

# **ADVENTURE\_iAgent**

**Integrated user support agent for ADVENTURE**

**Version: Beta 0.3**

**User's Manual**

April 5, 2002

**ADVENTURE Project**

## Contents

1. Overview .....	3
2. Operational Environment .....	3
3. Installation and Setup.....	3
4. Interconnections between iAgent, User and ADVENTURE System.....	4
4.1. Services Supported by iAgent .....	5
4.2. Graphical User Interface.....	5
4.3. Operational Order.....	7
4.4. Prevention of Unnecessary Operations .....	7
4.5. About User Control.....	7
5. What is “Analysis Case”? .....	8
6. Stress Analysis.....	9
6.1. Confirmation of User’s Objectives .....	9
6.2. Creation of Mesh and Patch .....	13
6.3. Setup of Material Properties and Boundary Conditions .....	16
6.4. Execution of Analysis Solver .....	19
6.5. Displaying of Analysis Results.....	26
7. Thermal Analysis.....	29
7.1. Confirmation of User’s Objectives .....	29
7.2. Creation of Patch and Mesh .....	33
7.3. Setup of Material Properties and Boundary Conditions .....	35
7.4. Execution of Analysis Solver .....	38
7.5. Displaying of Analysis Results.....	44
8. Thermal Stress Analysis.....	47
8.1. Confirmation of User’s Objectives .....	47
8.2. Reading of Mesh Data File .....	51
8.3. Setup of Material Properties and Boundary Conditions.....	52
8.4. Execution of Analysis Solver .....	55
8.5. Displaying of Analysis Results.....	60
Appendix 1. Format of Setup File.....	62
A1.1. Common Setup File .....	62
A1.2. User Setup File .....	64

## 1. Overview

ADVENTURE\_iAgent is a unified agent program, which helps the user to perform analyses in a way using the ADVENTURE system. ADVENTURE\_iAgent has the following features.

- 1). All commands used in the ADVENTURE System can be inputted using the Graphical User Interface (GUI).
- 2). Depending on analysis objectives, the user can create a plan and iAgent will navigate the user step by step through the analysis (from creation of patch up to displaying of analysis results).
- 3). iAgent can display the real-time operational conditions of each host of the cluster computers.
- 4). iAgent is compatible with the ADVENTURE\_Solid solver and the ADVENTURE\_Thermal solver.

## 2. Operational Environment

iAgent program can operate in the following environments.

- 1). Operating system

*UNIX, Linux*

- 2). Compiler

Java (JDK 1.2 or higher or JRE 1.2 or higher)\*

\* In accordance with Java characteristics, recompilation of iAgent program for different OS is unnecessary. The JDK or JRE are necessary as the operational environments.

## 3. Installation and Setup

After decompression of files from the tar+gz archive, read the installation instructions, which are provided in the file "INSTALL" located in the top directory.

After installation of necessary files, setup the common files in accordance with instructions given in *Appendix 1*.

Moreover, before you use the ADVENTURE\_iAgent, check whether the paths to each module of ADVENTURE system are specified correctly. As minimum requirements for setup of paths in this version, the following commands should be made executable:

