ADVENTURE SYSTEM

ADVENTURE_Impact

HDDM-based explicit dynamics solver
for elastic stress analysis with contact

Version: 0.82 β

User's Manual

April 1, 2004

ADVENTURE Project

Contents

1. Installation and Summary

1.1. Contents of Package

The following files are included in the current package:

Solver

The explicit dynamics solver based on the Hierarchic Domain Decomposition method (HDDM) includes the following codes:

GrandController:	`advimpact-g`
Controller:	`advimpact-c`
Analyzer:	`advimpact-a`
Combined version of the above codes:	`advimpact-t`

Pre-Processor Tools

AdvIO converter tools:	`advimpact-datamaker`
AdvImpact data assistant creator tools:	`advimpact-converter`
AdvImpact data viewer:	`advimpact-checker`

Post-Processor Tools

Tools for extraction of chronological nodal displacement records:	`advimpact-vnode`
Tools for simplified visualization of nodal displacement distribution:	`advimpact-vdisp`

1.2. Operational Environment

The system requires the following operational environment:

Operating System (OS):	Linux, FreeBSD
Compiler:	GNU C compiler

The operational environment used for development of the current codes is:

OS :	FreeBSD
CPU:	Intel
MPI:	mpich
C compiler:	gcc-2.95.3

1.3. Program Compilation

The path for *include* files and the path for *library* files are originally setup in the `Makefile` for the situation when the ADVENTURE_IO library is installed in the directory ADVENTURE. If another directory for installation is considered, the underlined path of the `ADVDIR` should be changed.

```
ADVDIR= $(HOME)/ADVENTURE
INCDIR= ${ADVDIR}/include
LIBDIR= $(ADVDIR)/lib
```

Execute the *make* command:

`make` [ENTER]

The program executable files will be created. To complete the installation, execute the command *make install*:

`make install` [ENTER]

The binary executable modules will be copied into the directory `$(ADVDIR)/bin` and the User's Manual will be copied into the directory `$(ADVDIR)/doc/AdvImpact` . After these procedures, the *AdvImpact* executable modules can be copied into any convenient directory.

1.4. Outline

1.4.1. Architecture

The code *AdvImpact* consists of 3 independent executable modules and 1 combined executable module. In addition, 2 necessary tools are used to reprocess the data.

Executable modules:

`advimpact-g`	*GrandController*	the number of processes = 1
`advimpact-c`	*Controller*	the number of processes = number of *Parts*
`advimpact-a`	*Analyzer*:	the number of processes = unlimited
`advimpact-t`	Combined version of the above codes	

Necessary Pre-Processor tools:

`advimpact-datamaker`	*AdvIO* converter tools
`advimpact-converter`	*AdvImpact* data assistant creator tools

1.4.2. GrandController (GC)

The module *GrandController* (GC) is used for overall controll and management of the information on the connected *Parts*. GC reads the input-output and setup files (described in Section 2 of the current Manual).

1.4.3. Controller

The *Controller* modules, which are activated for each *Part*, control the input-output data of all *Subdomains* in the *Part*. The file input-output processes are done in parallel mode for the number of *Controller* modules (number of *Controllers* = number of *Parts*).

1.4.4. Analyzer

The *Analyzer* performs the Finite Element analysis (FEA) of each *Subdomain*. The number of *Analyzer* processes are fixed and are set for each *Part* in the setup file. FEA is done in parallel mode for the number of *Analyzer* processes.

1.4.5. AdvImpact – data maker

The tool program `advimpact-datamaker` is used to convert the mesh data written in the text format into the ADVENTURE binary format.

1.4.6. *AdvImpact – converter*

The tool program `advimpact-converter` is used to convert the output files created by ADVENTURE_Metis into the format supported by *AdvImpact*.

1.4.7. *AdvImpact – checker*

The tool program `advimpact-checker` is used to convert the *AdvImpact* supported files into the text format for view.

1.5. Analysis Modes

AdvImpact can perform analyses in following modes.

From the viewpoint of analysis contents:

1). Dynamic analysis mode
2). Dynamic contact analysis mode

From the viewpoint of load decentralization

1). Dynamic decentralized load
2). Static decentralized load

1.5.1. Dynamic Analysis Mode

The analysis of dynamic problems is performed by the explicit dynamics solver.

1.5.2. Dynamic Contact Analysis Mode

The analysis of dynamic contact problems is performed by the explicit dynamics solver. The switching between dynamic analysis and dynamic contact analysis is assigned by the setup flags.

1.5.3. Dynamic Load Distribution

The decentralization of the load is done dynamically that results in high usage of CPU. Since, the stiffness matrixes are recalculated each time step, the computing takes much time. The scale of the problem is not limited by memory usage and large-scale problems can be analyzed using cluster computers.

