>
 <ResultInterval>, <Mode1|TimeHistory2>, <MonitoringNode>
 <Displacement>, <Velocity>, <Acceleration>, 0, 0, 0
!EIGENREAD
 <EigenAnalysisLog>
 <StartMode>, <EndMode>
!FLOAD
 NODE_ID, <DOC>, <LoadValue>
```

1.5.17 Analysis Step

```
!STEP, TYPE=<STATIC|VISCO>, SUBSTEPS=<NumOfSubsteps>, CONVERG=<Threshold>, MAXITER=<MaxItera
 <TimeIncrement>, <EndValueOfTimeIncrement>
 BOUNDARY, <GRPID>
 LOAD, <GRPID>
 CONTACT, <GRPID>
```

1.5.18 Auto Time Increment

```
!AUTOINC_PARAM, NAME=<AP1>
 <DecreaseRate>, <MaxIteration>, <TotalIteration>, <ContactIteration>, <NumOfDecreaseSubstep
 <IncreaseRate>, <MaxIteration>, <TotalIteration>, <ContactIteration>, <NumOfIncreaseSubstep
 <CutbackRate>, <NumberOfCutbacks>
!TIME_POINTS, NAME=<NameofList>, TIME=<STEP|TOTAL>
 <TIME>
!STEP, TYPE=<STATIC|VISCO>, SUBSTEPS=<MaxSubsteps>, CONVERG=<Threshold>, MAXITER=<MaxIteratio
 <InitialTimeIncrement>, <StepIncrement>, <UpperLimitOfTimeIncrement>, <LowerLimitOfTimeIncre
 BOUNDARY, <GRPID>
 LOAD, <GRPID>
 CONTACT, <GRPID>
```

Boundary Condition Type	Card
BOUNDARY	!BOUNDARY, !SPRING
LOAD	!CLOAD, !DLOAD, !TEMPERATURE
CONTACT	!CONTACT

1.5.19 Output

```
!WRITE, VISUAL, FREQUENCY=<OutputInterval>
!WRITE, RESULT, FREQUENCY=<OutputInterval>
!OUTPUT_VIS
 <OutputVariableName>, <ON|OFF>
!OUTPUT_RES
 <OutputVariableName>, <ON|OFF>
!OUTPUT_SSTYPE, TYPE=<SOLUTION|MATERIAL>
```

VariableName	Physical Value	Target
DISP	Displacement	VIS,RES
REACTION	Nodal Reaction 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
VEL	Velocity	VIS,RES
ACC	Acceleration	VIS,RES
TEMP	Temperature	VIS,RES

1.5.20 Restart

```
!RESTART, FREQUENCY=<n>
```

1.5.21 Local Coordinate

```
!ORIENTATION, NAME=<CoordinateSystem>, DEFINITION=COORDINATES
<ax,ay,az>,<bx,by,bz>,<cx,cy,cz>

!ORIENTATION, NAME=<CoordinateSystem>, DEFINITION=NODES
<a,b,c>
```

1.5.22 Section

```
!SECTION, SECNUM=<IndexOfSectionOfMeshData>, ORIENTATION=<CoordinateSystem>, FORM361=<FBAR|IC
```

1.5.23 Material Property

```
!MATERIAL, NAME=<NameOfMaterial>
!ELASTIC, TYPE=<ISOTROPIC|ORTHOTROPIC>, DEPENDENCIES=<0>
 <YoungsModulus>, <PoissonRatio>
!DENSITY
 <MassDensity>
!EXPANSION_COEFF, TYPE=<ISOTROPIC|ORTHOTROPIC>, DEPENDENCIES=<0>
 <LinearExpansion>

!PLASTIC, YIELD=MISES, HARDEN=BILINEAR, DEPENDENCIES=<0>
 <InitialYieldStress>, <CuringCoefficient>

!PLASTIC, YIELD=MISES, HARDEN=MULTILINEAR, DEPENDENCIES=<0>
 <YieldStress>, <PlasticStrain>
 <YieldStress>, <PlasticStrain>
  ...

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

!PLASTIC, YIELD=<Mohr-Coulomb|Drucker-Prager>, HARDEN=BILIENAR, DEPENDENCIES=<0>
 <Adhesive>, <InternalFrictionAngle>, <Curing>

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

!VISCOELASTIC
 <ShearRelaxationModulus>, <RelaxationTime>

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

1.5.24 Solver Control

```
!SOLVER, METHOD=<CG>, PRECOND=<1>, MPCMETHOD=<3>
 <MaxIteration>, <PreIteration>, <Krilov>, <Color>, <ReuseSetup>
 <TruncationError>, <DiagonalScale>, 0.0
```

Method	Notes
CG	
BiCGSTAB	
GMRES	Enable Number Of Krilov Subspaces
GPBiCG	
DIRECT	
DIRECTmkl	Use for Contact Analysis
MUMPS	

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

Value	Method of MPC
1	Penalty
2	MPC-CG
3	Explicit Elimination

1.5.25 Solver Control AMG

```
!SOLVER, METHOD=<CG>, PRECOND=5, MPCMETHOD=<3>
 <MaxIteration>, <PreIteration>, <Krilov>, <Color>, <ReuseSetup>
 <TruncationError>, <DiagonalScale>, 0.0
 <CoarseSolver>, <Smoother>, <MultigridCycle>, <MaxLevel>, <Scheme>, <Sweep>
```

1.5.26 Post Process for ParaView

```
!VISUAL
!output_type=VTK
```

1.5.27 Post Process output BMP

```
!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
```

1.5.28 Nonlinear Analysis

Analysis Type	Related Cards
Static Analysis	!SOLUTION, TYPE=NLSTATIC!STEP
Dynamic Analysis	!DYNAMIC, TYPE=NONLINEAR!STEP
Contact Analysis	!CONTACT!CONTACT_ALGO!STEP
Material Nonlinear	!PLASTIC!HYPERELASTIC!VISCOELASTIC!CREEP