ADVENTURE on Windows

A set of single-CPU ADVENTURE modules running on Microsoft Windows

Version: Beta 0.12

User manual

March 10, 2005

ADVENTURE Project

Table of Contents

1. Outline

This program is a collection of ADVENTURE modules including ADVENTURE_Solid ported to Microsoft Windows (referred to henceforth as Windows) for use on a single CPU. The program has the following features:

(1) Instead of entering all commands, the program may be used through a graphical user interface (GUI) similar to that of ADVENTURE_iAgent.
(2) Based on the user's intent, a plan of procedure is generated. The program guides the user through the whole process from surface mesh generation to result visualization.
(3) The current version supports a linear elastic stress analysis function. No non-linear analysis or thermal stress analysis are supported.
(4) Since the operating conditions of the machine cannot be displayed, use the Windows Task Manager.

The user interface offered by this program is called Agent which is an unique interface among existing software applications. Agent is a software application that stands between ADVENTURE and the user to make use of ADVENTURE more comfortable.

1.1 Module and Function to Support

ADVENTURE modules and the names of specific tools supported by this program are shown in Table 1-1.

Table 1-1. Supporting modules

Function	Module name	Tool name	Version
Surface patch generation	TriPatch Ver.1.8	maskMelon.exe	1.2
		ckpatch.exe	1.2
		mcpach..exe	1.00
		mrpache.exe	1.01
Mesh generation	TetMesh Ver.0.9	advtmesh9p.exe	0.9
		advtmesh9m.exe	0.9
		advtmesh9s.exe	0.9
Boundary condition sticking	Bctool Ver. 1.02	faceOfMesh.exe	1.02
		makepch.exe	1.02
		makefem.exe	1.02
Domain decomposition	Metis Ver.1.0	adventure_metis.exe	1.0
Stress analysis	Solid Ver.1.1	advsolid-s.exe	1.1
		hddmmrg.exe	1.1

The following functions are supported.

(1) A linear elastic stress analysis function is provided by ADVENTURE_Solid. (Thermal stress analysis functions are not supported.)
(2) Allowable input models are IGES file, surface patches in ADVENTURE format (*.pch) and tetrahedral mesh in ADVENTURE format (*.msh) which satisfy specific conditions.
(3) Standalone use of the GUI tool to apply boundary conditions is possible. (This boundary condition tool is newly developed for Windows and different from the bcGUI for Linux)

(4) Standalone use of GUI tool for result visualization. (This result visualization tool is newly developed for Windows and different from Visual for Linux)
(5) Although the Agent functionality is fairly similar to iAgent for Linux, there are a few differences.

2. Operation Condition

This program operates in the following environment.

(1) OS
Windows 2000 Professional, Windows XP Professional (some abnormality in the 3D display function on some laptops has been identified.)

(2) Compiler
A compiler is not required since the application is provided precompiled.

3. Installation and Setup

After logging in to Windows as an authorized user, execute the installer AdvWn012.exe. The window shown in Fig. 3-1 will appear and change as in Fig. 3-2 automatically.

Fig. 3-1 Initial stage of installation

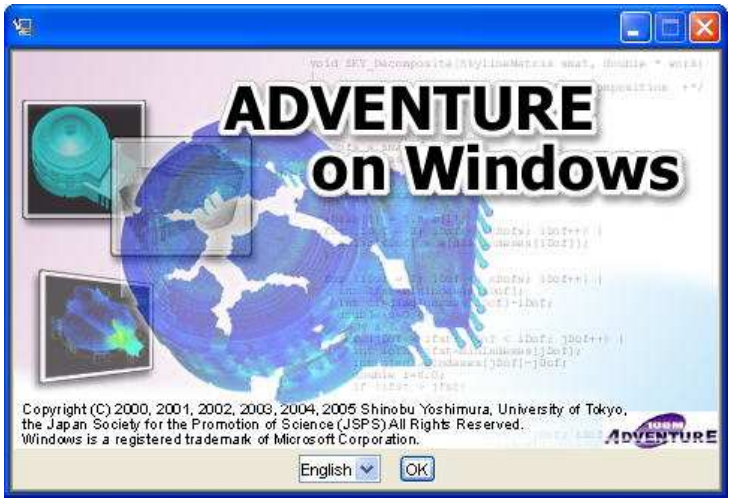


Fig. 3-2 Window for language selection at installation

Click “OK” on the screen shown in Fig. 3-2 to produce the screen shown in Fig. 3-3.

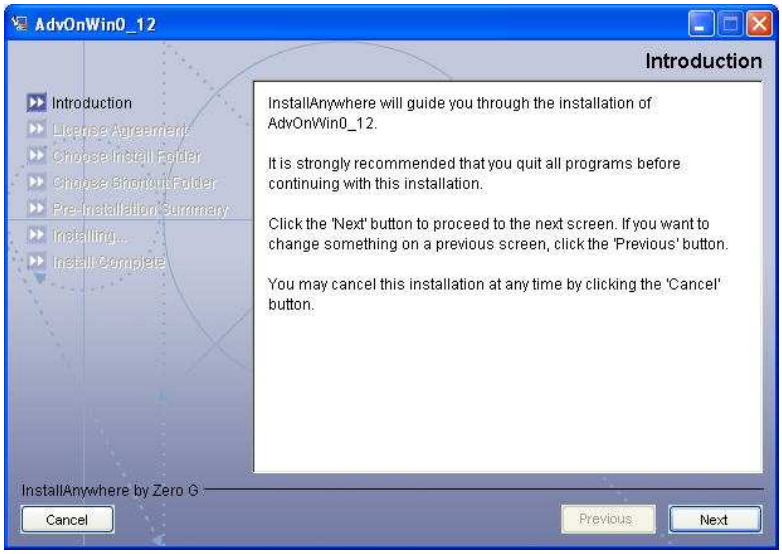


Fig. 3-3 Window at installation start-up

Click “Next” and the window in Fig. 3-4 will appear. You will be asked whether you accept the terms of the License agreement. To accept, select the “I accept the terms of the License Agreement” radio button and click “Next”.

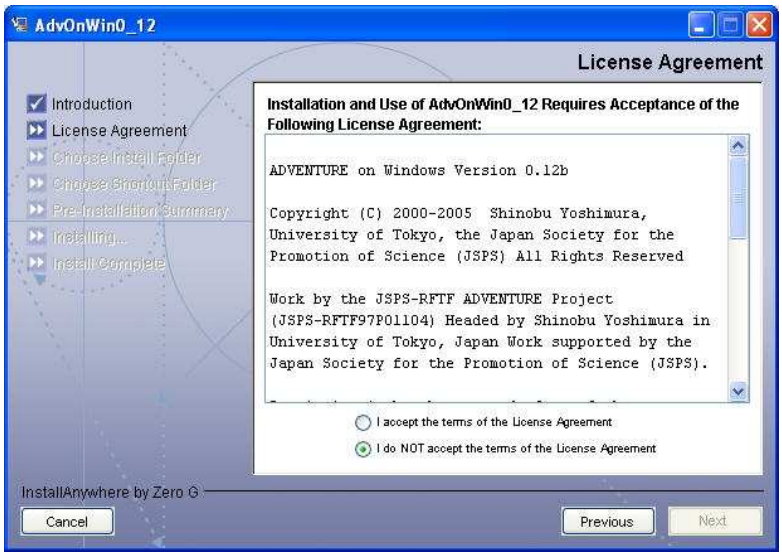


Fig. 3-4 License Agreement

The screen shown in Fig. 3-5 will appear requesting you to select a folder for installation. The C:\Program Files\AdvOnWin0_12 folder is set as a default. If you wish to specify a different folder, click “Choose…”

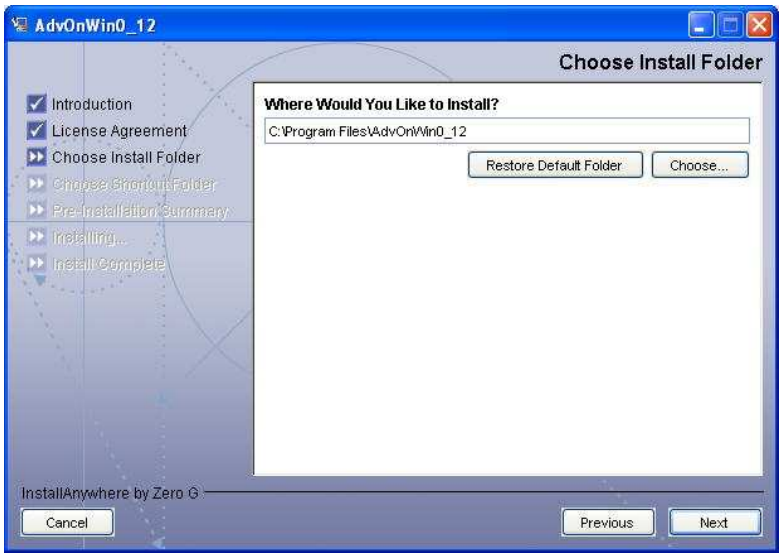


Fig. 3-5 Choosing an installation folder

Fig. 3-6 will appear and select a folder for installation.

You may return to the default installation folder even after selection of an alternative by clicking “Restore Default Folder”. After selecting a folder, click “Next”.

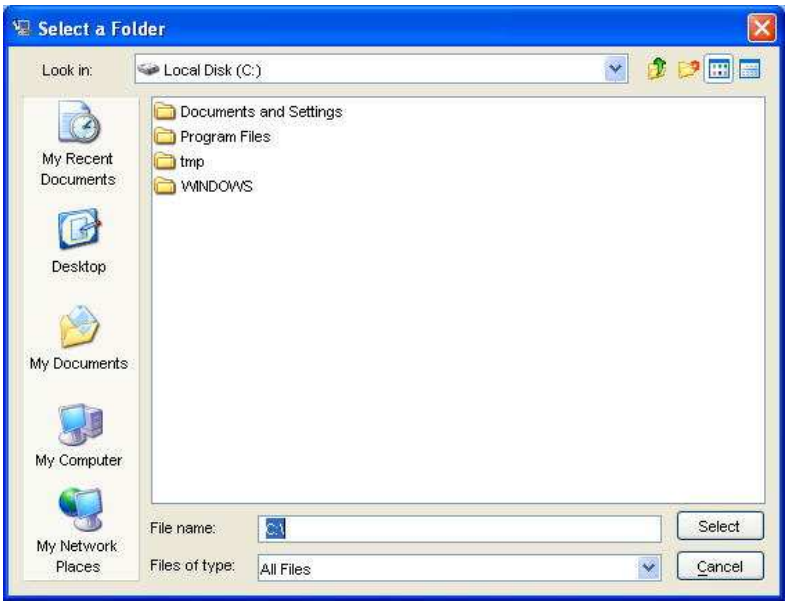


Fig. 3-6 Selecting a folder

At the window shown in Fig. 3-7, select a location for the shortcut for this program. Bear in mind that you can choose only one folder. If you require a shortcut on the Desktop in addition to the Start menu, select one here and create the other manually. After selecting a location, click “Next”.

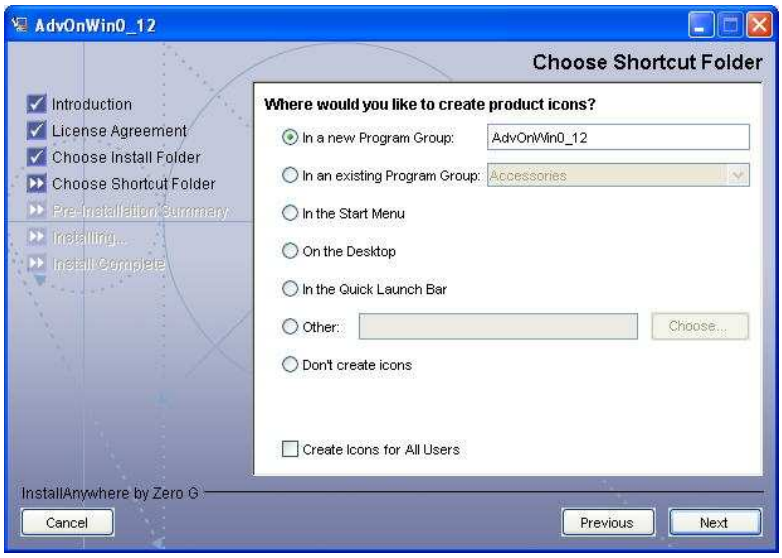


Fig. 3-7 Choosing a shortcut folder

The window shown in Fig 3-8 shows the installation settings you have selected and the available disk space. After confirming that there is no problem, click “Next”.

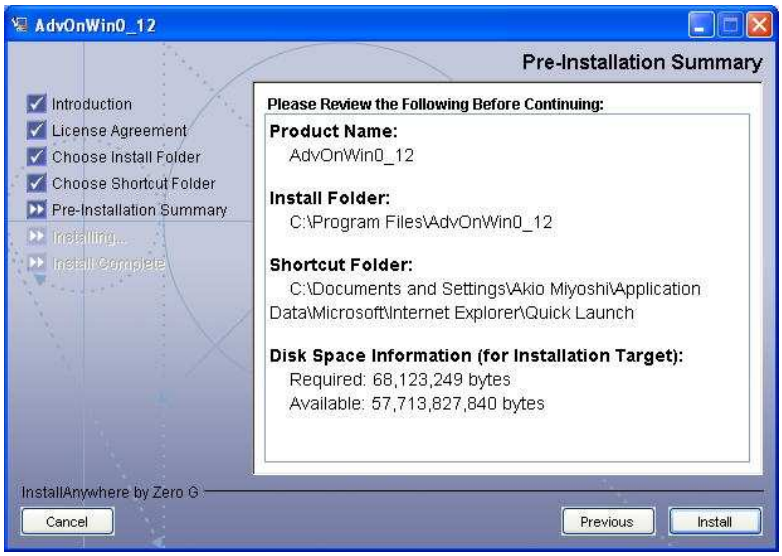


Fig. 3-8 Pre-installation summary

The window shown at Fig. 3-9 will appear and installation will begin.

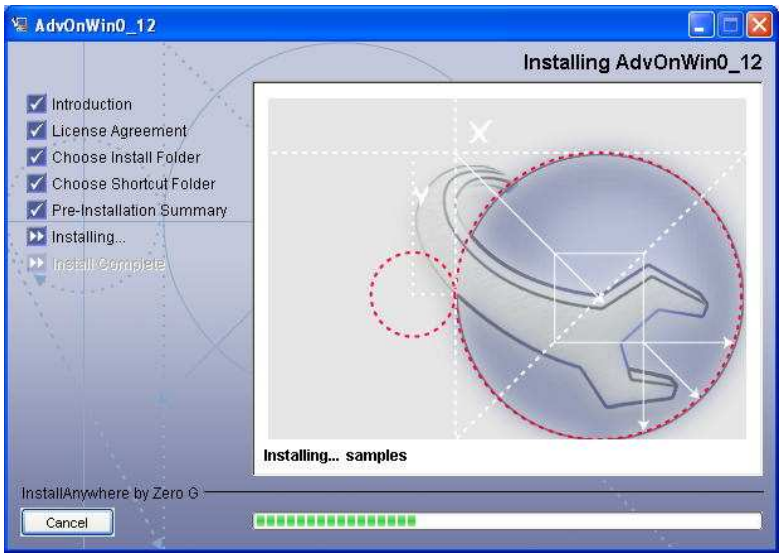


Fig. 3-9 Installing AdvOnWin0_12

After a period of installation activity, the window shown in Fig. 3-10 will appear automatically. Click "Done" to complete installation.