1.5.4. Static Load Distribution

The decentralization of the load is done statically and the stiffness matrixes are stored. It results in high computing speed, but the scale of analysis is limited by the computer platform memory. In the static load distribution, the calculations of each Subdomain are done by *Analyzer* which is preliminary assigned and cannot be changed during analysis.

2. Analysis Flow

The analysis is performed in the following way.

1). Preparation of the Data File 1 (A text file with non-fixed filename)
```
% vi mesh.dat
```
2). Preparation of the Data File 2 (A text file with non-fixed filename)
```
% vi fdcurve.dat
```
3). Preparation of the Data Configuration File 1 (A text file with non-fixed filename)
```
% vi in_config1.txt
```
4). Preparation of the Data Configuration File 2 (A text file with non-fixed filename)
```
% vi in_config2.txt
```
5). Preparation of the Input Configuration File 1 (A text file with fixed filename)
```
% vi AdvImpact_init_files.txt
```
6). Preparation of the Input Configuration File 2 (A text file with fixed filename)
```
% vi AdvImpact_init_contact.txt
```
7). Preparation of the Output Configuration File (A text file with fixed filename)
```
% vi AdvImpact_init_outputflag.txt
```
8). Preparation of the Environment Setup File 1 (A text file with fixed filename)
```
% vi AdvImpact_init_config.txt
```
9). Preparation of the Environment Setup File 2 (A text file with fixed filename)
```
% vi AdvImpact_init_loadbalance.txt
```
10). Preparation of the Initial Displacement Data File

11). Conversion of the input data from a text format to the ADVENTURE binary format
```
% ./advimpact-datamaker < in_config1.txt
```
12). Hierarchical Domain Decomposition using ADVENTURE_Metis
```
% mpirun -np 3 advneture_metis mesh.adv out 64
```
13). Preparation of the Data File for *AdvImpact*
```
% ./advimpact-converter < in_config2.txt
```
14). Conformation of the Data File for *AdvImpact*
```
% ./advimpact-checker < in_config2.txt
```
15). Analysis of the data by solvers

In the case of `advimpact-t` :
```
% mpirun -np 32 advimpact-t
```
Here, the number of processors = 32, the number of *GrandControllesr* =1, the number of *Controllers* = 3, the number of *Analyzers* = the number of *Parts* = 28.

In the case of `advimpact-g`, `advimpact-c`, and `advimpact-a` :
```
% mpirun -p4pg pgfile advimpact-g
```
The allocation of each binary file is described in the file `pgfile`. Refer to the MPI User's Manual for details.

16). Visualization of the analysis results

To display the chronological nodal displacement records:
```
% ./advimpact-vnode < in_visual_node.txt
```
To display the nodal displacement distribution using UCD file (*MicroAVS*):
```
% ./advimpact-vdisp < in_visual_disp.txt
```

3. Files

3.1. Data File 1 (*mesh.dat*)

The Data File 1 contains the following information.

- Element's data
- Node's data
- Forced displacement data
- Load data
- Material properties data
- Contact data (in the case of the dynamic contact analysis)

3.1.1. Element's Data

(Example: linear tetrahedral element)

```
-----------------------------------------------
200
1 2 3 4
3 21 5 8
.  .  .  .
------------------------------------------------
```

Number of elements
Node number Node number . . . (Element's nodes)
Node number Node number . . . (Element's nodes)
. . . .

```
------------------------------------------------
```

3.1.2. Node's Data

(Example)

```
-------------------------------------------------------
250
0.0000 0.0000 0.0000
0.1000 0.0000 0.0000
0.1000 0.1000 0.1000
.  .  .  .  .  .  .  .
-------------------------------------------------------
```

Number of nodes
X-coordinate Y-coordinate Z-coordinate
X-coordinate Y-coordinate Y-coordinate
. . . .

```
-------------------------------------------------------
```

3.1.3. *Forced Displacement Data*

(Example)

```
--------------------------------------------------------
5
12 0 0 1.000
12 1 0 1.000
12 2 0 1.000
82 2 2 2.000
83 1 1 1.500
--------------------------------------------------------
```

Number of forced displacements
Node's number   Axis's number   Displacement curve's number   Magnifier
Node's number   Axis's number   Displacement curve's number   Magnifier

```
.  .  .  .  .  .  .  .  .  .  .  .
--------------------------------------------------------
```

Fixed displacement conditions, both static and dynamic, are treated in the same way.
In the above-mentioned example, the forced displacements are set on:

Node No.12 in X, Y, and Z directions,
Node No.82 in Z direction,
Node No.83 in Y direction.

The values obtained from the displacement curve, set by the displacement curve's numbers, are magnified by the magnifier values to set the final forced displacement.