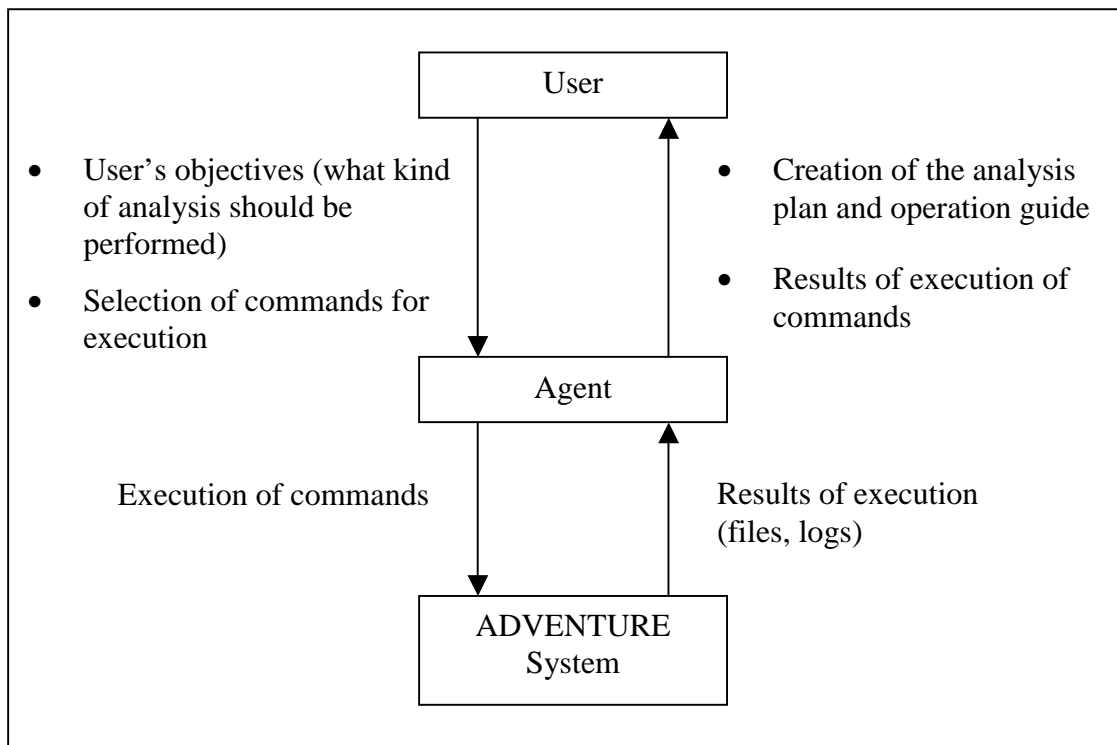
- faceOfMesh (the ADVENTURE\_BCtool module)
- makepch (the ADVENTURE\_BCtool module)

#### 4. Interconnections between iAgent, User and ADVENTURE System

The most of ADVENTURE system modules are controlled by commands, which should be inputted from the command line. In this case, the user should have knowledge not only about the flow of analyses, but also about the methods of using of the commands.

iAgent creates a plan of analysis according to the objectives set by the user, recalls the necessary system modules to make the analysis according to the calculation flow, and displays the run-time information to the user. It is designed to decrease the technical complexity and help the user to make the analysis flow smooth.

The interconnections between iAgent, the user, and the ADVENTURE system are shown in *Fig. 1*.



*Fig. 1. Interconnections between iAgent, User, and ADVENTURE System*

## 4.1. Services Supported by iAgent

The services supported by iAgent are listed below.

- iAgent shows the analysis flow in accordance with the analysis selected.
- iAgent provides the input interface for each of parameter required for analysis.
- iAgent executes the ADVENTURE modules according to the analysis flow.
- iAgent displays the necessary operational flow for the AdvBCtool and AdvVisual modules.
- iAgent can terminate and restart the analysis at any stage.
- iAgent provides information on operational status for every CPU (host) of parallel-connected computers (CPU usage, memory usage).

The services, which are not supported by the current version of iAgent, are listed below.

- Termination and restart of calculations done by the solvers
- Reconstruction of the analysis plan resulted by out-of-plan operations
- Cancellation of operations
- Self-restart of the previously terminated analysis

## 4.2. Graphical User Interface

After starting of the iAgent program, the window shown in *Fig. 2* will appear on the screen.



*Fig. 2.*

If you press the “Start” button, the windows of iAgent’s initial display will appear on the screen (*Fig. 3*).