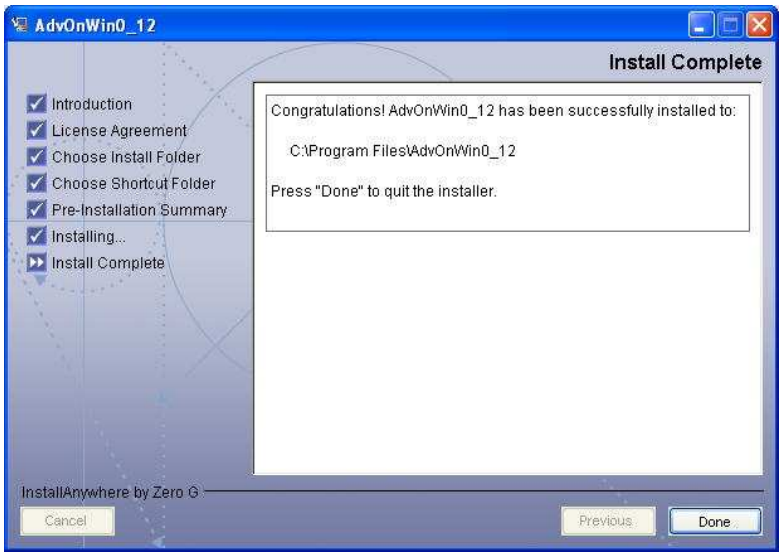


Fig. 3-10 Completion of installation

4. Relation among Agent, User and the ADVENTURE System

Since most ADVENTURE modules are provided as command line tools, the user is required to know how to use each command and its operating procedures as well as comprehend the simulation problem to be analyzed.

Agent creates an analysis plan by interpreting the intention of the user. It follows this plan to call each module of ADVENTURE and shows the result of the command to the user. It aims to reduce complexity from user's operation and realize smooth and easy operation. In addition, it maintains and controls an analysis plan, all operations of the user and parameter values, and analysis results as an analysis case. An

analysis case can be saved and opened.

The relationship between the user and Agent, and Agent and the ADVENTURE system can be seen in the following Fig. 4-1.

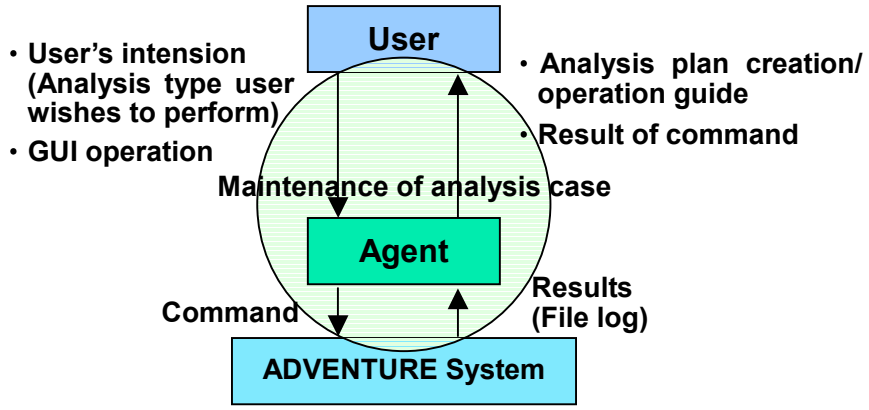


Fig. 4-1 Relationship of the user, Agent and the ADVENTURE system

4.1 Services Which Agent Provide/ Does Not Provide

Specific services which Agent provides are as follows.

- Creation and indication of analysis procedure suitable for analysis type
- Indication of input interface for each parameter.
- Execution of each module in ADVENTURE system.
- Indication of operating procedure of BCtool.
- Committing the analysis plan, all operations of the use and the results to an analysis case.
- Saving analysis plan to a file and reading it from the file.
- Interruption and restart of analysis cases at an arbitrary stage.
- Bilingual Japanese and English operation.

The following functions are not provided.

- Interruption and restart of each ADVENTURE module for mesh generation or solver execution, etc.
- Automatic restructuring of analysis plan after an out-of-plan operation.
- Cancellation of operation.
- Automatic restart of analysis at the stage where user's last operation was suspended.
- Display of operating condition (CPU utilization/ memory utilization) of the user's machine.

4.2 Screen Configuration

The following additional step is required if you wish to use the English version. Edit the file AdvOnWin0_12.lax in the installation folder, commenting out the line

lax.command.line.args=$CMD_LINE_ARGUMENTS$

with a #, as below

#lax.command.line.args=$CMD_LINE_ARGUMENTS$

and inserting a new line

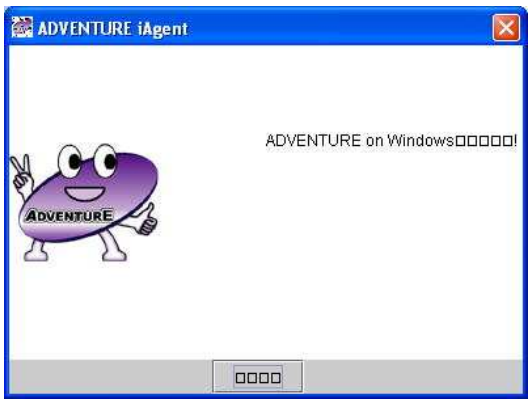
lax.command.line.args=-e

then start the application.

After launching, a flash window should be shown for a moment before changed to the greeting window as shown in Fig. 4.2-1.

Fig. 4.2-1 English version of the greeting screen

Note that if the lax.command.line.args line is not modified as indicated above, the greeting screen will come up in garbled language as shown in Fig. 4.2-2.



4.2-2 Garbled greeting screen

Click “Start" and various windows for Agent are shown. These screens are shown in Fig. 4.2-3

a) Menu window

This window is to invoke all analysis operations. At the lower part of the window, the current analysis type is shown. Please bear in mind that a default analysis type is set automatically at start.

For menu items and their instructions for their use, please consult chapter 5 through 7.

b) Message window

Advice from Agent is shown. (Operation details and information on current operation)

c) Operation flow

The current operation plan is shown. By clicking the button to the left of each item, an outline of the operation for the item is shown in the message window.

Besides the above windows, information on ADVENTURE and errors are indicated in windows in the

middle of display as often as necessary.

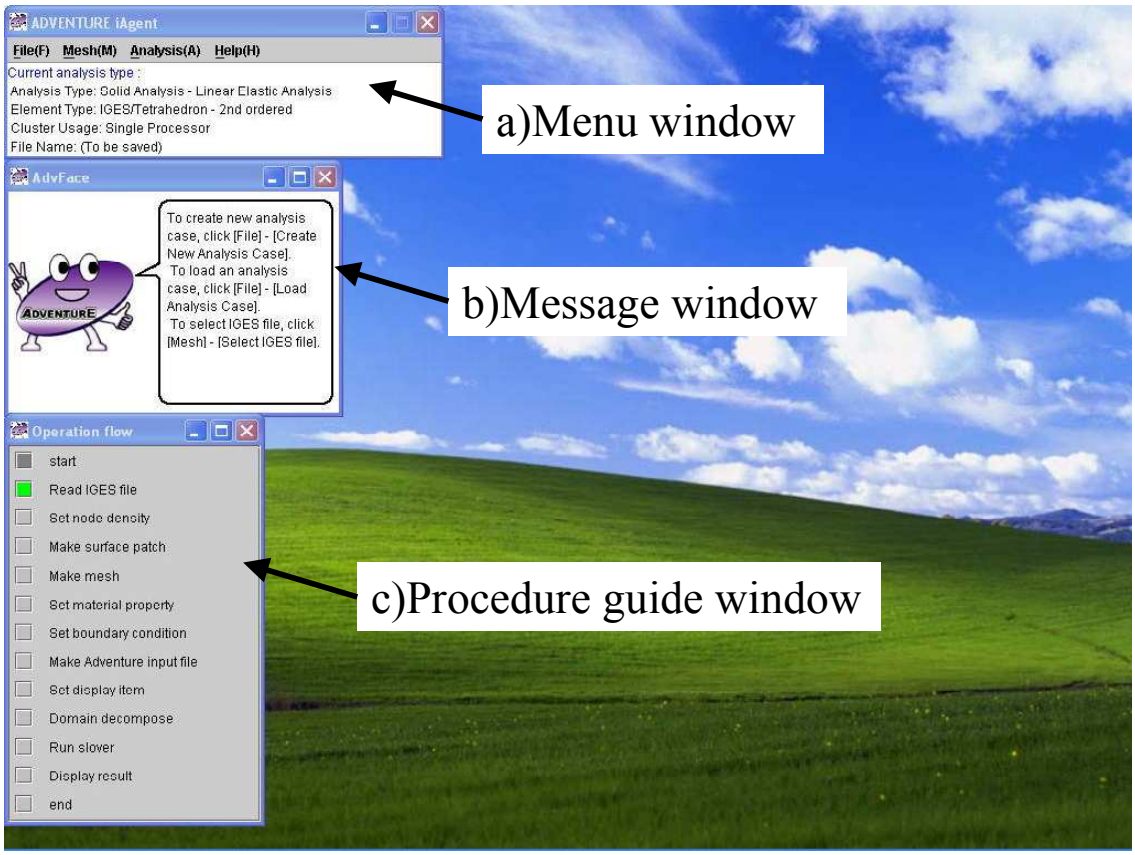


Fig. 4.2-3 Screens for operation

4.3 Operation Order

An operation indicated in the Operation flow mostly requires the results of preceding operations. Therefore, if one of these prerequisite operations is skipped, the window shown in Fig. 4.3-1 may appear.

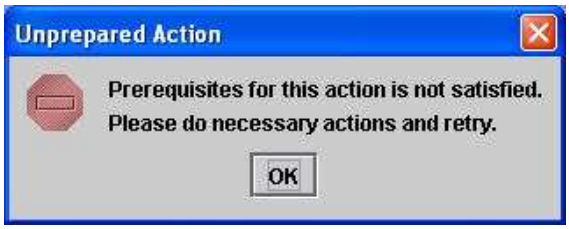


Fig. 4.3-1 Improper operation order

If this window displays, check whether there are any skipped operations by referring to the Operation flow.

4.4 Prevention of Unnecessary Operations

Each operation of analysis plan indicated in the Operation flow window is a necessary and sufficient operation in accordance with the current analysis plan. Therefore, if an out-of-plan operation is executed, the window shown in Fig. 4.4-1 may appear.

Fig. 4.4-1 Operation out of plan

If this window appears it is because you have performed an unnecessary operation. If the operation was in fact necessary, your analysis case setting was improper. Create a new analysis case and select a proper analysis plan.

For more information about analysis case, refer to chapter 5.

4.5 User Control

User operation executed via Agent is stored as user-specific operation history.

In future version, there is possibility to provide services depending on usage situations based on stored user information.

5. What Is an Analysis Case?

Agent manages the whole analysis session as an instance of “analysis case”. This is what is generally known as a “project”.

It controls the following items as an analysis case,

- Analysis type and analysis plan which is derived from analysis type.
- Operation executed by user.
- Various values input by user.
- File name and the file itself which is automatically created for each module of ADVENTURE (all files are included).

Agent includes functionality to save an analysis case and then to restore later the saved state by reading the saved analysis case. At that time, files for each module of ADVENTURE are automatically saved/ restored.

Using this functionality, the user may suspend and restart analysis operations with ease. However, please be careful it doesn’t mean the interruption while solver, etc. is running.

To save analysis case, select “File(F)” > “Save Analysis Case (S)”. The file selection window will appear. Select a folder in which to save and enter a file name. If the file name suffix is not “.iag”, it is added automatically as the file name extension.

Files which were created automatically for ADVENTURE modules are saved in a folder called “(file name).files” which is created in the same folder..

For example, if you save analysis case to My Documents\data folder as analysis.jag, a folder named My Documents\data\analysis.files will be created automatically and the file for each module will be copied into it.

For reading an analysis case, select “File(F)”> “Open Analysis Case(O), and .iag from file selection dialogue. Then saved plan, the set values and the files will automatically be restored and analysis can be restarted.

To write over an existing file, select “File(F)” > “Save Analysis Case(S)”. To save as a different name,

select “File(F)” > “Save Analysis Case As(A)”.

The .iag file format is the output of the workingMemory.AnalysisCase class using the serialization mechanism of Java. The file format not intended for use by a different language.

6. Operation Instruction for Stress Analysis

First let’s look at the system of units in ADVENTURE. ADVENTURE's design does not depend on any system of units. Thus ADVENTURE can generate analysis results in any system of units. Care should be taken however to ensure consistency among units.

Immediately after Agent is started, a new analysis case and analysis plan is automatically prepared in accordance with the standard analysis type which the administrator or user has set beforehand. For the procedure to set a standard analysis type, refer to Appendix. 1.

The analysis settings currently used is indicated under the menu in the main window (Fig. 6-1). If you wish to change to an analysis type different from the current one, create a new analysis case and let Agent know what kind of problem you want to analyze.

For clarity, we will briefly explain the operation details using the analysis case “elShape.iag” which may be found in the samples folder under the installation folder. The instruction for general operation procedure will be addressed later.

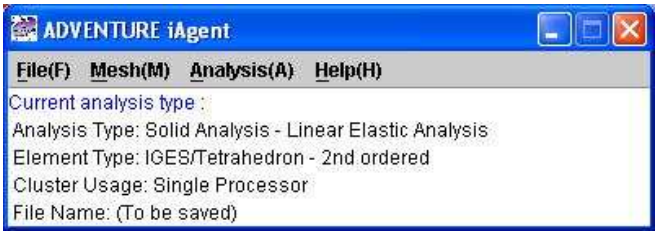


Fig. 6-1 Analysis settings currently selected

6.1 Reading elShape.iag

elShape.iag is an analysis case on an L-shape part which is one of the ADVENTURE’s official samples. Since information up to the results is included in this analysis case, it is possible to display the analysis result directly. No errors occur if you execute any other operation at random. So, it is easier to operate for a first time user of ADVENTURE.

To read the elShape.jag analysis case, select “File(F)” in the menu window > “Open Analysis Case(O)”. The installation folder will appear as shown in Fig. 6.1-1. Select the “samples” folder and click “Open” or doubleclick the “samples” folder(Fig. 6.1-2). Then select the elShape.jag file and click “Open” or doubleclick the elShape.iag. The window will change as shown in Fig. 6.1-3. Since the message window says: “Now you have finished all tasks…”, the whole processing is complete on this analysis case. In the Operation flow window, the “end” stage at the bottom is green, indicating where you are.

This elShape analysis case is to generate second order tetrahedron mesh from the initial input of IGES geometry file and execute a linear elastic stress analysis. Now let’s look at the analysis result. Select “Analysis(A)” > “View Result(V)” from the menu window.

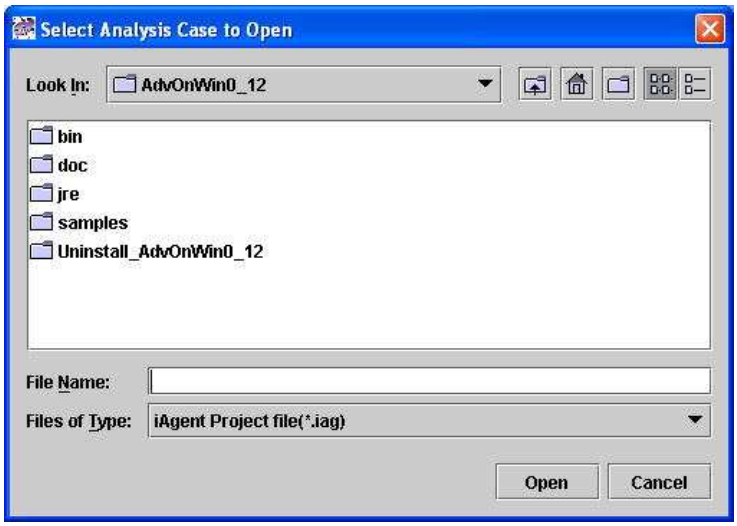


Fig. 6.1-1 Window for reading analysis case

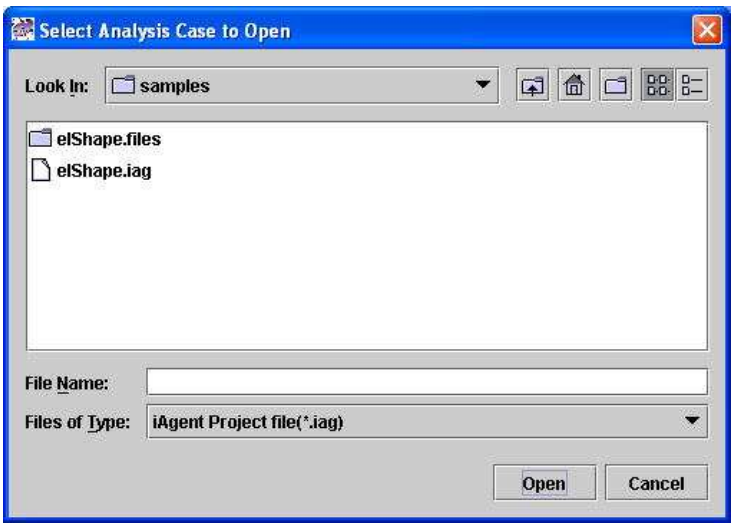


Fig. 6.1-2 In samples folder

Fig. 6.1-4 will appear and click “OK” in the window. The Visualtool window appears as shown in Fig. 6.1-5 and x-direction displacement in the analysis result is displayed as color contour map. It is also deformed at the default magnification factor of 1.0. Maximum/ minimum value of displacements can also be confirmed. Details of operation procedure will be explained later. Here we return to the beginning of analysis procedure. Close the result display by selecting “File(F)” > “Exit(X)” in the Visualtool window.