(Example: the displacement curve 1 from the Chapter 3.2.1)

```
--------------------------------------------------------
0 0.00 0.00
1 0.10 0.10
2 0.50 0.20
3 0.60 0.30
.  .  .  .  .  .
9 1.00 0.90
--------------------------------------------------------
```

At time t=0.40, the displacement of the node No.83 in Y direction is:
[Displacement decided from the displacement curve at t=0.40]   x   [Magnifier] =

$$= \left\{ \frac{(0.40 - 0.10)}{(0.50 - 0.10)} \times (0.20 - 0.10) + 0.10 \right\} \times 1.5 \quad =$$

= 0.2625

3.1.4. Load Data

(Example)

```
------------------------------------------------------------------------
5
15 0 0 1.000
15 1 0 2.000
15 2 0 1.000
70 2 2 2.000
75 1 1 1.500
------------------------------------------------------------------------
```

Number of set loads
Node's number   Axis's number   Load curve's number   Magnifier
Node's number   Axis's number   Load curve's number   Magnifier
. . . .

```
------------------------------------------------------------------------
```

In the above-mentioned example, the loads are set on:
 Node No.15 in X, Y, and Z directions,
 Node No.70 in Z direction,
 Node No.75 in Y direction.

The values obtained from the load curve, set by the load curve's number, are magnified by the magnifier values to set the final load.

3.1.5. Material Properties Data

(Example)

```
-------------------------------------------------------
2
1 2.05e11 0.3 7.86e3 0.0
1 7.03e10 0.345 2.69e3 0.0
0 0
1 0
2 0
3 0
4 1
5 1
. . . . . . . . . . .
197 1
198 0
199 0
-------------------------------------------------------
```

Material's number
Type of material number 0 Data Data (Number of data depends on material type)
Type of material number 1 Data Data (Number of data depends on material type)
. . . .
Element's number Material's number
Element's number Material's number
Element's number Material's number

The following types of material data are supported.

Elastic Isotropic:

Type number: 1
Material data: Young modulus, Poisson's ratio, density, and viscosity.

Treatment of the material viscosity is discussed in *Appendix A*

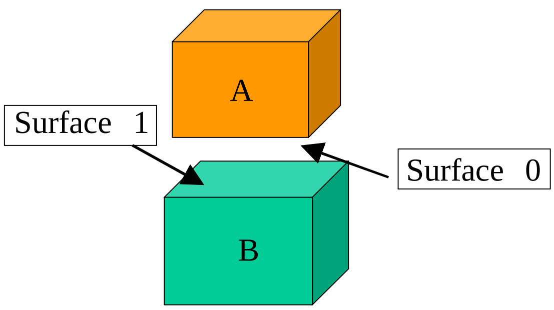
3.1.6. *Contact Data*

(Example)

```
-------------------------------------------------------
2
0.00 0.00 0.00
0.00 0.00 0.00
0 5
21 2
25 5
27 4
28 4
29 0
1 4
51 1
52 1
55 3
56 5
-------------------------------------------------------
```

Number of contact surfaces
Friction factor of the contact surface 0 (Static friction factor, dynamic friction factor, damping parameter)
Friction factor of the contact surface 1 (Static friction factor, dynamic friction factor, damping parameter)
Contact surface's number Number of contacting segments
Element's number Element's surface number
Element's number Element's surface number
. . . .
Element's number Element's surface number
Contact surface's number Number of contacting segments
Element's number Element's surface number
Element's number Element's surface number
. . . .
Element's number Element's surface number

The contacted surfaces are counted as two surfaces.



The friction factor is treated according to the equation

$$\mu = \mu_d + (\mu_s - \mu_d)e^{-c|v|}$$

Here, μ is the friction factor, μ_d is the dynamic friction factor, μ_s is the static friction factor, c is the damping parameter, and v is the relative velocity.

3.2. Data File 2 (fdcurve.dat)

Data File 2 contains the data for:

- Forced displacement curve
- Load curve

3.2.1. Forced Displacement Curve Data

(Example)

```
-------------------------------------------------------
3
2
0 0.00 0.00
1 1.0e30 0.00
10
0 0.00 0.00
1 0.10 0.10
2 0.50 0.20
3 0.60 0.30
.  .  .  .  .  .  .  .  .
9 1.00 0.90
1001
0 0.00 0.00
1 1.0e-3 0.002
2 2.0e-3 0.004
.  .  .  .  .  .  .  .  .
1000 1.00 0.9520
-------------------------------------------------------
```

Number of forced displacement curves
Number of points defining the curve No.0
Number   Time for point No.0   Displacement at point No.0
Number   Time for point No.1   Displacement at point No.1
Number of points defining the curve No.1
Number   Time for point No.0   Displacement at point No.0
Number   Time for point No.1   Displacement at point No.1
. . . .