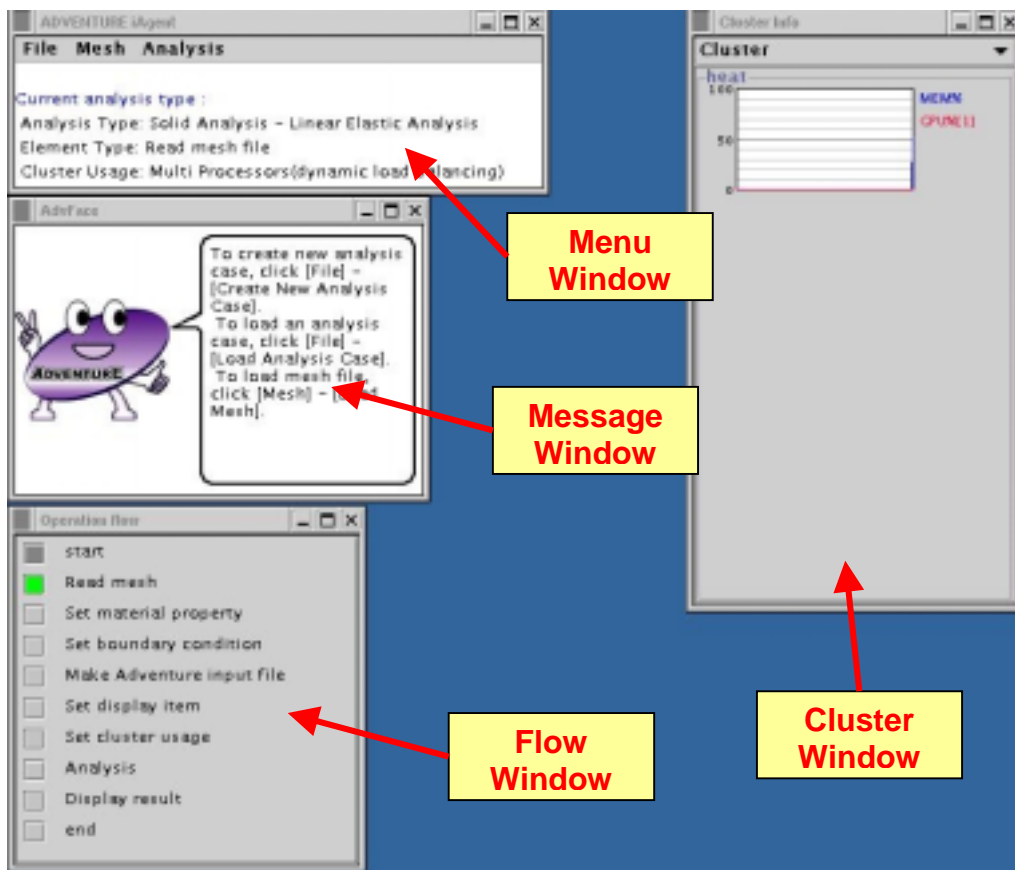


Fig. 3. Initial Display

a). *Menu window*

The Menu window is used to control the flow of analysis. The current type of analysis is displayed on the bottom part of the window. Information about the menu items is provided in *Chapter 5* of the current Manual.

b). *Message window*

The Message window is used to display help information about the concrete operational methods and currently executed commands.

c). *Flow window*

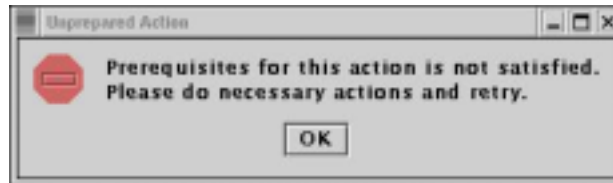
The Flow window is used to display the current plan of operations. If the button located next to the item is clicked, the outline of operation done by the item will be displayed in the Message window.

d). *Cluster window*

The Cluster window is used to display the run-time information on usage of host's resources by graphs. The blue line shows the memory usage and the red line shows the usage of CPU.