The Operation flow is as shown in Fig. 6. 1-6. Click the gray button on the left of “Read IGES file” at the top of the flow. The button changes to green and the message window says “To select IGES file…”. In this way, instruction at any step can be confirmed. To start to read IGES file, select “Mesh(M)” > “Select IGES file(I)” in the menu window and the window shown in Fig. 6.1-7 will be appear. Confirm that a folder: advOnWin, is created in My documents automatically. Error.log is saved in it automatically. Move from My documents to installation folder. If you go to samples folder, it will be displayed as Fig. 6.1-8, then select elShape.igs and click “Open”.

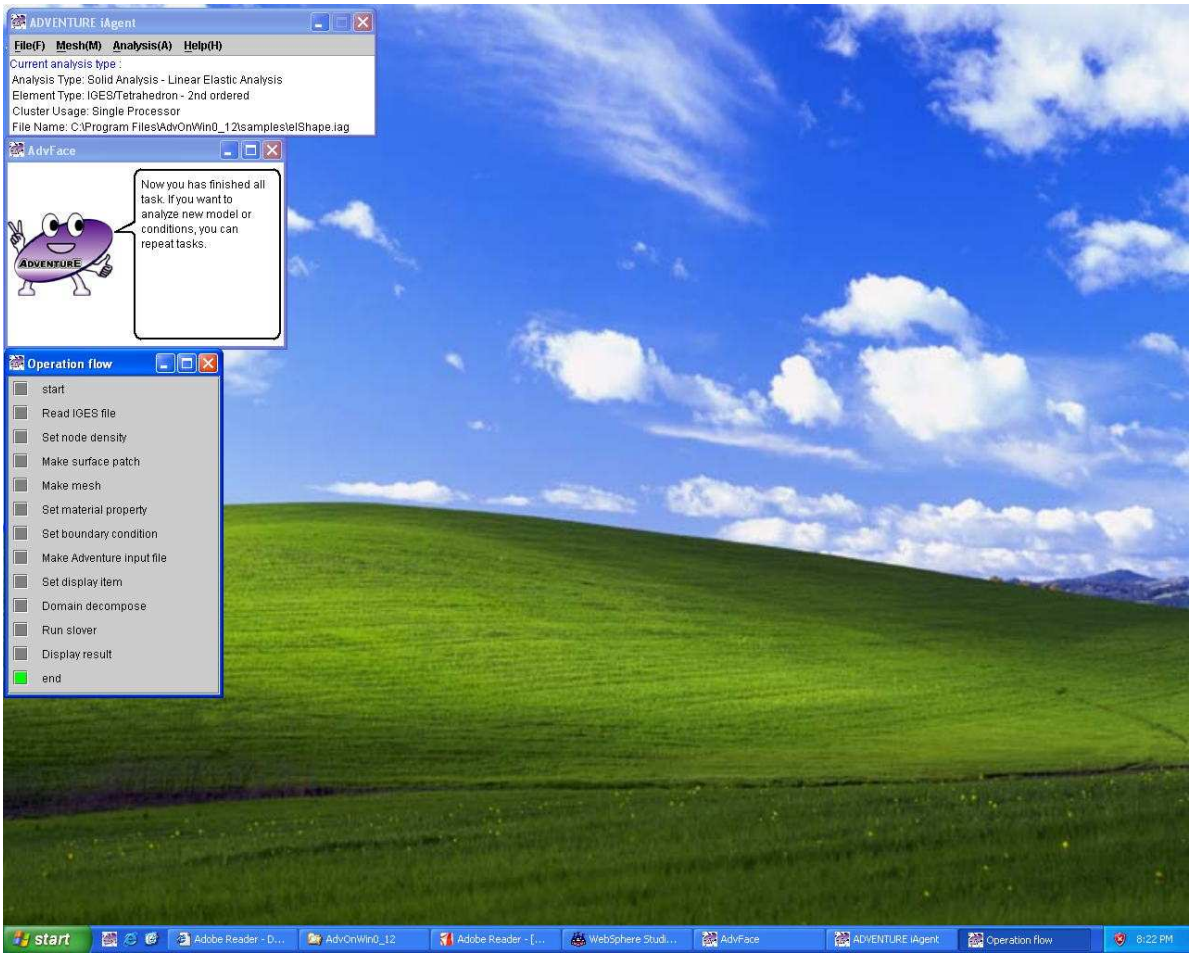


Fig. 6.1-3 Immediately after opening elShape analysis case

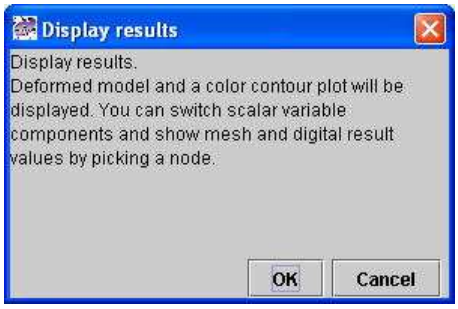


Fig. 6.1-4 Dialog after selecting the result display menu item

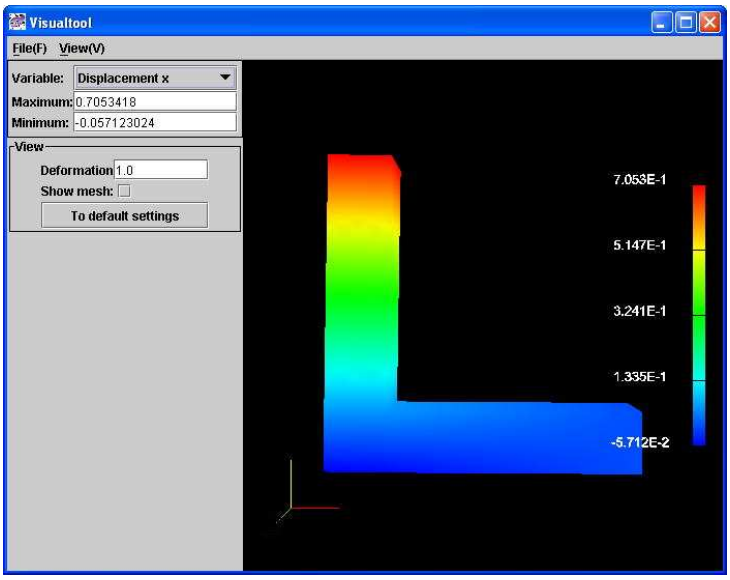


Fig. 6.1-5 Display of analysis result

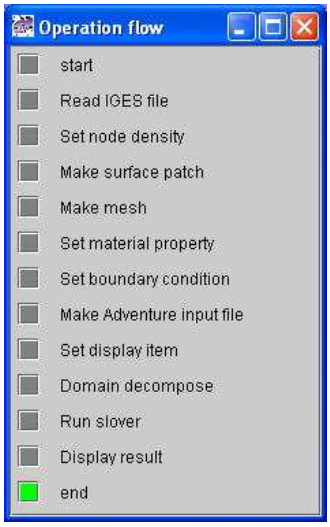


Fig. 6.1-6 After completion of result display

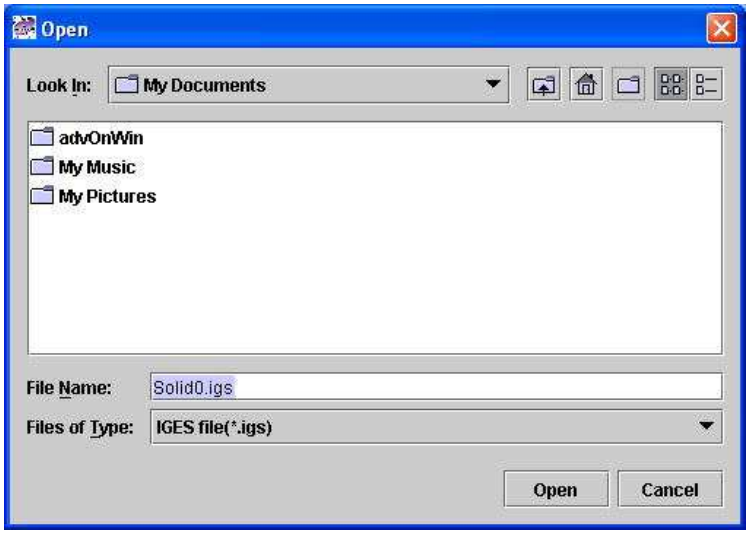


Fig. 6.1-7 File selection dialog for an IGES file

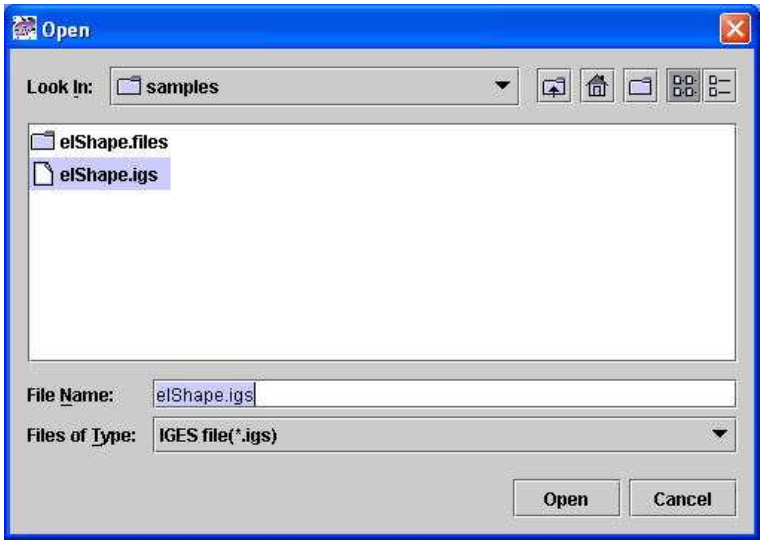


Fig. 6.1-8 Selecting elShap.igs

6.2 Generation of Patch and Mesh

Next is the setting for node distance for mesh size control. Select “Mesh(M)” > “Set node density(M)” (Fig. 6.2-1). The displayed value 3.0 indicates “Base distance” and was the value entered when this analysis case was being generated. In this manner, user’s all inputs are saved in an analysis case. When no values are shown for other data items, as in this case, it means that the node distance is uniform. When you select a node distance, ensure that the mesh size is within the limit of your computer capacity, since the mesh size is proportional to the cube of the node distance. The size of elShape are approximately 50mm on the longest edge and the sectional shape is 10mm x 10mm. This will not make a too large number of elements if a 3.0 value is used.

Click “OK” without changing the value.

The next process is to generate surface patches. Select “Mesh(M)” > “Make Patch(P)” and the window for patch generation will appear (Fig. 6.2-2). Click “OK” to start generating patches.

After patch generation, generate a tetrahedron mesh. Select “Mesh(M)” > “Make Mesh(M)” and a window for mesh generation will appear (Fig. 6.2-3). Click “OK” to start generating mesh.

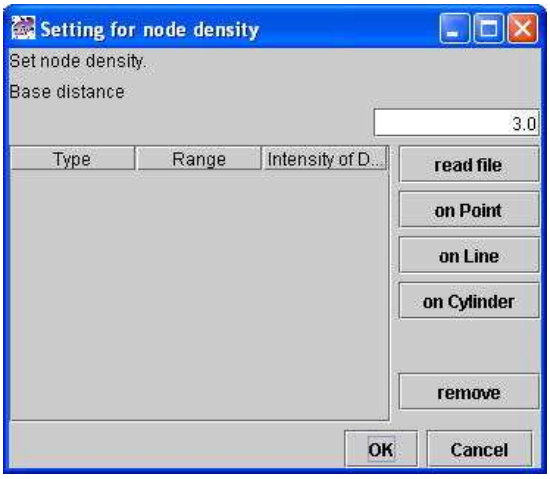


Fig. 6.2-1 Settings of base distance

Fig. 6.2-2 Generating surface patches

The generation process is reported as shown in Fig. 6.2-4. After completion, mesh information is displayed as shown in Fig. 6.2-5. Click “OK”. This completes patch and mesh generation. Next, set material properties and boundary conditions.

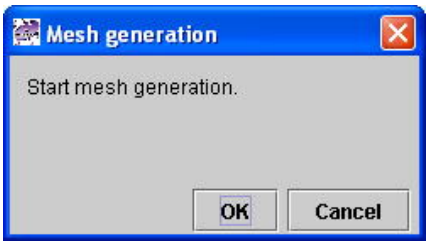


Fig. 6.2-3 Mesh generation

Fig. 6.2-4 One of mesh generation processes

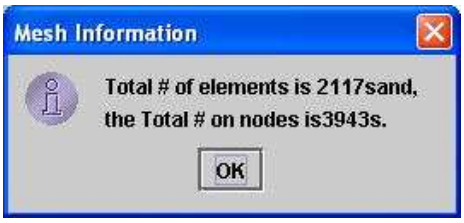


Fig. 6.2-5 Display of mesh information

6.3 Setting of Material Properties and Boundary Condition

For material properties, input in order of “Analysis(A)” > “Set Material property(M)” > “Solid(S)” (Fig. 6.3-1). Since values are also already given as part of this sample, click “OK” without changing anything.

Incidentally, the items in gray are values that need not be set for current analysis type (linier elastic analysis). If you want your analysis to consider gravity effect, tick “Gravity on” at the bottom.

Next, set boundary conditions. GUI module called BCtool which is newly developed for Windows is used for it and Agent guides the operation procedure.

First select “Analysis(A)” > “Boundary Conditions(B)”, then the window shown in Fig. 6.3-2 appears.

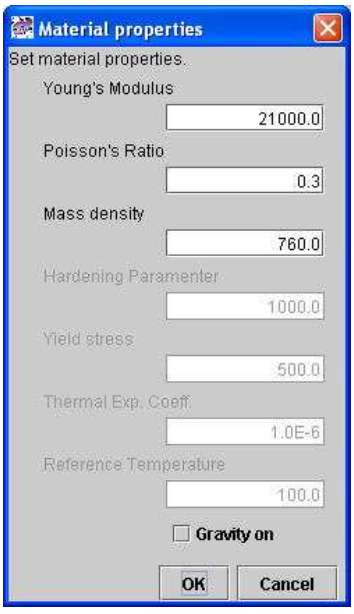


Fig. 6.3-1 Inputting material properties

Since mesh in ADVENTURE contains no topology information as CAD, after extracting surface triangles from the generated mesh, it is necessary to classify surface patches into faces based on normal vector direction of each triangle on mesh surface. By this classification, boundary condition can be attached face by face not triangle by triangle. As the resolution for grouping, the angle between the normal directions is used. Practically, integer value to divide 180 degrees by shall be input. The larger the value, the finer the face resolution becomes. Specify a divisor at (a) shown in Fig. 6.3-2. After inputting the value, press Enter key so that the angle that indicates actual resolution appears on the right: (b). When "OK" is clicked, the face is classified automatically and BCtool starts (Fig. 6.3-3).