In the case of fixed displacement, the settings of displacements should be done for sufficiently long time interval.
The displacement at the current time will be obtained from the neighbor points by linear interpolation.   For example, if the input parameters are

```
1 1.0e-3 0.002
2 2.0e-3 0.004 ,
```

then, at time `1.7e-3`, the displacement will be interpolated as:

`(1.7e-3 – 1.0e-3)/(2.0e-3 – 1.0e-3) x (0.004 – 0.002) + 0.002`.

3.2.2. *Load Curve Data*

(Example)

```
------------------------------------------------------------------------
2
5
0.00 0.00
1.0e-3 1.00
5.0e-2 1.00
5.1e-2 0.00
1.0e30 0.00
2
0.00 0.00
1.00 1.00
------------------------------------------------------------------------
```

Number of load curves
Number of points defining the curve No.0
Number Time for point No.0 Load at point No.0
Number Time for point No.1 Load at point No.1
Number of points defining the curve No.1
Number Time for point No.0 Load at point No.0
Number Time for point No.1 Load at point No.1

```
.  .  .  .  .  .  .  .  .  .  .  .  .  .
------------------------------------------------------------------------
```

The load at the current time will be obtained from the neighbor points by linear interpolation. For example, if the input parameters are

```
1 1.0e-3 0.002
2 2.0e-3 0.004 ,
```

then, at time `1.7e-3`, the load will be interpolated as:

`(1.7e-3 – 1.0e-3)/(2.0e-3 – 1.0e-3) x (0.004 – 0.002) + 0.002`.

3.3. Data Configuration File 1 (*in_config1.txt*)

The Data Configuration File 1, used by `advimpact-datamaker` tool, contains the following information:

- The name of the file after data conversion from the text format to ADVENTURE binary format.
- The number of nodes per one element.
- The flag for contact analysis.

(Example)

```
-------------------------------------------------------
mesh.dat
mesh.adv
4
1
-------------------------------------------------------
```

Name of the initial data file
Name of the converted data file (Text format -> ADVENTURE format)
Number of nodes per one element
Flag for contact analysis (0: no contact analysis; 1: perform contact analysis)

```
-------------------------------------------------------
```

3.4. Data Configuration File 2 (*in_config2.txt*)

The Data Configuration File 2, used by `advimpact-converter` and `advimpact-checker`, contains the following information.

- The flag for contact analysis.
- The number of *Parts*.
- The name of the file produced by ADVENTURE_Metis (HDDM-type FEA model data).
- The filename of *AdvImpact* Data File 1.
- The filename of *AdvImpact* Data File 2.
- The filename of Data File 2.

(Example)

```
-------------------------------------------------------
1
3
advhddm_in_0.adv
advhddm_in_1.adv
advhddm_in_2.adv
advimpact_in_0.adv
advimpact_in_1.adv
advimpact_in_2.adv
advimpact_gdlist.adv
fdcurve.dat
-------------------------------------------------------
```

Flag for contact analysis
Number of *Parts*
Name of the HDDM-type model data files x Number of *Parts*
Filename of *AdvImpact* Data File 1 x Number of *Parts*
Filename of *AdvImpact* Data File 2
Filename of Data File 2

```
-------------------------------------------------------
```

The filename of *AdvImpact* Data File 1 (for a number of *Parts*) and the filename of *AdvImpact* Data File 2 can be specified optionally.

3.5. Input Setup File 1 (*AdvImpact_init_files.txt*)

The Input Setup File 1 contains the following information:

- The filename of HDDM-type FEA model data.
- The filename of *AdvImpact* Data File 1.
- The filename of *AdvImpact* Data File 2.
- The filename of Data File 2.
- The filename of output files.
- The flag for the initial displacement.
- The name of the file contained the initial displacement.
- The number of Degrees-of-freedom (DOF) of one node.
- The number of Gauss points.
- The flag for contact analysis.
- The flag for load distribution.
- The number of time steps.
- The width of time gap.

(Example)

```
------------------------------------------------------------
0 advhddm_in_0.adv
1 advhddm_in_1.adv
2 advhddm_in_2.adv
0 advimpact_in_0.adv
1 advimpact_in_1.adv
2 advimpact_in_2.adv
advimpact_gdlist.adv
0 result0/res
1 result1/res
2 result2/res
0
3
2
1
2
5000
1.0e-8
------------------------------------------------------------
```

(*Part*'s number Filename of HDDM-type FEA model data) x number of *Parts*
(*Part*'s number Filename of *AdvImpact* Data File 1) x number of *Parts*
Filename of *AdvImpact* Data File 2
(*Part*'s number Filename of output file) x number of *Parts*
Flag for the initial displacement (0: set to 0; 1: read from file)
Number of DOF of one node
Number of Gauss points
Flag for contact analysis (0: no contact analysis; 1: perform contact analysis)
Flag for load distribution (1: static; 2: dynamic)
Number of time steps
Width of time gap

The solver sets the name of the output file by adding digits to the end of filename. For example, the filename `result0/res` will be completed by digits as `result0/res_0001000101.adv`.