### 4.3. Operational Order

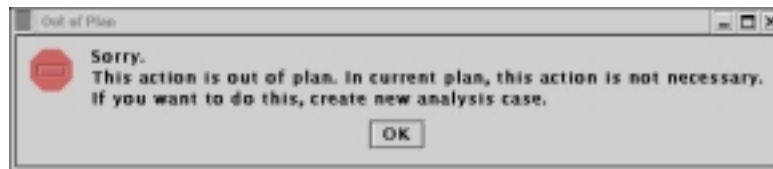
The operations displayed in the Flow window require the results of previously done operations. If some operations have been missing or done incompletely, the message shown in *Fig. 4* will be displayed on the screen. In this case, check the messages displayed in the Flow window and complete the necessary operations.



*Fig. 4*

### 4.4. Prevention of Unnecessary Operations

The analysis plan includes all necessary operations, and if you try to perform some operations, which are not included in the analysis plan, iAgent will show you the following message (*Fig. 5*). If this window appeared on the screen (the operation you performed is not necessary) but the operation is necessary, the system will respond in non-appropriate selection of analysis type. In this case, make a new Analysis Case and select the analysis plan, which would satisfy your requirements. For more information about the Analysis Case, refer to *Chapter 5* of the current manual.



*Fig. 5*

### 4.5. About User Control

In the current version of the program, the history of operations done by the user is accumulated only for iAgent's inner needs. The support for undo operations according to the operation history will be provided in the future versions of the program.

## 5. What is “Analysis Case”?

iAgent controls analysis operations as units named here the “Analysis Case”. In general, sometimes it calls “Project”. The following items of the “Analysis Case” are controlled:

- The type of analysis with the operational plan prepared for the current analysis
- Operations done by the user
- Setup values inputted by the user during operations
- The names of files, which are automatically created by each of the ADVENTURE system module

iAgent has features of saving of the “Analysis Case” and, by reading the previously saved data, the conditions of analysis (at the time of saving) can be restored. The files are saved and read automatically by each of the ADVENTURE system module. Using this feature, the user can easily stop and restart the analysis plan.

To save the “Analysis Case”, select the submenu “Save Analysis Case” of the menu “File”. The window for file selection will appear on the screen. Choose the directory name and the filename for saving of data. If the filename is specified without a file extension, the file extension “. iag” will be added to the filename automatically.

In this case, the files automatically created by each ADVENTURE module will be stored in the subdirectory called “(filename).files”, which will be created in the same directory with the “filename.iag”. For example, if the “Analysis Case” with the name “analysis.iag” is stored in the directory “/home/user/data”, the directory “home/user/data/analysis.files” will be automatically created and the files for each ADVENTURE module will be copied in this directory automatically.

To read the “Analysis Case”, select the submenu “Load Analysis Case” from the menu “File”, and specify the data filename (.iag). The information on the analysis plan and the setting parameters saved at the specified moment will be read and the analysis can be restarted from that point.

If the analysis was saved once, you can save the analysis specifying the filename, which is differ from the filename of saved analysis. To perform such operation, select the submenu “Save Analysis Case As” from the menu “File” and specify the filename in the dialog box.

Moreover, the file .iag has a format, which is generated by using the class “workingMemory.AnalysisCase” supported by Java serialized structure. The use of another languages is not considered.



## 6. Stress Analysis

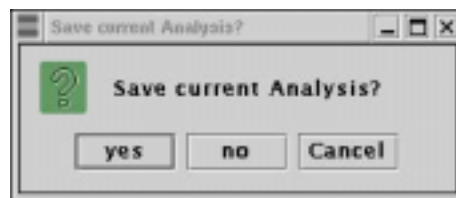
After starting of iAgent, a new “Analysis Case” is prepared automatically taking into account the user’s settings (the type of analysis). The setup for basic analysis types is explained in *Appendix 1* of the current Manual.