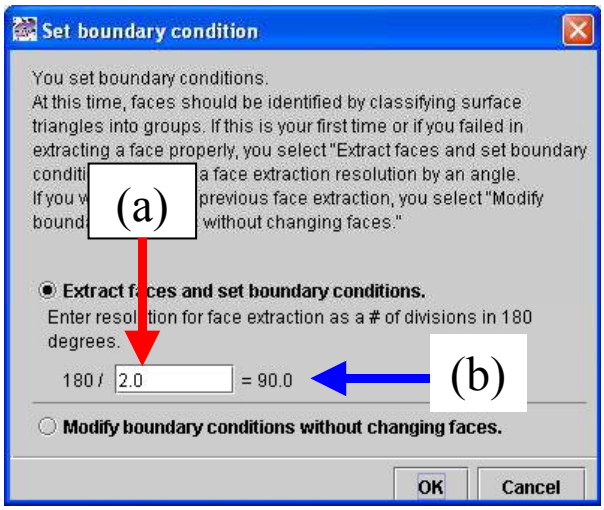


Fig. 6.3-2 Inputting grouping parameter of faces on mesh surface

At the same time, the Operation flow switches from current analysis procedure to operation procedure of BCtool (Fig. 6.3-4). Refer to the message window that indicates operation procedure and explanation.

For model rotation, drag the left button of the mouse and for zooming, drag while pressing the mouse wheel, the middle button or both right and left buttons at the same time, and for shifting, drag the right button.

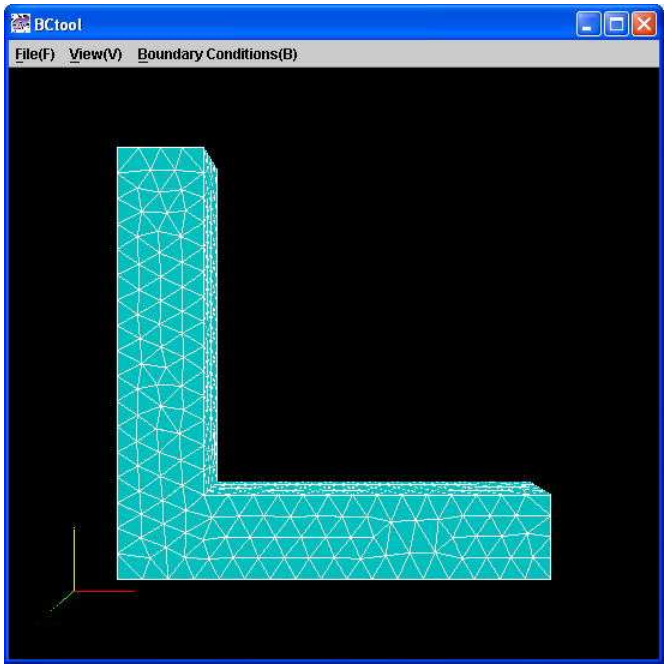


Fig. 6.3-3 Start up of BCtool

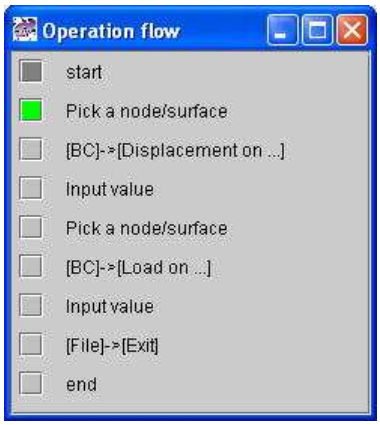


Fig. 6.3-4 Operation Flow of BCtool

For selecting face or node, just click objects with the left button of the mouse. As shown in Fig. 6.3-5, the selected face turns green and the selected node turns black at the same time.

At first, we explain the setting procedure for displacement, since in the Operation flow window displacement settings come ahead of load settings. After selecting face/node, select “Boundary Conditions(B)” > “Displacement on Face(D)” in the menu of BCtool, so that the constraint for the face can be specified (Fig. 6.3-6).

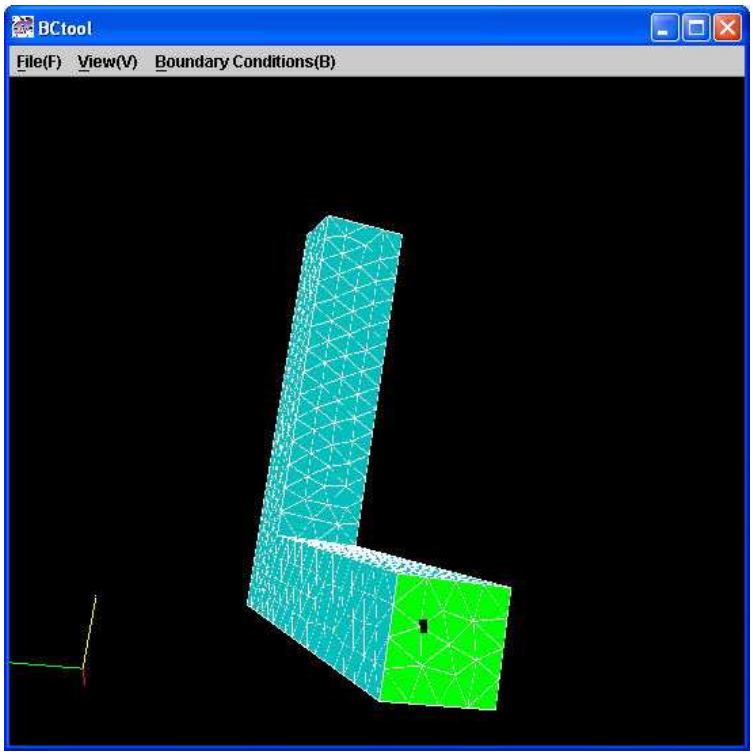


Fig. 6.3-5 Window of BCtool

In fact, the constraint setting is also used for the prescribed displacement. When only the directions to constrain is ticked and specified as zero, it can be constraint. If a value other than zero is entered, it specifies prescribed displacement.

When the “Boundary Conditions(B)” > “Displacement on Node(D)” is selected without changing face/node selection, constraint or prescribed displacement at the selected node can be specified as shown in Fig. 6.3-7.

Fig. 6.3-6 Specifying displacement on face

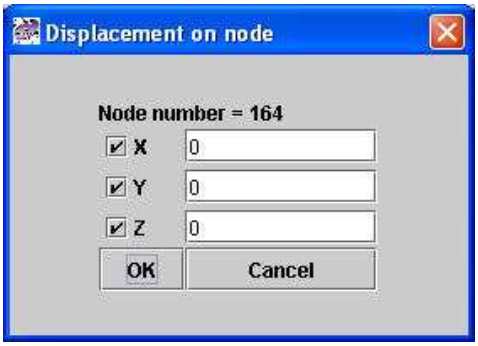


Fig. 6.3-7 Specifying displacement on node

Next, select face/node for load similarly (Fig. 6.3-8). By selecting “Boundary Conditions(B)” > “Load on Face(L)”, the load on the face can be specified as shown in Fig. 6.3-9. Tick necessary directions and input each component of the load vector. Bear in mind that the load is applied per unit area in this case. Enter values other than zero at least in one space. You will be warned if zero is entered for all values (Fig. 6.3-10). If you select “Boundary Conditions(B)” > “Load on Node(N)” with the same face/ node

selection, you can specify the load at the selected node as shown in Fig. 6. 3-11.

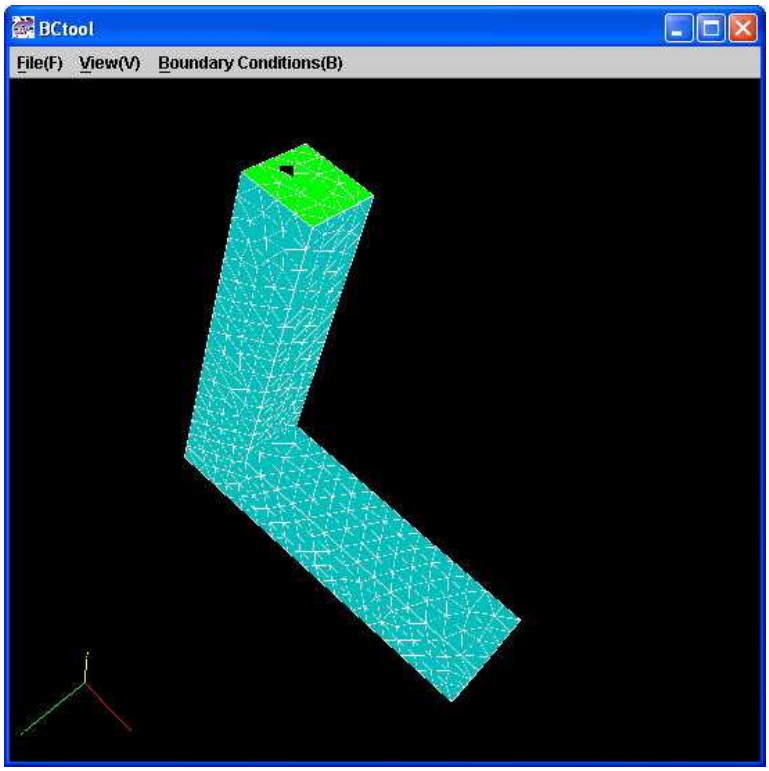


Fig. 6.3-8 Specifying a face/ node to set load

To confirm the settings you just entered, go to "View(V)" > "Boundary Conditions(B)" from BCtool menu (Fig. 6.3-12). A load on face 4 (1.0 per unit area in x direction) is described in line 3 to 5, and constrains on face 7 is described in lines 6 to 8.

Fig. 6.3-9 Specifying load on a face

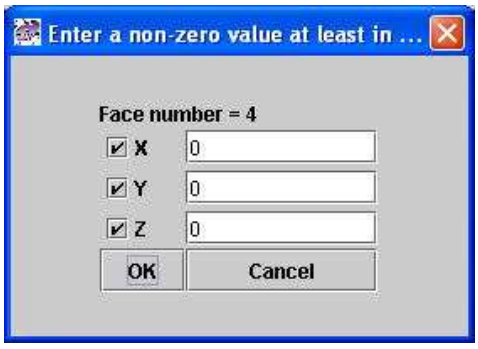


Fig. 6.3.10 Warning when all the values are zero

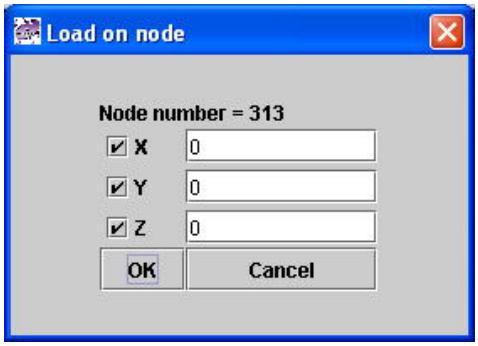


Fig. 6.3-11 Specifying load on a node

The top line displays gravity acceleration, which will not take effect unless the combo box “Gravity on” was ticked at the window for material property setting (Fig. 6.3-1). After completion of boundary setting, select “File(F)” > “Exit(X)” from the menu of BCtool to close BCtool. The boundary conditions are saved to a file automatically and identified by Agent, so that there is no need for the user to save conditions.

Fig. 6.3-12 Confirming boundary conditions

Several text files are created during the operation of boundary condition setting. Next, select “Analysis(A)” > “Convert to Input File(C)” to create an integrated input file in the binary format (Fig. 6-3-13). This concludes setting material properties and boundary conditions.

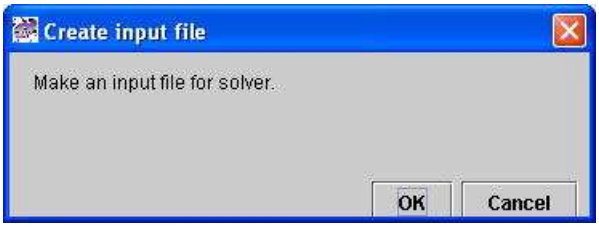


Fig. 6.3-13 Creating input file for solver

6.4 Running Solver

Before executing a solver, select items for results display and perform domain decomposition.

For setting items for result display, select “Analysis(A)” > “Set Result Display Item(O)” > “Solid(S)” from the menu window (Fig. 6.4-1). The items displayed in black are in effect for the current analysis type and all available items are ticked already. Click “OK”.

Although domain decomposition was originally required only for parallel analysis, please understand that it is needed even for single CPU analysis according to the characteristics of software ADVENTURE_Solid. Select “Analysis(A)” > “Domain Decompose(D)” from the main menu (Fig. 6.4-2). There is an option for domain decomposition, However, do not change anything and click “Start” as long as you use BDD solver, which we recommend.

After the above operation, calculation by solver commences. Select “Analysis(A)” > “Run Solver(R)”, then the window as shown in Fig. 6.4-3 appears.

If you want to give various options to the solver, give them using this window. Items that are not explicitly set will be set to standard solver settings.

In most cases, you just have to click “Start” and leave the settings as they are.

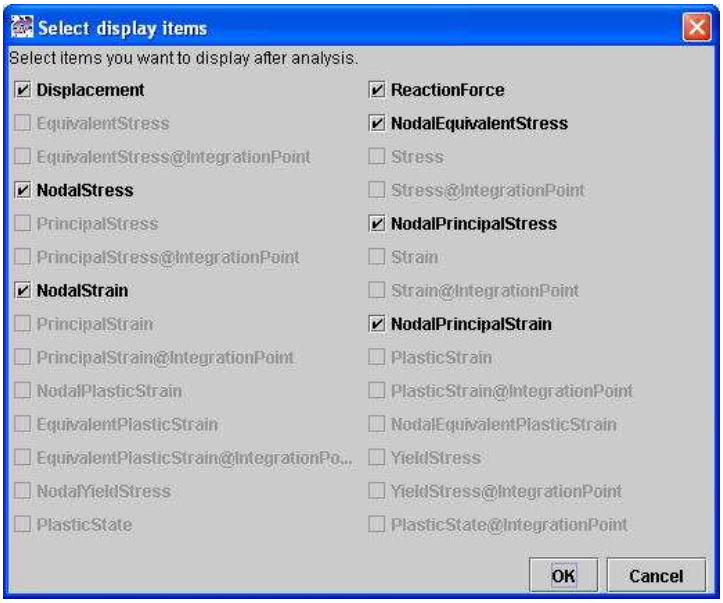


Fig. 6.4-1 Selecting result display items

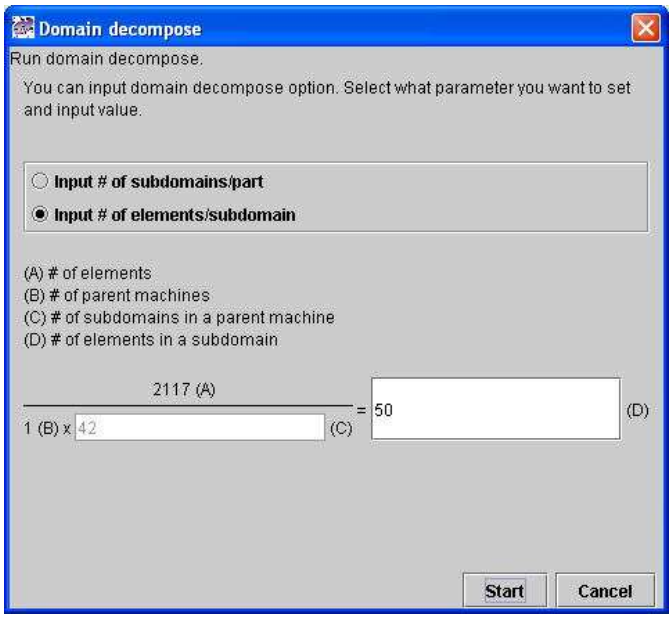


Fig. 6.4-2 Domain decomposition

In this case, the default BDD type will be selected. For your information, the CPU time required for this analysis on a laptop PC running Windows XP with Pentium Mobile 1.3GHz, 1GB memory is shown in Table 6.4-1. Please use this table as a rough guide since calculation time will be influenced by presence of OS's services, remaining capacity of hard disk or virus monitoring software, etc. In the case of the sample analysis: elShape, it took 3 seconds, 22MB, with the number of iterations to convergence 26 on the same machine (solver type is BDD).