The digits have the following meaning.
First 4 digits from the head of the name: the *Part* number.
Next 4 digits: the *Subdomain* number
Next 2 digits: the physical quantities

The physical quantities are:
00: The nodal displacement;
01: The strain at the integral point;
02: The stress at the integral point.

The flags for initial displacement can be selected from:
0: All initial displacements are set to 0;
1: Reads the initial displacements from a file.

If flag 1 is selected, the binary file containing initial displacements should be prepared in advance. In this case, the name of each file is `dp0/dp_00010001`. The digits have the following meaning. First 4 digits from the head of the name: the *Part* number. Next 4 digits: the *Subdomain* number.
The contents of the file are written in binary (`float 64`) format:

```
---------------------------------------------------------
    For node No.0: X displacement, Y displacement, and Z displacement
    For node No.1: X displacement, Y displacement, and Z displacement
    For node No.2: X displacement, Y displacement, and Z displacement
---------------------------------------------------------
```

The filename should be set in the following way:

```
---------------------------------------------------------
    0 dp0/dp
    1 dp1/dp
    2 dp2/dp
---------------------------------------------------------
  (Part number  Name of the file with initial displacements)  x  number of Parts
---------------------------------------------------------
```

The number of Gauss points should be selected from:
Linear tetrahedral element: 1
Quadratic tetrahedral element: 4
Linear hexahedral element: 2
Quadratic hexahedral element: 3

The flag for workload distribution should be selected from:
1: Static workload distribution
2: Dynamic workload distribution

The width of time gap is set equal for all steps of analysis in the current version of the program.

3.6. Input Setup File 2 (AdvImpact_init_contact.txt)

The Input Setup File 2 contains information on the interval between subsequent global contact searches in term of number of time steps.

(Example)

```
---------------------------------------------------------
50
---------------------------------------------------------
```

Number of time steps for interval between global contact searches

```
---------------------------------------------------------
```

If the range of deformations and displacements are larger than the size of an element, the value of global contact search interval should be smaller.

Note: this setup file is necessary ONLY for contact analyses

3.7. Output Setup File (*AdvImpact_init_outputflag.txt*)

The Output Setup File contains the setup flags for output.

(Example)

```
-------------------------------------------------------
0 0
1 0
2 0
3 0
4 0
5 7
6 0
.  .  .  .  .  .  .  .
4995 0
4996 0
4997 0
4998 0
4999 7
-------------------------------------------------------
```

(Time step Setup flag for output) x number of time steps

```
-------------------------------------------------------
```

The setup flags should be selected from the following table and set for EVERY step individually.

Setup flags for output

Flag	Displacement	Strain	Stress
0	Not saved to the disk	Not saved to the disk	Not saved to the disk
1	Saved to the disk	Not saved to the disk	Not saved to the disk
2	Not saved to the disk	Saved to the disk	Not saved to the disk
3	Not saved to the disk	Not saved to the disk	Saved to the disk
4	Saved to the disk	Saved to the disk	Not saved to the disk
5	Saved to the disk	Not saved to the disk	Saved to the disk
6	Not saved to the disk	Saved to the disk	Saved to the disk
7	Saved to the disk	Saved to the disk	Saved to the disk

3.8. Environment Setup File 1 (AdvImpact_init_config.txt)

The Environment Setup File 1 contains the information on

- The number of *Parts*, and
- The number of *Analyzers*.

(Example)

```
-------------------------------------------------------
3
0 8
1 8
2 8
-------------------------------------------------------
```

Number of *Parts*
(*Part* number Number of *Analyzers*) x number of *Parts*

```
-------------------------------------------------------
```

The number of Analyzers should be set for each Part. It will define the number of processes in analysis. Usually, one *Analyzer* is set for one processor.

For above-mentioned example, the total number of processes is 28.

GrandController: 1 (usually one)
Controller: 3 (same as the number of *Parts*)
Analyzer: 24 (8+8+8)

3.9. *Environment Setup File 2 (AdvImpact_init_loadbalace.txt)*

The Environment Setup File 2 contains the information on the *Analyzer* that treats *Subdomain* (should be set for each *Subdomain*)

(Example)

```
------------------------------------------------------
0 0 8
0 1 8
0 2 8
.  .  .  .  .
2 5 8
2 6 8
2 7 8
0 0 0
0 0 1
0 0 2
0 0 3
0 0 4
0 0 5
0 0 6
0 0 7
0 1 8
0 1 9
.  .  .  .  .
2 7 61
2 7 62
2 7 63
------------------------------------------------------
```

(*Part*'s number   *Analyzer*'s number   Number of *Subdomains*)   x   number of *Analyzers*   x   number of *Parts*

(*Part*'s number   *Analyzer*'s number   *Subdomain*'s number)   x   number of *Subdomains*   x   number of *Parts*

The first setup line defines the number of *Subdomains* for each *Analyzer* and the second setup line defines the *Subdomain*'s number. In the example described above, 8 *Analyzers* defined for 1 *Part* treat 8 *Subdomains*. To achieve the best performance in parallel processing environments, the number of Analyzer processes for each Part should be set by considering the performance of each processor. If performances differ significantly among processors, the codes advimpact-g, advimpact-c, and advimpact-a should be used instead of the combined code advimpact-t.

<u>Note: The Environment Setup File 2 is used only if the load is distributed statically. In the case of dynamic load distribution, this file is unnecessary.</u>

3.10. Processors Setup File (pgfile)

The Processors Setup File is used by MPI (see the MPI User's Manual for detais).

(Example)

```
-------------------------------------------------------
hoge001 advimpact-g 0
hoge002 advimpact-c 1
hoge003 advimpact-c 1
hoge004 advimpact-c 1
hoge005 advimpact-c 1
hoge006 advimpact-a 1
.  .  .  .  .  .  .  .  .
hoge032 advimpact-a 1
-------------------------------------------------------
```

(Hostname Name of executable file Number) x number of CPUs

```
-------------------------------------------------------
```

3.11. Visualization Setup File 1 (in_visual_node.txt)

(Example)

```
------------------------------------------------------
3
0 res0/res
1 res1/res
2 res2/res
node_res.dat
1
1
5
0 0 0 5
1 0 5 375
2 1 32 53
3 1 60 876
4 2 23 572
3
0 0 0 2
1 1 0 27
2 2 3 62
------------------------------------------------------
```

Number of *Parts*
(Part's number Filename of the analysis data) x number of *Parts*
Filename for output of time-dependent data at the nodes/elements specified below
Output flag for displacements of nodes
Output flag for stresses at integration points in elements
Number of nodes for output of dynamic response
(Number *Part*'s number *Subdomain*'s number Node's number) x number of nodes for output of displacements
Number of elements for output of stresses
(Number *Part*'s number *Subdomain*'s number Element's number) x number of elemens for output of stresses

----

Output flag : 0: No output
1: Output

The output file contains following data:
(Example)

```
--------------------
5 3
0 0 5
0 0.00 0.00 0.00
1 0.00 0.00 0.00
2 0.00 0.00 0.00
0 5 375
0 0.00 0.00 0.00
1 0.00 0.00 0.00
```

```
2 0.00 0.00 0.00
 ・
 ・
--------------------
```

Number of nodes for output　　Number of times for data output
Part's number　*Subdomain*'s number　Node's number
(Number X displacement　Y displacement　Z displacement) x　number of times for data output

```
 ・
 ・
--------------------
3 2
0 0 2 8
0 0.00 ・・・ (48 real valued data) ・・・0.00
1 0.00 ・・・ (48 real valued data) ・・・0.00
1 0 27 8
0 0.00 ・・・ (48 real valued data) ・・・0.00
1 0.00 ・・・ (48 real valued data) ・・・0.00
 ・
 ・
--------------------
```

Number of elements for output　　Number of times for data output
Part's number　*Subdomain*'s number　Element's number　Number of integration points
Number ((xx-stress yy-stress zz-stress xy-stress yz-stress zx-stress) x number of integration points) x number of times for data output times

```
 ・
 ・
--------------------
```

3.12. Visualization Setup File 2 (in_visual_disp.txt)

The Visualization Setup File 2 contains the settings for `advimpact-vdisp`. The time-dependent distributions of node displacements can be visualized (for necessary time steps) using MicroAVS.

(Example)

```
-------------------------------------------------------
2
avs.inp
3
advhddm_in_0.adv
advhddm_in_1.adv
advhddm_in_2.adv
result0/res
result1/res
result2/res
-------------------------------------------------------
```

Order (from the beginning) of the time step for sampling
Name of output file (MicroAVS input file)
Number of *Parts*
(Name of mesh data file) x number of *Parts*
(Name of results output data file) x number of *Parts*

```
-------------------------------------------------------
```

The settings of time steps should be done in accordance with the following rule. For example, if the time-dependent output was done for steps 100, 200, and 300, to view the displacement distributions at time step 200, set the 'Order of the time step for sampling' to 2.

Appendix

A. Explicit Dynamics Method

The Explicit Dynamics method is implemented in the basic algorithm of the present code. The simple explanations will be done here for reference.