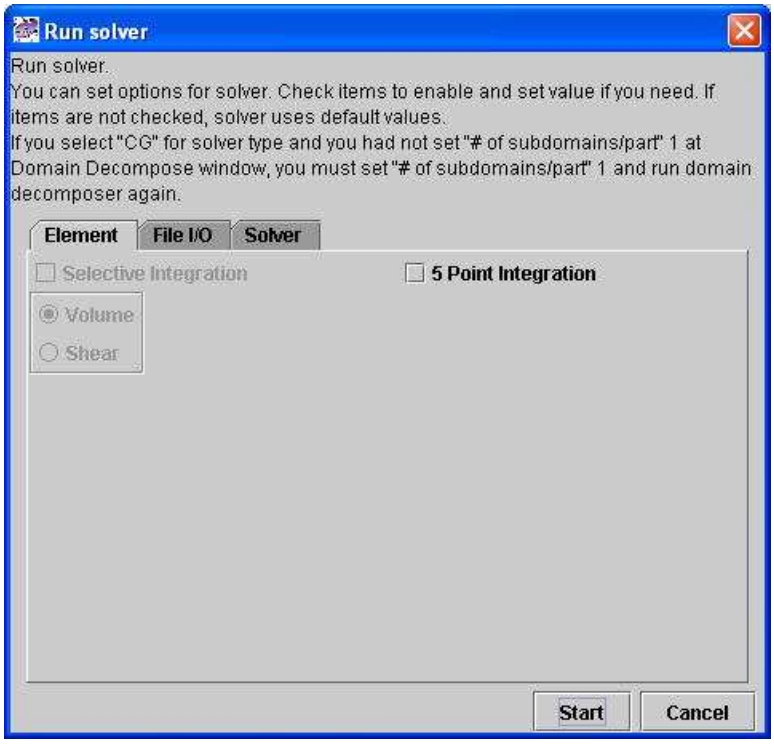


Fig. 6.4-3 Settings of solver options

After clicking “Start”, solver begins to analyze. During analysis, the window shown in Fig. 6.4-4 is displayed. The current version of Agent cannot display convergence history in real time (Future versions will address it as far as is possible). For solver interruption, terminate the process of advsolid-s.exe using the “Process” tab of the Window’s Task manager. Fig 6.4-5 will be displayed. If you click “Read details”, you can confirm convergence history up to interruption (Fig. 6.4-6).

Table 6.4-1 Reference CPU time for each solver type
(Problem description: Bending of an L-shape part, total elements 56.977, total nodes 84,947)

Solver type	Time (Second)	Used memory (Mbytes)	Iteration to convergence	Time/ Iteration (Second)
BDD	158	640	40	4.0
BDD-DIAG	139	429	43	3.2
HDDM	1109	343	2140	0.5
CG(1CPU=1 domain)	660	174	3727	0.2
HDDM (stiffness matrix not stored in memory)	17175 (4hr 46min.)	55	2140	8.0

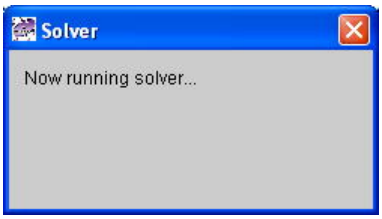


Fig. 6.4-4 During solver run

Fig. 6.4-5 Error display on interruption by Task manager

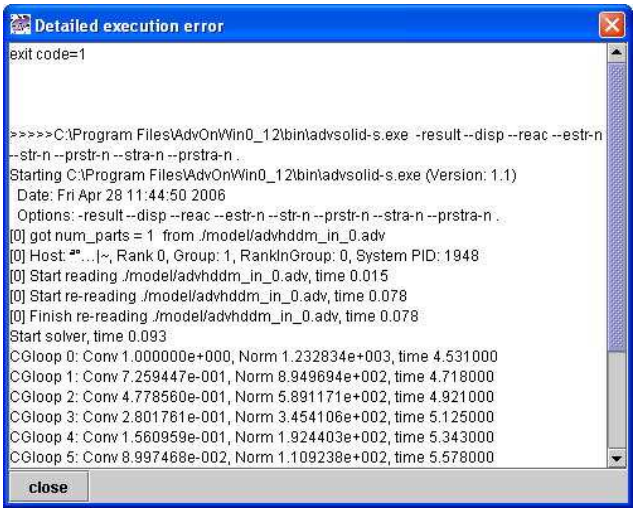


Fig. 6.4-6 Convergence history until interruption

6.5 Result Display

For result display, Visualtool which is a newly developed GUI module for Windows is used. Select "Analysis(A)" > "View Result(V)" from the menu, then the confirmation window (Fig. 6.5-1) will appear. In this window, the function of Visualtool is explained. After clicking "OK", Visualtool starts (Fig. 6.5-2). Unlike when starting BCtool, the Operation flow does not change to the operation procedure for Visualtool. This is because the operation procedure is simple and it is difficult to specify the order of operation in advance. The result displayed in Fig. 6.5-2 is for the elShape analysis. Please check whether you recover the same results.

Visualtool displays results of one component as a color contour at start up. In this example, x direction displacement is displayed as is shown on the right side of the label "Variable:" in the block just beneath the menu bar. Below the component name, maximum and minimum values of x direction displacement are displayed. Following them, the magnification factor of deformation is indicated as 1.0 in the block of "View". You will see that the vertical area of the L-shape is leaning slightly to the right. There is a check box "Show mesh:" below "Deformation". Tick the box to simultaneously show mesh (Fig. 6.5-3). The model is slightly rotated since mesh would be difficult to see if a model face is parallel to the plane of the display.

Mouse operation of rotation, etc. is the same as for BCtool. For model rotation, drag the left button of the mouse, for zooming, drag while pressing the mouse wheel, middle button or both right and left buttons at the same time, and for translation, drag the right button.

Mesh display is canceled by clicking the check box again.

Fig. 6.5-1 Window for visualization start-up

To change the magnification factor of deformation, modify the value of 1.0 and press the Enter key. If you wish to display the undeformed shape, set “Deformation” as 0. To restore the initial view after changing view settings, click “To default settings” in the View block.

In addition, to switch display components, pull down the combo box on the right of “Variable:” in the top block and show other displacement list. Select one and click it to switch (Fig. 6.5-4). An important function the setting panel does not have but is accessed via the menu is “View(V)” > “Pick-node mode(N)”. Using this, coordinate values at the picked node and the nodal value of the currently selected variable component can be displayed. As shown in Fig. 6.5-5, the block titled “Node info” will be added to the setting panel and the value such as “Node ID” will be indicated. Moreover, the picked node will be highlighted in yellow as shown at the blue area in the figure.

This is the end of operation instruction for stress analysis using the sample case elShape.iag.

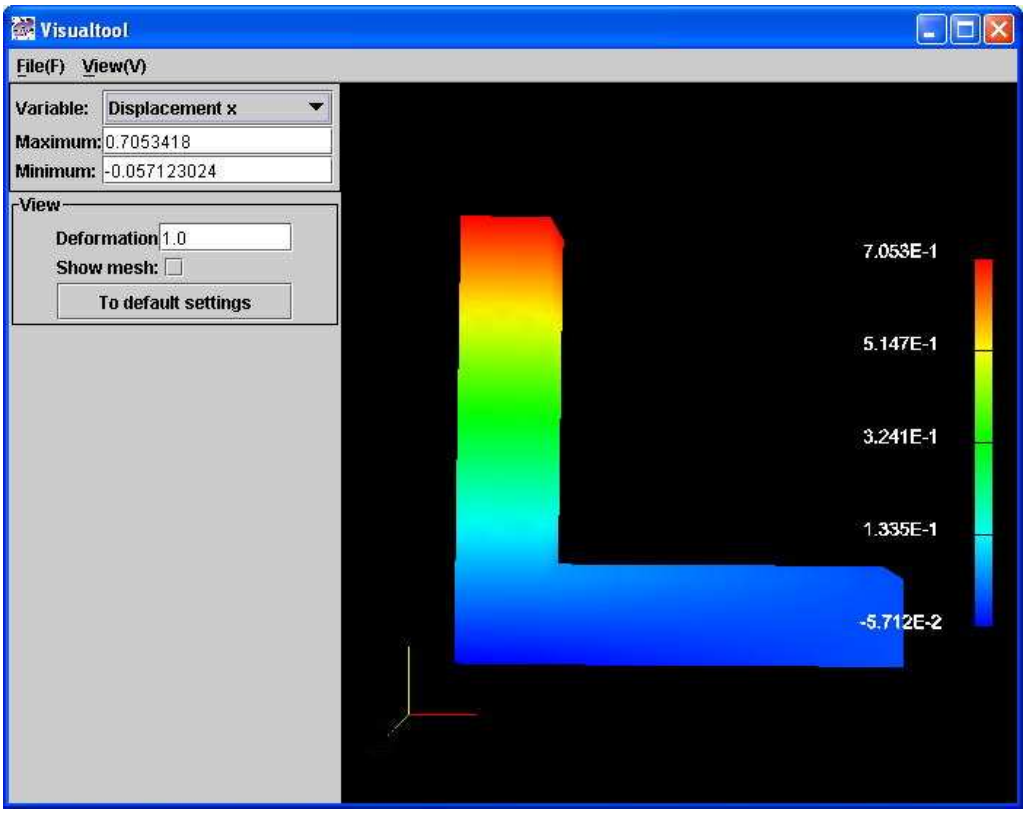


Fig. 6.5-2 At Visualtool start-up (values are recovered from sample elShape analysis case)

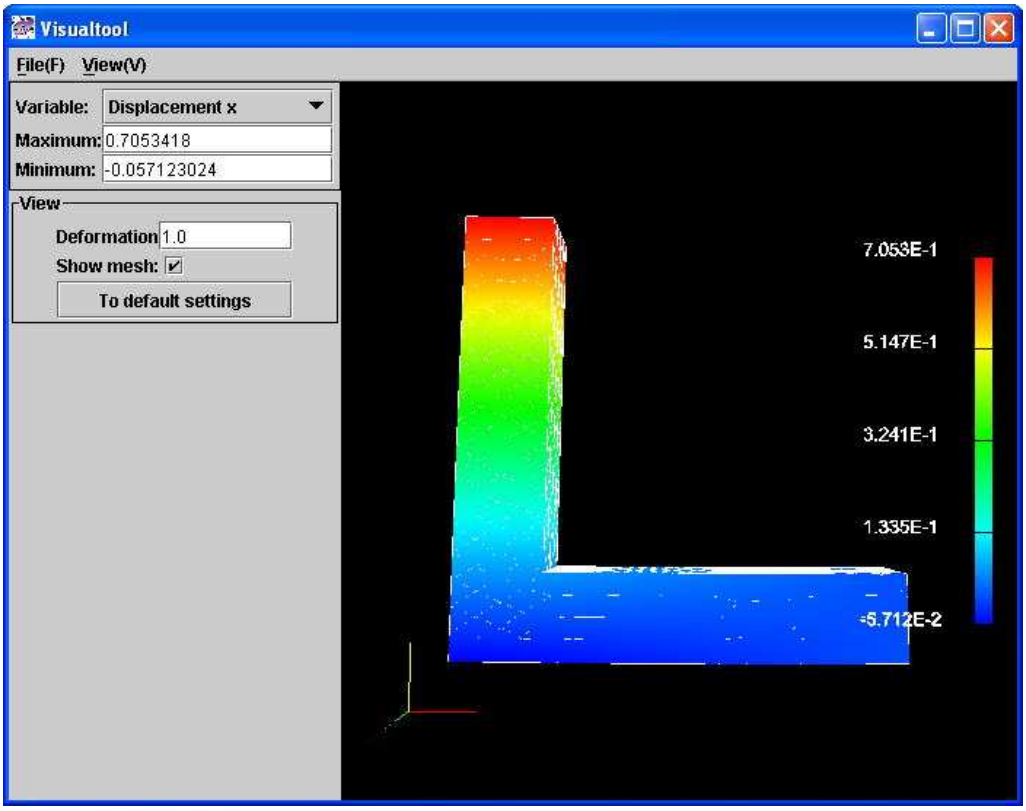


Fig. 6.5-3 Simultaneous display of mesh with result (values restored from elShape analysis case)

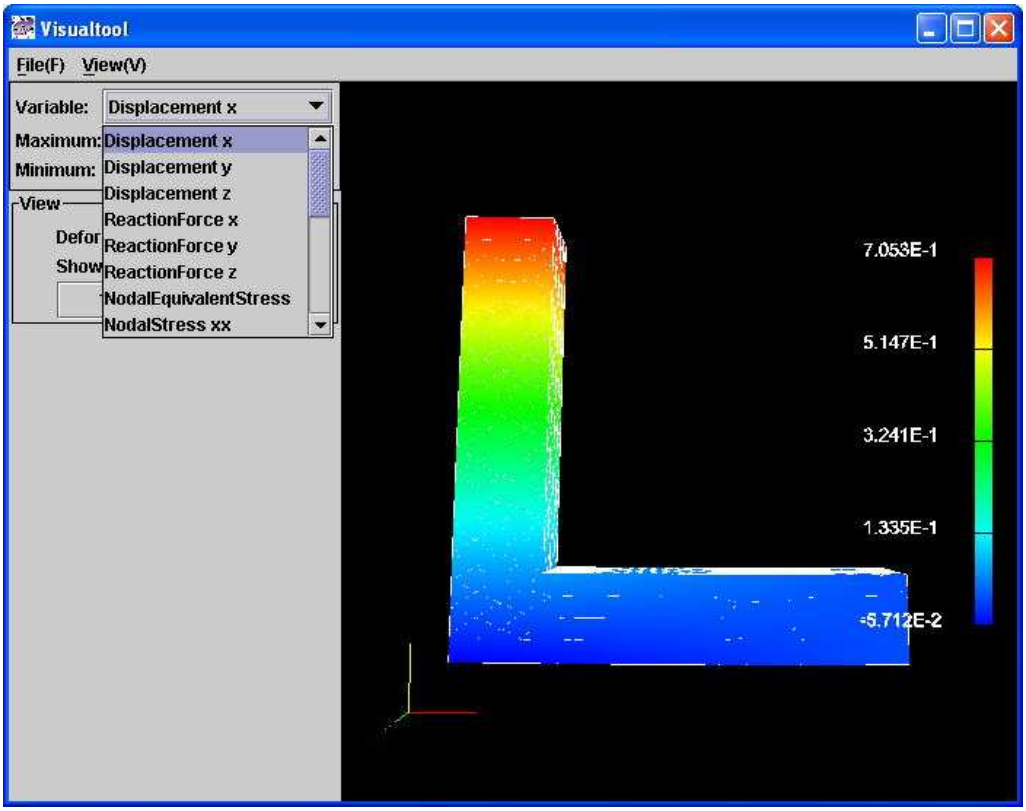


Fig. 6.5-4 Switching variable components to display (values are restored from elShape analysis case)

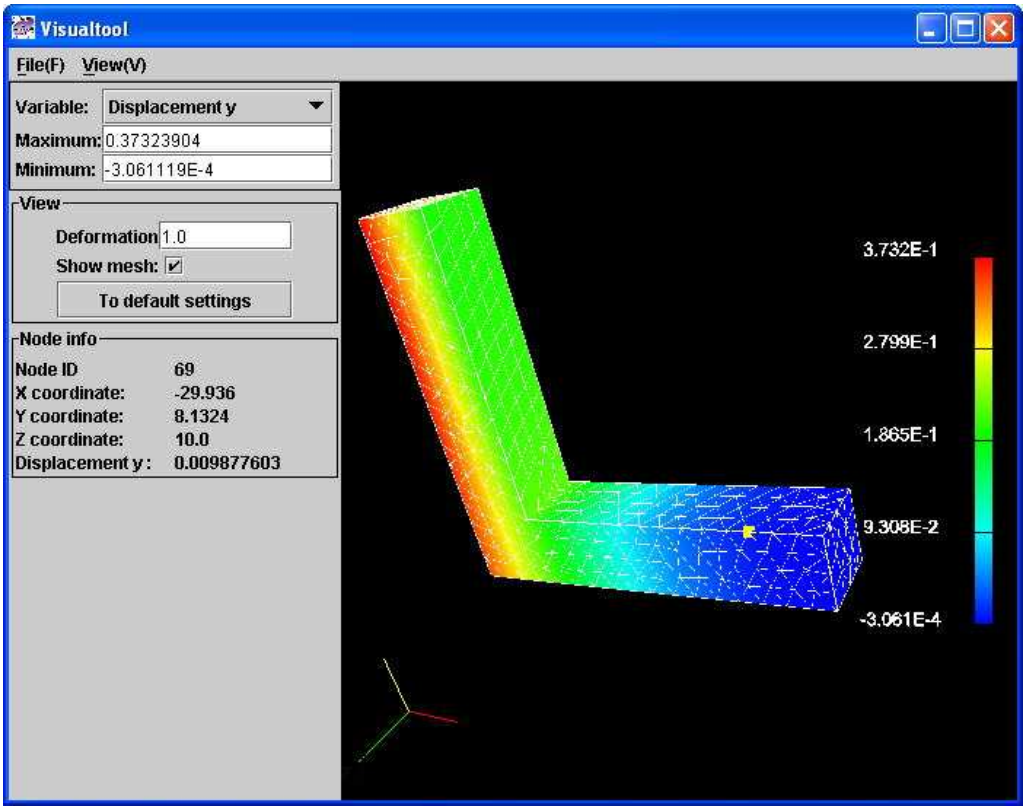


Fig. 6.5-5 Information display at node (each value is left as elShape analysis case)

7. Instructions of Other Functions

7.1 Creation and Saving of an Analysis Case

To start a new analysis case, select “File(F)” > “Create New Analysis Case(N)” from the menu window. You will be asked whether you wish to save the current analysis case (Fig. 7.1-1). If you need to save it, click “Yes” and if no, click “No”.