The spatial discretization of the equation of motion in structural dynamics can be formulated as

$$[M]\{\ddot{u}\}+[C]\{\dot{u}\}+[K]\{u\}=\{f\} \tag{1}$$

where $\{\ddot{u}\},\{\dot{u}\},\{u\}$ are the acceleration, the velocity, and the displacement vectors, correspondingly; $[M]$ is the mass matrix; $[C]$ is the damping matrix; $[K]$ is the stiffness matrix; and $\{f\}$ is the load vector. The time discretization of Eq. 1 done by the central finite difference scheme will results in

$$\begin{aligned}\left(\frac{1}{(\Delta t)^2}[M]+\frac{1}{2\Delta t}[C]\right)\{u\}_{n+1}=\{f\}_n-\left([K]-\frac{2}{(\Delta t)^2}[M]\right)\{u\}_n \\ -\left(\frac{1}{(\Delta t)^2}[M]+\frac{1}{2\Delta t}[C]\right)\{u\}_{n-1}\end{aligned} \tag{2}$$

Here, $\{u\}_{n+1},\{u\}_n,\{u\}_{n-1}$ are the displacements at the *(n+1)*th , *n*th, and *(n-1)*th steps correspondingly, and Δt is the time step width. If $[M]$and $[C]$ are the diagonal matrixes, the matrix inversion is not necessary. Such method is called the Explicit Dynamics method. The Explicit Dynamics method is conditionally stable. The limitations are imposed on the width of time gap for Courant conditions.

If the Rayleigh damping is considered, the damping matrix $[C]$ will be defined as

$$[C]=c_K[K]+c_M[M] \tag{3}$$

If we use the concentrated mass matrix and set $c_K=0$, it is possible to introduce the damping due to viscosity. In the present code, c_M can be treated as a value of material properties (viscous material damping coefficient).

B. Figures of Basic Elements

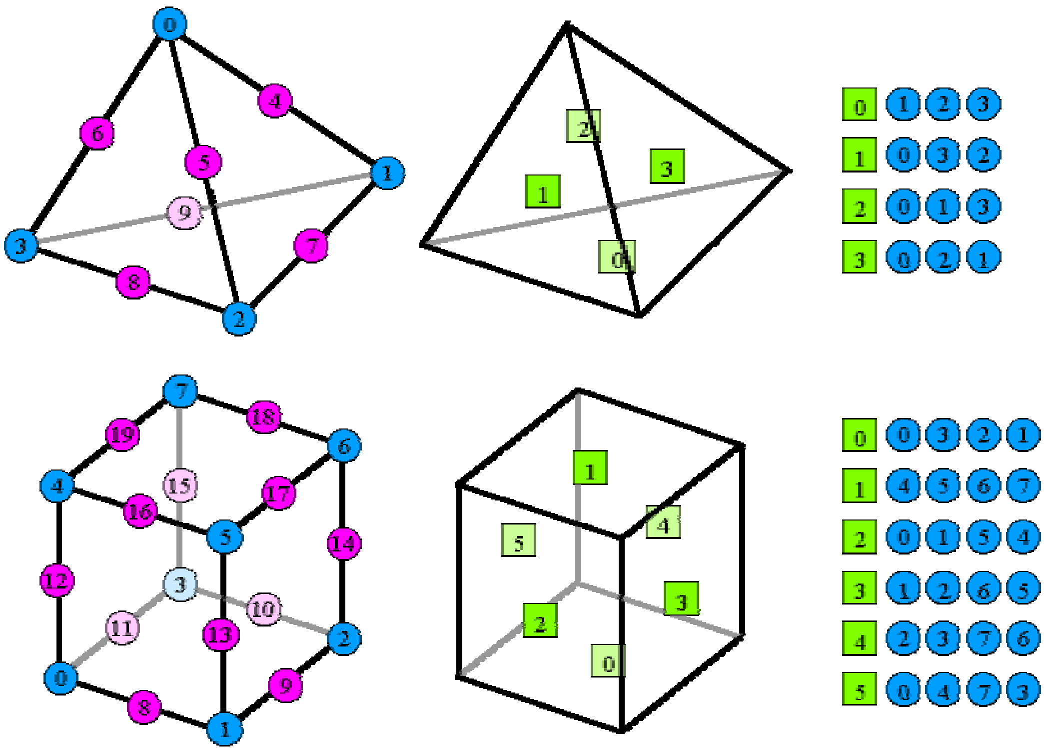


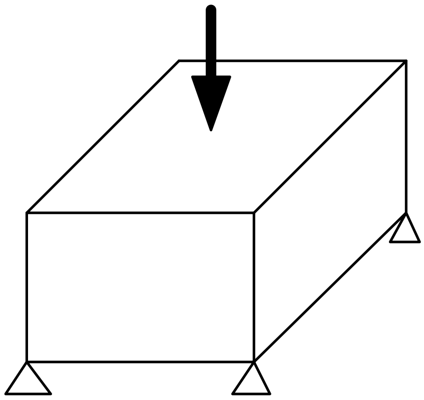
Fig. Basic Elements (the node numbers are shown in circles, and the surface numbers are shown in squares)

C. Sample Data

Two sets of sample data are supplied with the current code.

`sample/data01/`	Dynamic-elastic analysis of wave propagation without contact
`sample/data02/`	Dynamic-elastic analysis of wave propagation with contact

(1). `Data01`



Model:

Dimensions: 16mm x 16mm x 8mm
Material: Steel, elastic, isotropic without viscous damping for all elements
Fixation: 4 corners of the lower surface are fixed
Load: distributed mechanical load, 2mm x 2mm, 2MHz, 1cycle, applied in the middle of the upper surface

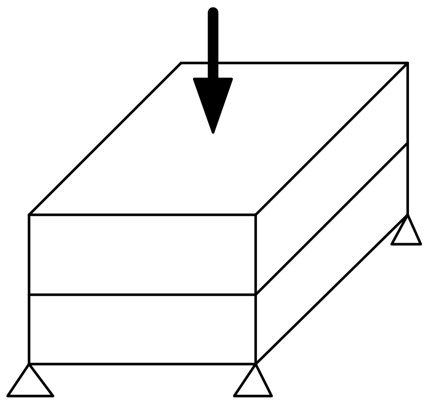
Element size: 1mm x 1mm x 1mm
Number of elements: 2048 linear hexahedral elements
Number of nodes: 2601

Sample files:

`mesh01.dat`	Mesh data
`fdcurve01.dat`	Displacement and load curves data
`AdvImpact_init_files.txt`	Input Configuration File 1
`AdvImpact_init_outputflag.txt`	Output Configuration File
`in_config1.txt`	Data Configuration File 1
`in_config2.txt`	Data Configuration File 2

Each setup file should be adjusted to the system operational environments.

(2) `Data02`



Model:

Dimensions: 16mm x 16mm x 8mm
Material: steel, elastic, isotropic, without viscous damping for all elements
Fixation: 4 corners of the lower surface are fixed
Load: distributed mechanical load, 2mm x 2mm, 2MHz, 1cycle, applied in the middle of the upper surface
Contact: one horizontal surface in the middle part of the model, 16mm x 16mm with 0mm gap
Friction between contacted surfaces: none

Element size: 1mm x 1mm x 1mm
Number of elements: 2048 linear hexahedral elements
Number of nodes: 2890

Sample files:

`mesh01.dat`	Mesh data
`fdcurve01.dat`	Displacement and load curves data
`AdvImpact_init_files.txt`	Input Configuration File 1
`AdvImpact_init_contact.txt`	Input Configuration File 2
`AdvImpact_init_outputflag.txt`	Output Configuration File
`in_config1.txt`	Data Configuration File 1
`in_config2.txt`	Data Configuration File 2

Each setup file should be adjusted to the system operational environments.

D. Limitations

(1). Supported elements

Use linear hexahedral elements instead of linear tetrahedral elements in the contact analysis.

(2). System incompatibility

Since, the data are treated in binary formats, some errors after transfer from one operational environment to another can occur due to difference in binary formats and compiler options (for example, Sun SPRAC uses a big endian format, whereas Intel Pentium uses a little endian format).

E. Troubleshooting

Symptom：Running advimpact results in error messages. The following error message is an example.

```
GrandController mytid=0 START time=1073281650
part=0 mytid=1 Controller START time=1073281650
p4_17086:  p4_error: net_recv read:  probable EOF on socket: 1
part=2 mytid=3 Controller START time=1073281650
part=1 mytid=2 Controller START time=1073281650
p1_17071:  p4_error: interrupt SIGSEGV: 11
ADVENTURE/bin$ bm_list_17067: (3.357025) wakeup_slave: unable to interrupt
slave 0 pid 17066
```

Solution：

（1） Please create directories for output data before running advimpact.
For example, if you are going to use following AdvImpact_init_files.txt,

```
------------------------------------------------------
0 advhddm_in_0.adv
1 advhddm_in_1.adv
2 advhddm_in_2.adv
0 advimpact_in_0.adv
1 advimpact_in_1.adv
2 advimpact_in_2.adv
advimpact_gdlist.adv
0 result0/res
1 result1/res
2 result2/res
0
3
2
1
2
5000
1.0e-8
------------------------------------------------------
```

you must prepare 3 subdirectories,

`result0  result1  result2`

in the current directory BEFORE execution.