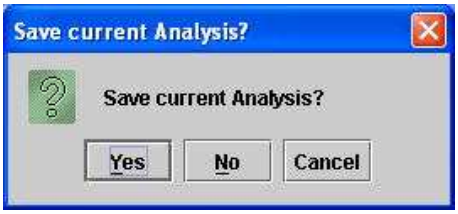


Fig. 7.1-1 Confirmation of the saving an analysis case

From the subsequent window, Agent starts to ask the user about their intensions in creating analysis case. Click “Next>” and go to the next stage (Fig. 7.1-2). First, select the type of problem to analyze. Since Agent currently supports only structure analysis, click “Next>” without making changes in the window (Fig. 7.1-3). Second, select a type of structure analysis. Again, since the Agent currently supports only linier elastic analysis, just click “Next>” (Fig. 7.1-4).

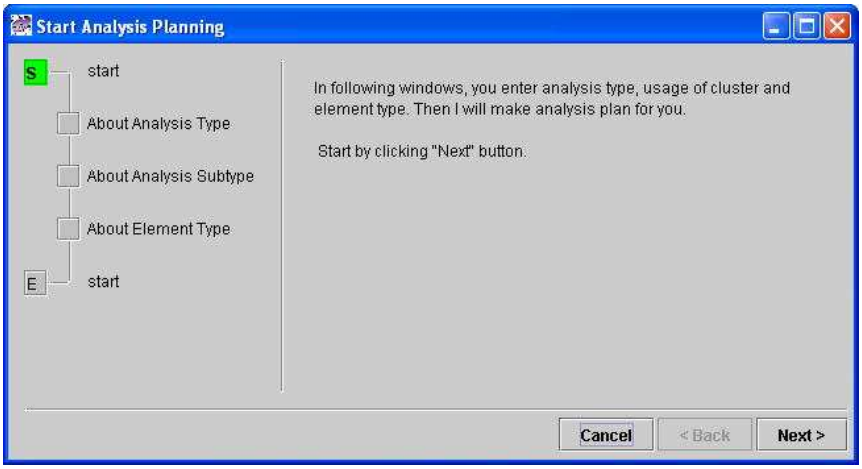


Fig. 7.1-2 Start of asking user intensions

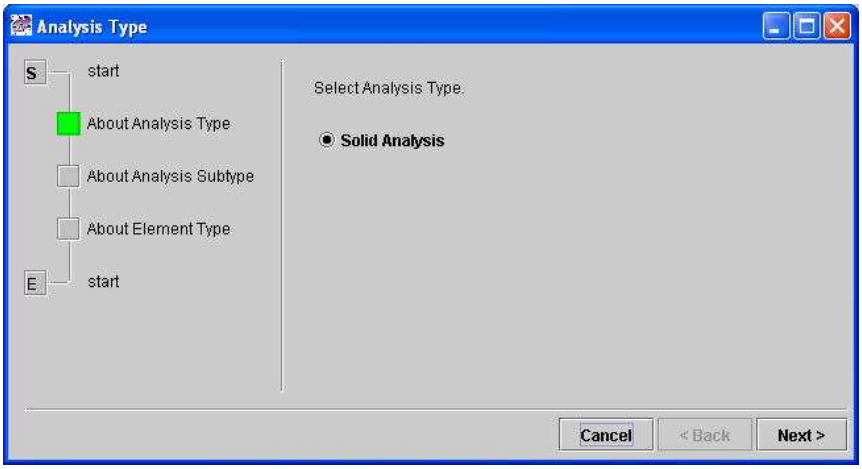


Fig. 7.1-3 Asking user intensions Part 2

At the last window Fig. 7.1-5, you must specify two items. One is the model type to input (Geometry model block) and the other is element type (Analysis model block) to use. There are three types for input geometry: IGES files which are a CAD models, mesh file (*.msh) and surface patches (*.pch) in ADVENTURE format. *.pcm surface patch file is supported if it has only 1 volume(single material type). The supported IGES file format will be explained later in chapter 10.1.

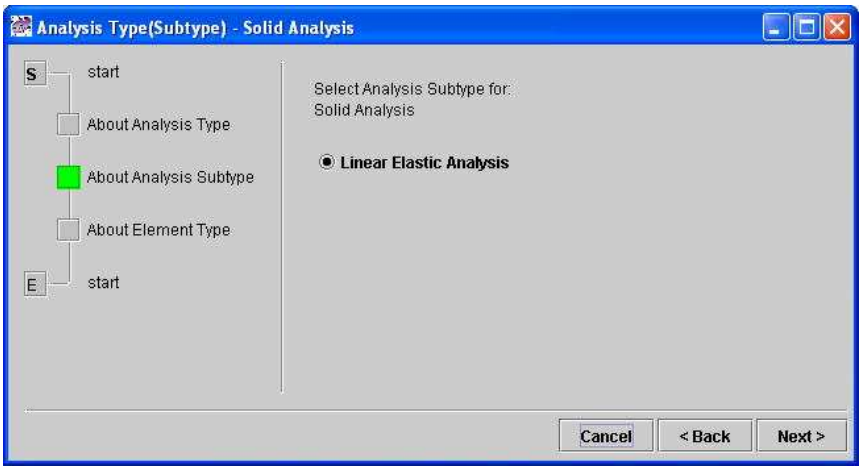


Fig. 7.1-4 Asking user intensions Part 3

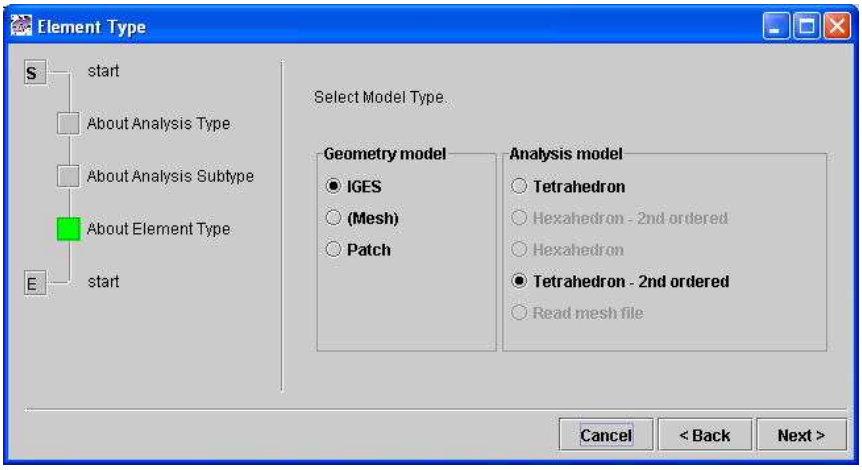


Fig. 7.1-5 Asking user intensions Part 4

Supported element types are the first and the second order tetrahedron. “Tetrahedron” means the first order tetrahedron. After selecting one for Geometry model and Analysis model, click “Next>”. If you select “(Mesh)” for the Geometry model, the display in the Analysis block changes as in Fig. 7.1-6.

After clicking “Next>”, the display in the menu window changes as shown in Fig. 7.1-7. Follow the instructions given by Agent.

To save a current analysis case, select “File(F)” > “Save Analysis Case As(A)” or “File(F)” > “Save Analysis Case(S)”. Even if a latter menu item is selected at the first time, the dialog for saving an analysis case will be displayed as in Fig. 7.1-8. Specify a file name for analysis case (*.iag). “.iag” will be added automatically even you do not put it.

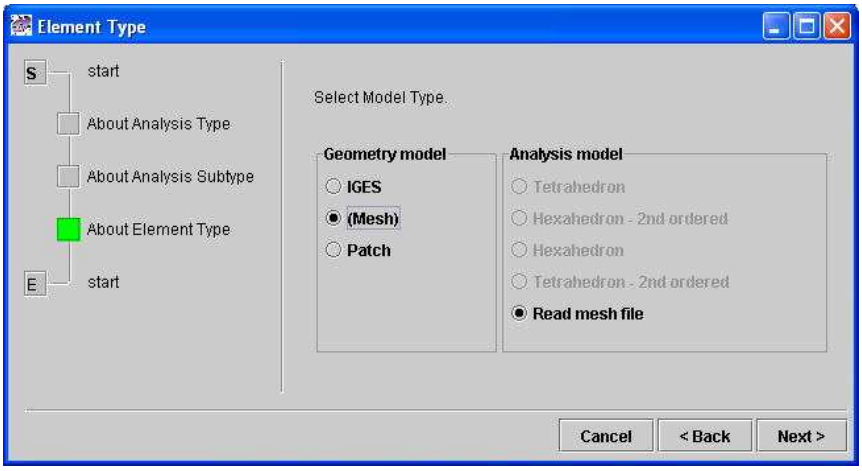


Fig. 7.1-6 Asking user intensions Part 4-2

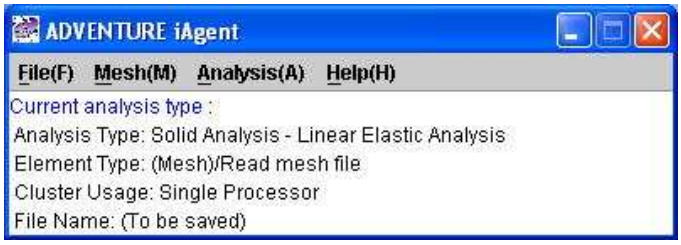


Fig. 7.1-7 Changed analysis case

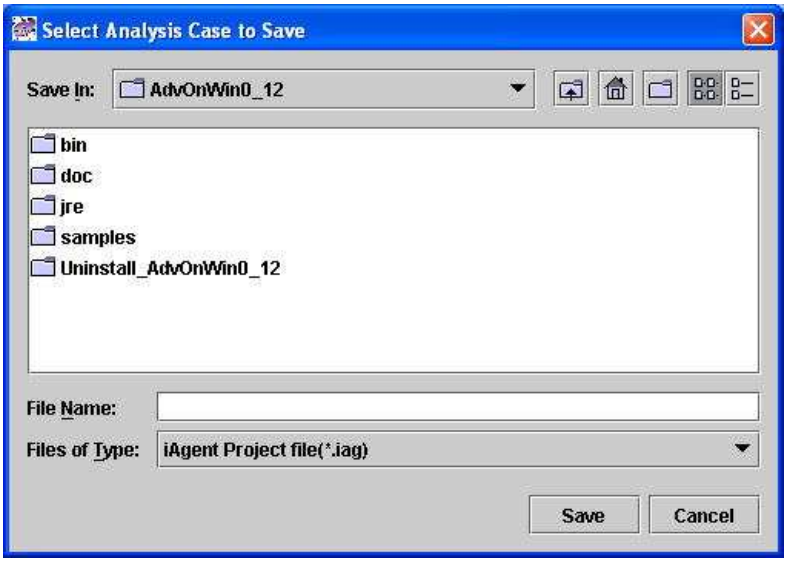


Fig. 7.1-8 Dialog to save an analysis case as a different name

7.2 Reading Geometrical Model and Display of Surface Patch/ Mesh

"Mesh(M)" > "Select AdvCAD File(C)"

This function is not currently available.

"Mesh(M)" > View Patch(V)"

Specify like:

VRMLViewer = c:/Program Files/Meshman_ViewerV0_7/Meshman_ViewerVer.0.7beta.exe

in line 40th of the iAgent.conf file in the installation folder. The absolute path of the surface patch file, Solid.pcm will be passed to the program specified on the right hand side of "=" as the first argument. Any programs that satisfies this specification can be used. Change to this configuration will be reflected when this program is started after the modification. At present, Meshman_Viewer (Free) of Insight, Inc. is identified as an applicable program.

"Mesh(M)" > "View Mesh(H)

Agent specifies the same program as for surface patch as a viewer for tetrahedron mesh. In other words, the visualization program described in iAgent.conf needs to correspond to both of *.msh and *.pcm.

"Mesh(M)" > "Load Patch(R)"

A surface patch file is read. Specify the surface patch file in the dialog shown in Fig. 7.2-1. A *.pcm file that has only one volume is also readable.

"Mesh(M)" > "Load mesh(L)"

A mesh file is read. Specify the mesh file in the dialog as shown in Fig. 7.2-2.

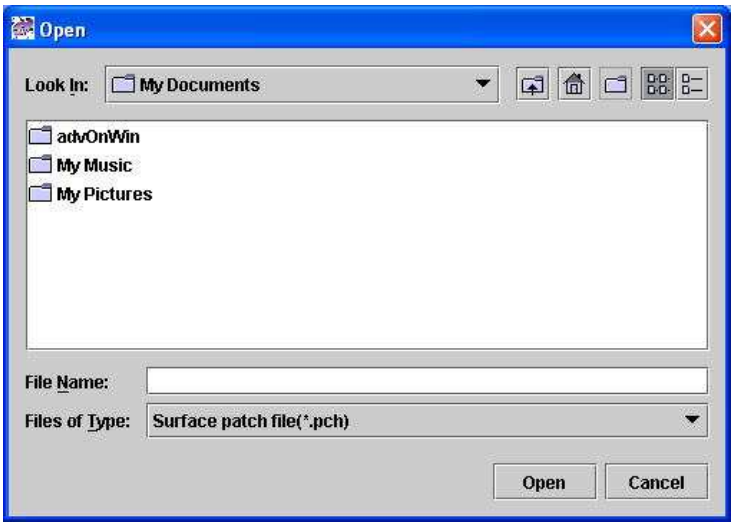


Fig. 7.2-1 Reading a surface patch file

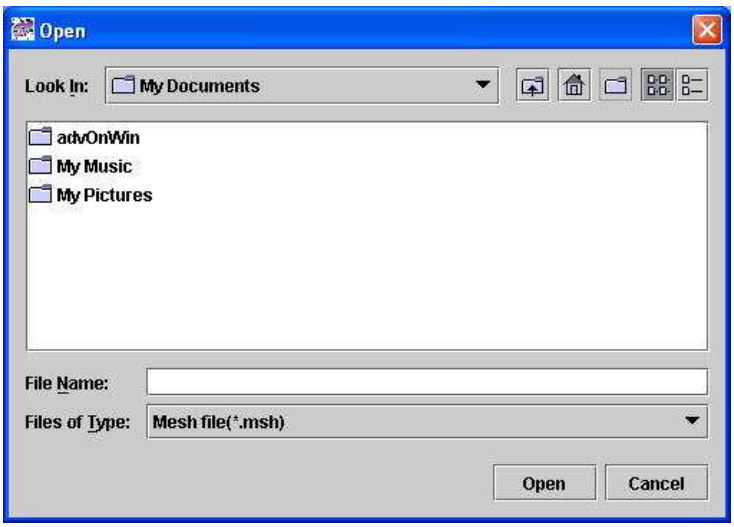


Fig. 7.2-2 Reading a mesh file

7.3 Boundary Condition Setting Function

Additional explanation concerning the function of BCtool.

"File(F)" > "*Open .pch & *.pcg(O)"

This is the function to use only when you use BCtool as a standalone tool. It will not be used when you use BCtool via Agent.

"File(F)" > "Save *.cnd(S)" and "File(F)" > "Save *.cnd as(A)"

These are also functions to be used only when you use BCtool as a standalone tool. It is not required when you use BCtool via Agent.

"View(V)" > "Model Info(I)"

Model information as shown in Fig. 7.3-1 will be displayed.

"Boundary Conditions(B)" > "Clear Boundary Conditions(C)"

A confirmation dialog as shown in Fig. 7.3-2 will appear. If it is ok, click "OK".

"Boundary Conditions(B)" > "Gravity Acceleration(G)"

After inputting each component of the gravity acceleration vector in the dialog of the gravity

acceleration - Fig. 7.3-3, click “OK”. This function is effective only when you select the checkbox “Gravity on” in the setting of material properties mentioned above at section 6.3.

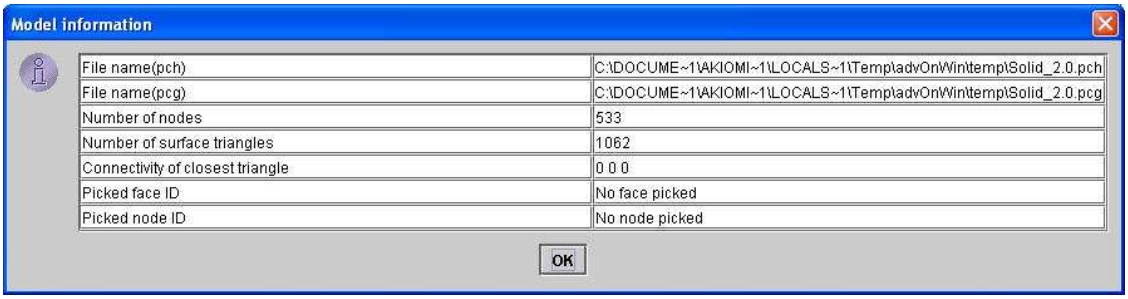


Fig. 7.3-1 Display of model information

Fig. 7.3-2 Confirmation to clear boundary conditions

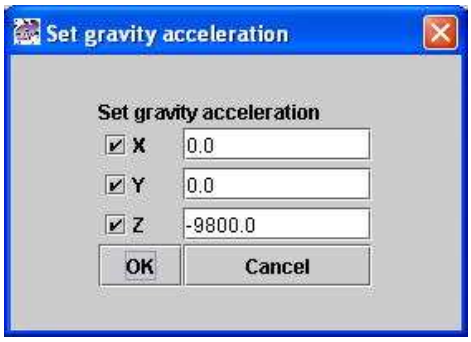


Fig. 7.3-3 Setting of gravity acceleration

7.4 Domain Decomposition Function

When using the domain decomposition function, care must be taken if you choose the CG method as the solver. The number of subdomains per part is to be one. In the dialog displayed when you select “Analysis(A)” > “Domain decompose(D)” from the main menu (Fig. 6.4-2), select the radio button “Input # of subdomains/part”. Then the item (C) i.e. the number of subdomains in a parent machine switches from disabled state to enabled state. Enter 1 in the item (C) (Fig. 7.4-1).

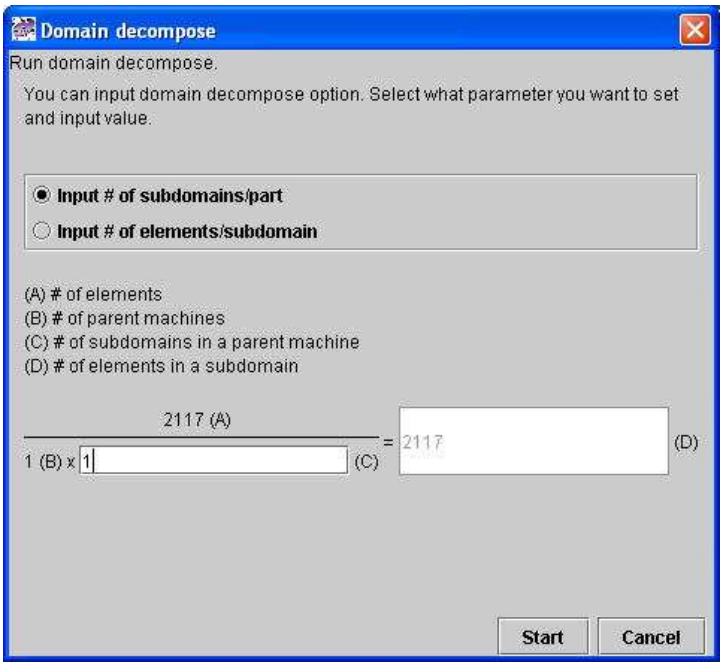


Fig. 7.4.1 Setting domain decomposition parameter for the CG method

The figure for the item (D) changes automatically so that equality holds.

7.5 Solver Execution Function

Here is the instruction that has not been given in the instruction for dialog of solver option settings in Section 6.4. With the “Element” tab, options for elements can be set. (Fig. 7.5-1).

- If you select the “5 Point Integration” checkbox, a 5 point integration will be used for element integration (this option is available only for the second order tetrahedron).

With “File I/O” tab (Fig. 7.5-2), the following settings on input or output of files are available.

- If you select the “Parallel File I/O” checkbox, input/output of files is performed in parallel on each process.
- Maximum memory size can be set for each process.

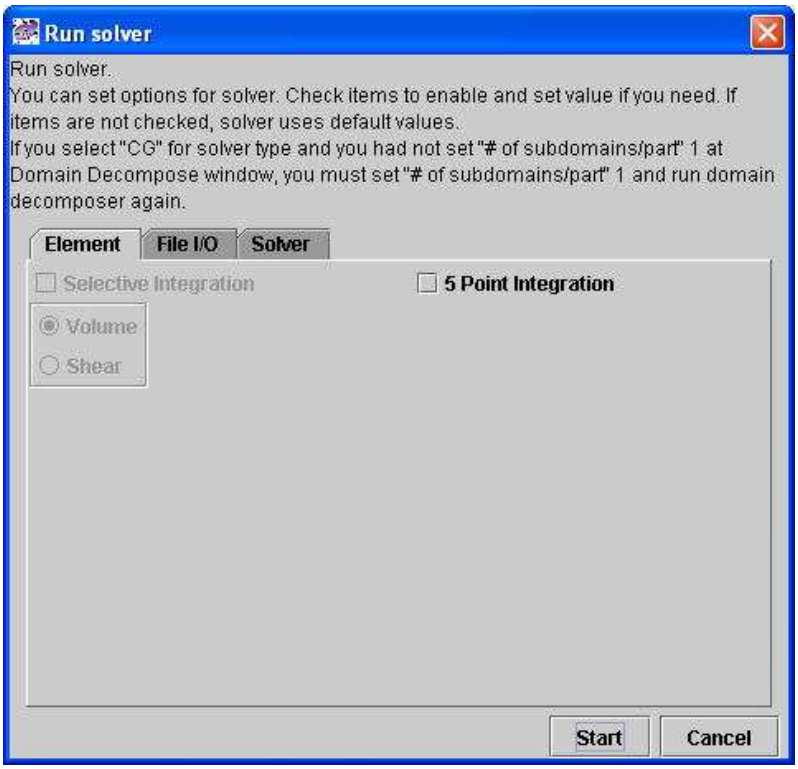


Fig. 7.5-1 Element option settings when running solver

With the “Solver” tab (Fig. 7.5-3), the following settings for the solver/ iterative method are available.

- Type of solver by ticking “Solver type”.
- Tolerance for convergence of CG method by ticking “CG Tolerance”.
- The upper limit for the number of CG iterations by ticking “Max CG Loop count”.
- Memory saving by not memorizing a stiffness matrix by clicking “Not Keep K-Matrix” (valid only for the HDDM solver).
- On ticking “Regr. Parameter” you can specify regularization parameters for Neumann-Neumann preprocessing (valid only for the BDD solver).
- Setting the tolerance for global convergence by ticking “Newton Tolerance” (currently not available since it applies only to non-linear analysis).
- Setting the upper limit for the total number of iterations by ticking “Max Newton Loop” (currently not available since it applies only to non-linear analysis).

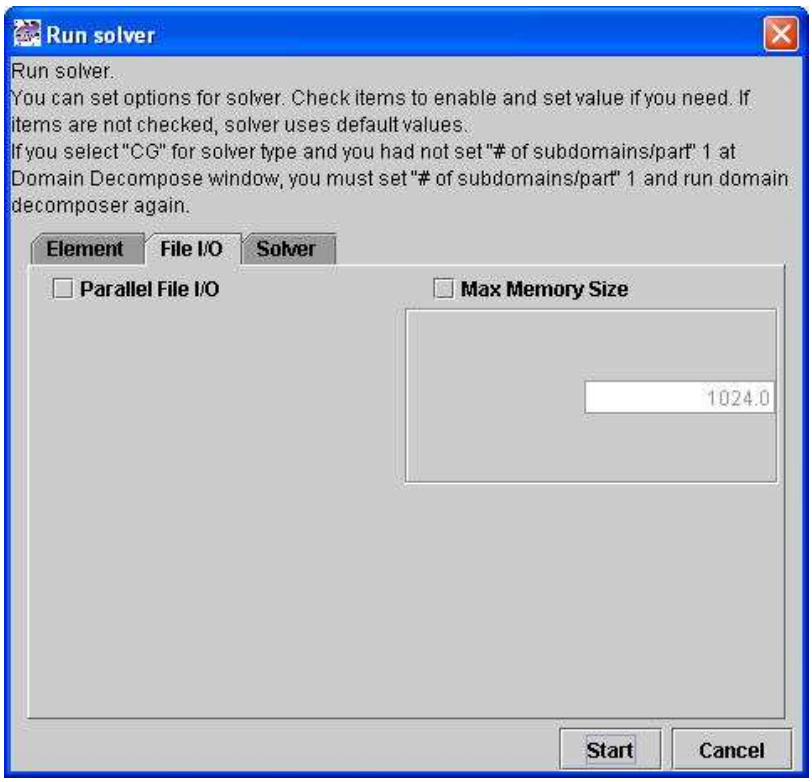


Fig. 7.5-2 Option settings on file input/output when running a solver

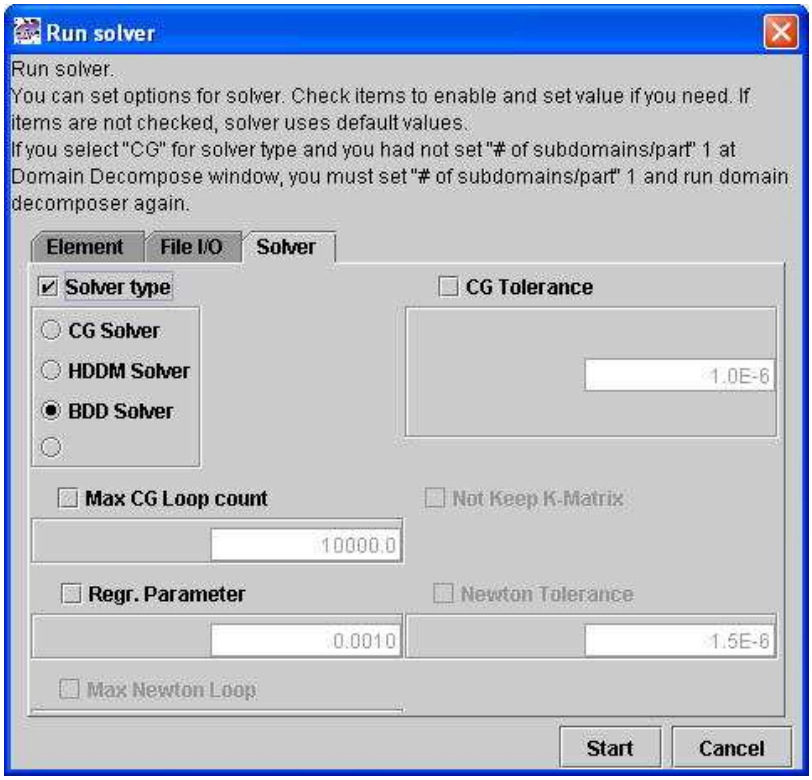


Fig. 7.5-3 Solver option settings when running a solver

7.6 Result Visualization Function

There are additional instructions for functionality of Visualtool.

“File(F)” > “Save Image(S)”

The image displayed in the visualization area can be captured and saved in JPEG format (Fig. 7.6-1).

“View(V)” > “Projection(P)” > “Orthographic(O)”

Switches to the parallel projection where far objects and near objects appear at the same scale. Zooming capability is disabled.

“View(V)” > “Projection(P)” > “Perspective(P)”.

Switches to the perspective projection. This is the default (Fig. 7.6-2).

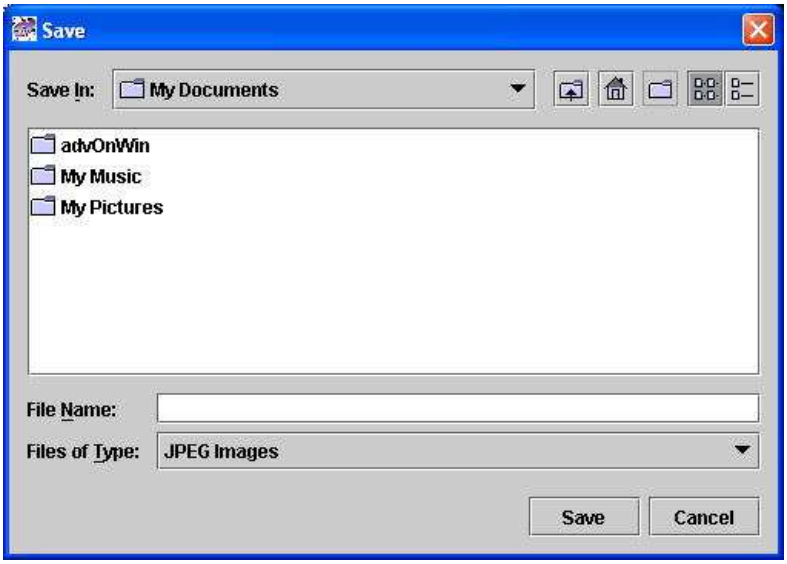


Fig. 7.6-1 Specifying a file to save image

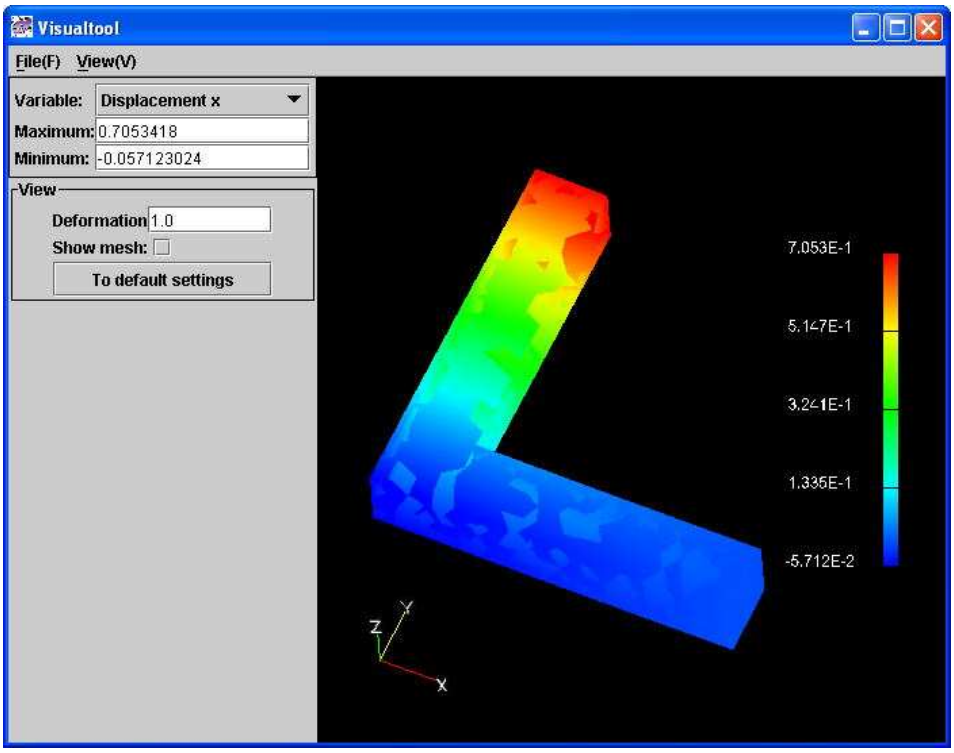


Fig. 7.6-2 Orthographic view

8. Procedure for Standalone Use of GUI Tools

8.1 Procedure for Standalone Use of BCtool

Standalone use of BCtool is available only in Japanese mode. If you wish to use it in Japanese mode, please refer to the Japanese manual.

8.2 Procedure for Standalone Use of Visualtool

Standalone use of Visualtool is available only in Japanese mode. If you wish to use it in Japanese mode, please refer to the Japanese manual.

9. Sample File

Sample files are prepared in the folder “samples” which is in the folder where ADVENTURE on Windows is installed. Table 9-1 shows the list of sample files.

Table 9-1 Sample files

File name (or folder name)	Explanation
elShape.igs	IGES file of an L-shape part
elShape.iag	Analysis case using the IGES file of an L-shape part
elShape.files□	All files in this folder are related to the elShape.iag analysis case. Caution: if any one file is deleted, the analysis case will not be readable. In this analysis case, whole process is complete. Therefore if this analysis case is loaded, Results can be shown without calculation.
4 files that begin with cShape	Sample files for a standalone use of the BCtool.
Solid.msh initialize.list Displacement.dat NodalStress.dat MaximumNodalPrincipalStress.dat	Sample files for a standalone use of visualization tool.

10. Analysis of An Arbitrary Shape File

10.1 IGES File

IGES files applicable to the ADVNEURE on Windows have a number of restrictions. The following is an excerpt from the manual of ADVENTURE_TriPatch.

*** Start of excerpt. ***

(1) Compliant with IGES specifications Ver5.3 (ASCII format)

(2) Supports NURBS (Non Uniform Rationalization B-spline Surface) curved surface solid format.

・When IGES data is created as solid input, entity number 186 must exist.

・If entity number 186 does not exist, the program will generate an error.

(3) The program assumes IGES files are created by one of the following CAD programs.

a. I-DEAS Master Serise 8

b. MicroCADAM V4R2

c. Any other CAD that uses DESIGNBASE as a kernel

(4) Supported entities

No	Entity number	Entity name
1	100	Circular arc
2	110	Line
3	124	Transformation matrix
4	126	Rationalization B-spline curve
5	128	Rationalization B-spline surface
6	186	Manifold solid B-Rep object
7	502	Vertex
8	504	Edge
9	508	Loop
10	510	Surface
11	514	Shell

*** End of excerpt. ***

10.2 Surface Patch File

Since the current program cannot handle multi-volumes, these instructions are related to *.pch format only. Refer to this section for surface mesh conversion from other formats. Connectivity of each triangle is clockwise as seen from outside of the object.

576 ->1st line, total number of nodes
-2.0 -2.5 0. ->576 lines data, nodal coordinate values x, y and z for the 0th through 575th node
Snip
1148 ->total number of triangles
259 0 28 ->1148 lines data, connectivity of the 0th through 1147th triangle element
Omit the rest

10.3 Tetrahedral Mesh File

The extension is msh. Both the node number and the element number start at zero. Connectivity of the tetrahedron is as shown in Fig. 10.3-1. The following is an example of a second order tetrahedron mesh.

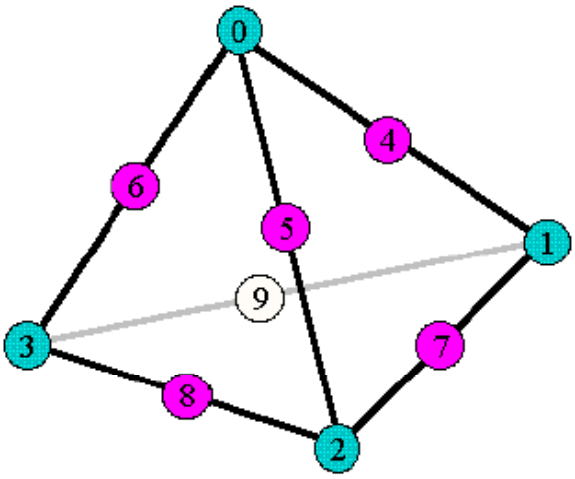


Fig. 10.3-1 Connectivity of tetrahedral element in ADVENTURE

3313 ->1st line, total number of elements
28 0 23 259 851 852 853 854 855 856 ->3313 lines data, connectivity of the 0th to 3312th element
Snip
5588 ->total number of nodes
-2.5 –2.5 0 ->5588 lines data, nodal coordinate values x, y and z for the 0th to 5587th node
Omit the rest

11. Log and Work Folders

11.1 Log

An execution log of each ADVENTURE module/ tool is saved in a folder created as advOnWin under "My documents". Refer to these files when you need detailed log information on ADVENTURE modules/ tools. The following Table 11-1 shows the relation between module/ tool names and log file names.

Table 11-1 Log file names of ADVENTURE on Windows

Function	Module name/ Tool name	Log file name
Surface patch generation	TriPatch	CreatePatch.log
Surface patch display	External mesh viewer	ViewSurfaceVRML.log
Surface patch correction	advtmesh9p.exe	CorrectPatch.log
First order mesh generation	advtmesh9m.exe	GenerateMesh.log
Conversion to second order mesh	advtmesh9s.exe	AttachSecondNode.log
Mesh display	External mesh viewer	ViewMeshVRML.log
Surface extraction	faceOfMesh.exe/makepch.exe	CreateSurfaceGroup.log
Input file creation	makefem.exe	ConvertAdvIO.log
Domain decomposition	adventure_metis.exe	ExecDomainDecomposeForWin.Log

Stress analysis	advsolid-s.exe	ExecSolverForWin.log
Converting results into text format	hddmmrg.exe	ExecHddmMerg.log
Saving an error message whatever the module generates an error.		Error.log

Agent itself would not normally output a log. If you need a log report for a bug, edit the following two lines in the file AdvOnWin0_12.lax in the folder where the program is installed.

```
lax.stderr.redirect=
lax.stdout.redirect=
```

Change the above lines as follows:

```
lax.stderr.redirect=console
lax.stdout.redirect=console
```

After saving the .lax file, start the program normally, a “command prompt” window will appear containing the log. Caution: when you shut down this program, the “command prompt” will disappear. If you specify a specific file name instead of “console”, the “command prompt” window will not appear and a log will be recorded in the file specified. Be sure to use two consecutive □ symbols as a separator for each folder name as in the following example: C:\\tmp\\advOnWin0_12.log.

11.2 Work Folder

When you are using this program, the program automatically creates a work folder

Documents and Settings\your user name\Local Settings\Temp\advOnWin\temp\

Be careful if you delete files in this folder while this program is running, an error will occur. They may be deleted after the program has been shutdown.

12. Modification of the Maximum Value for Heap Memory

Depending on the scale of problems being analyzed, heap memory for the Agent (especially for Visualtool) may be exhausted. In such a situation, the program will stop responding. Check if you can see the log of Agent using the procedure described in chapter 11. If you can, the following will be seen on the last line.

java.lang.OutOfMemoryError

Edit the following lines in AdvOnWin0_12.lax to increase the initial value and the maximum value of the heap memory. The default is 128MBytes and 512MBytes respectively. Please be careful that the memory does not mean the one the solver uses.

```
# LAX.NL.JAVA.OPTION.JAVA.HEAP.SIZE.INITIAL
# -----------------------------------------
# Initial heap memory

lax.nl.java.option.java.heap.size.initial=128M

# LAX.NL.JAVA.OPTION.JAVA.HEAP.SIZE.MAX
# -------------------------------------
```

```
# max heap memory

lax.nl.java.option.java.heap.size.max=512M
```

13. Uninstalling Procedure

Select “Control Panel” and then “Add or Remove Programs” from “Start Menu” on Windows (on Windows 2000, “Start” > “Settings” > “Control Panel”). Select “AdvOnWin0_12” from the list of installed programs, then click the “Remove” button. The window shown in Fig. 13-1 will appear and click “Uninstall” (Fig. 13-2). After completion of uninstalling, the window shown in Fig. 13-3 will appear. Click “Done” (If you have saved data files to the samples folder, etc., they will not be deleted).

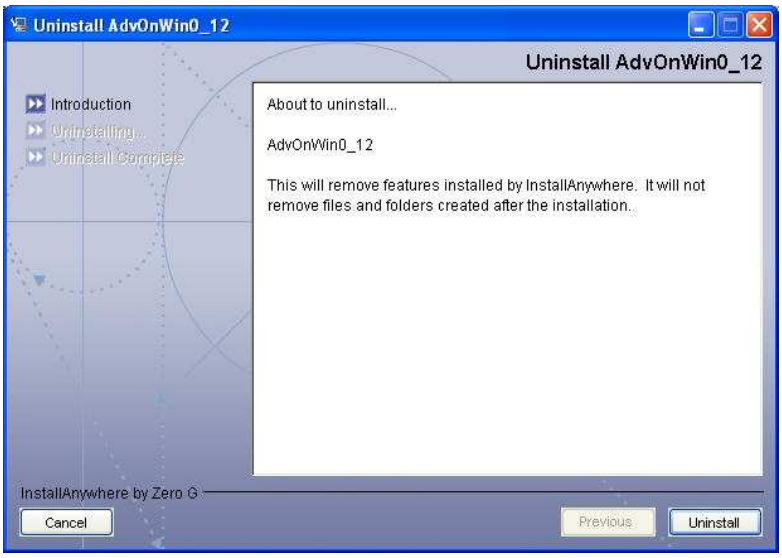


Fig. 13-1 Start of uninstalling

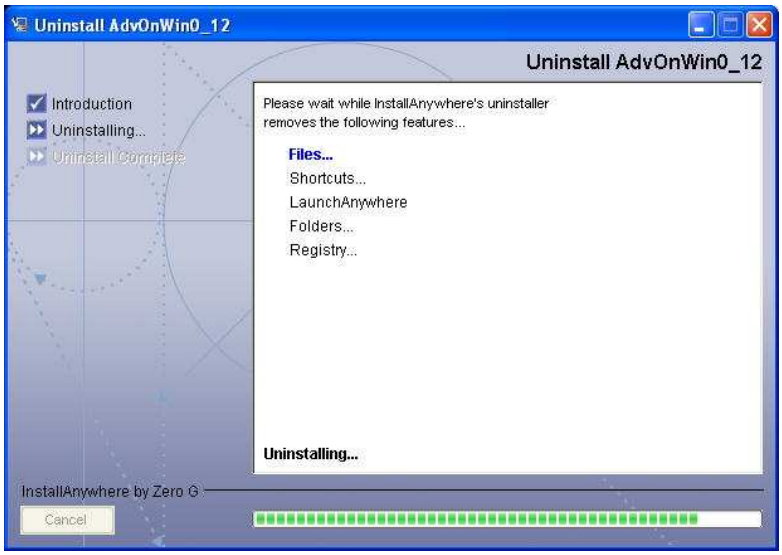


Fig. 13-2 During uninstalling

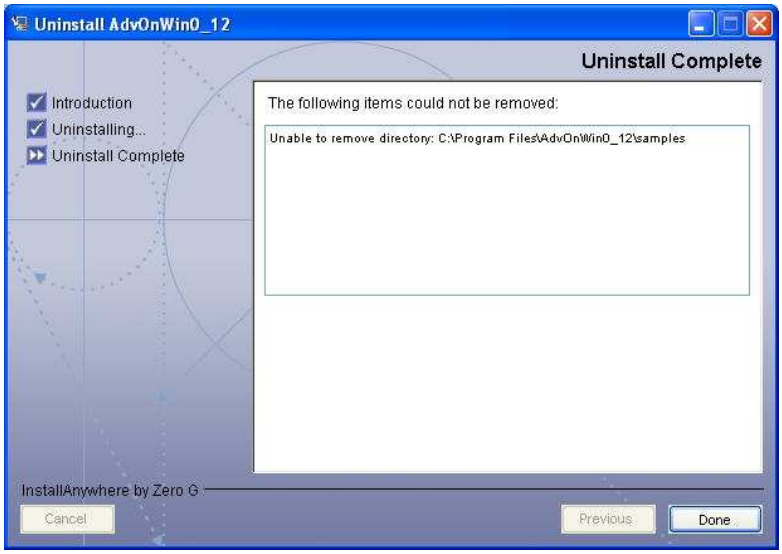


Fig. 13-3 Uninstalling complete

14. FAQ

The following Table 13-1 is a FAQ list.

Table 13-1 FAQ

#	Question	Answer
1	Can you change the operating method from the current menu based one to the Windows wizard type in which a dialog of next step is displayed one after another?	We cannot change it since this software is based on the idea of iAgent that the program can be used as an ordinary software without users relying on wizards or guidance after they have become familiar with it.
2	What is the appropriate parameter value of domain decomposition?	Here, we applied a value within the recommended range given in the manual for ADVENTURE_Solid. For the range, refer to the manual. For the CG method, use caution. (Section 7.4)
3	After inputting a density value in the “Setting for node density” dialog, focus cannot be moved by means of the Tab key.	To move focus from the area where to input a value (TextField), press the Tab key while pressing Ctrl. Also, to move focus in the reverse direction, press the Tab key while pressing Shift + Ctrl.
4	Zoom cannot be used after switching projection method to orthographic view in Visualtool.	This is the specification for the orthographic view since its view point is infinite. However, this may be possible by changing field of view, etc. The issue is under investigation.

5	Is there any problem when a caution like “harmful script is included” is displayed on surface patch generation, boundary condition setting or result display?	There is no risk although the following three scripts are used for start-up of ADVENTURE modules. ADVENTURE_TriPatch.js msh2pch.js hddmmrg_all.js

Appendix.1. Format of Configuration File

A1.1 The Common Configuration File

The common configuration file is a configuration file common to all the users of Agent. Since there is no GUI tool for modification of the configuration file currently, it must be modified by means of a text editor.

The common configuration file is located at “installation folder of the program□iAgent.conf”. Setting items in the current version are as follows.

```
*DefaultAnalysis =(default analysis type)
*DefaultParallel = (default method to use cluster. The first letter
                    should be in lower case.)
*DefaultMesh     = (default method of analysis model creation)
*DefaultModel    = (default input geometry model type)

TriPatch.PatchGenerator = cscript.exe (relative path of ADVENTURE_TriPatch.js)
TetMesh.PatchMerger    = (relative path of mrpach.exe)
TetMesh.PatchCorrector = (relative path of advtmesh9p.exe)
TetMesh.MeshGenerator  = (relative path of advtmesh9m.exe)

BCtool.SurfaceGroup     = cscript.exe (relative path of msh2pch.js)
BCtool.MakeFEMInput     = (relative path of makefem.exe)

Metis.DomainDecomposer  = (relative path of adventure_metis.exe)

Solid.Solver            = (relative path of advsolid-s.exe)
Solid.HddmMerg      = cscript.exe (relative path of hddmmrg_all.js)

VRMLViewer       = (absolute path of an executable for patch/ mesh viewer)
[The rest is omitted as they cannot be changed.]
```

Caution: Lines than begin with # or empty lines are ignored as comments.

Under the current version, supported “analysis type”, “input geometry model type” and “method of analysis model creation” are as follows:

Analysis type
- `Solid.LinearElastic` (linear elastic stress analysis)

Input model type
- `IGES` (loading an IGES file)
- `Patch` (loading a surface patch file)
- `LoadMesh` (loading a mesh file)

Method of analysis model creation
- `Tetra` (first order tetrahedron)
- `Tetra2O` (second order tetrahedron)
- `ReadMeshFile` (reading a mesh file)

To specify LoadMesh as the input model type, be sure to specify ReadMeshFile as the method of analysis model creation. If other combination is used, malfunction will occur.