The type of executing analysis is displayed in the lower part of the Main window. If you need to start the analysis, which is different from the currently running analysis, you should create a new “Analysis Case” and select the type of analysis.

The analysis flow will be explained on example of Stress analysis.

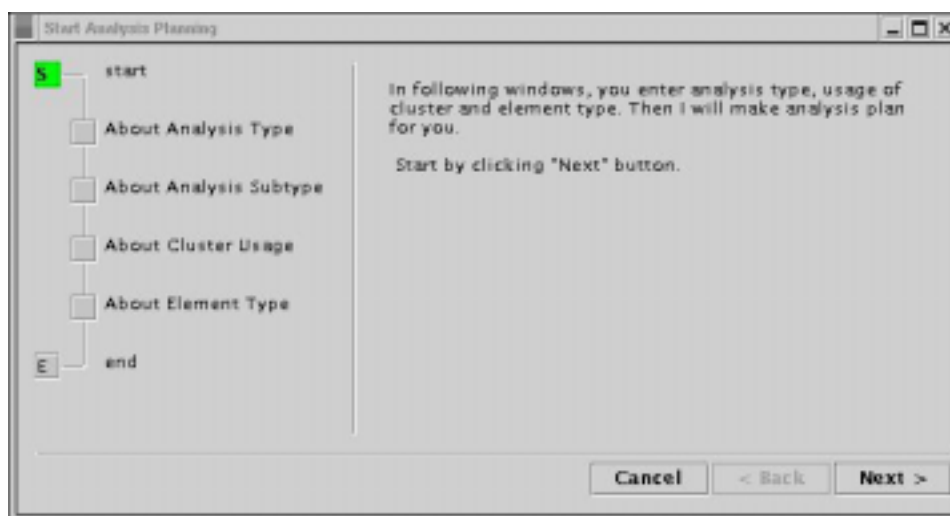
### 6.1. Confirmation of User’s Objectives

To start new analysis, select the submenu “Create New Analysis Case” in the menu “File”. The iAgent will ask you whether it is necessary to save the current analysis or not (see *Fig 6*). If you need to proceed with the current analysis or if you need to save the current data, select the button “Yes”. Otherwise, select “No”.

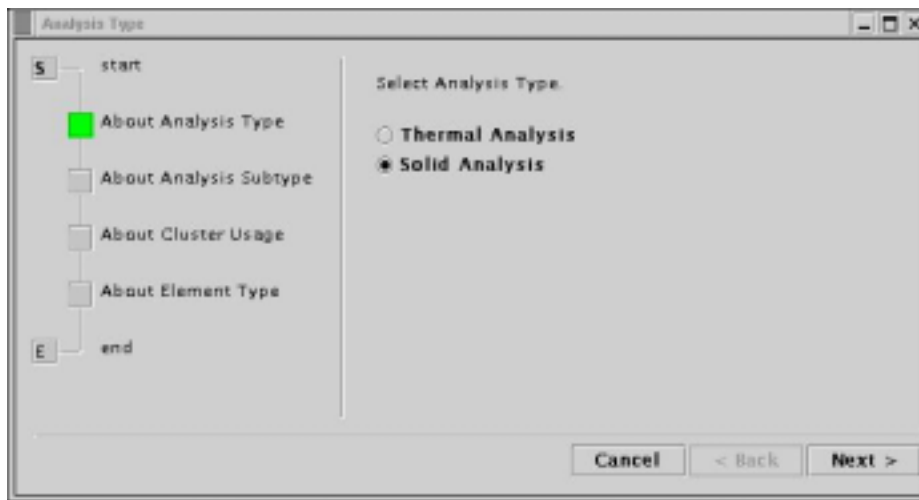


*Fig. 6*

To create new “Analysis Case”, iAgent will start to ask the user some questions to navigate him through analysis. Go to the next analysis stage by clicking on the “Next” button.

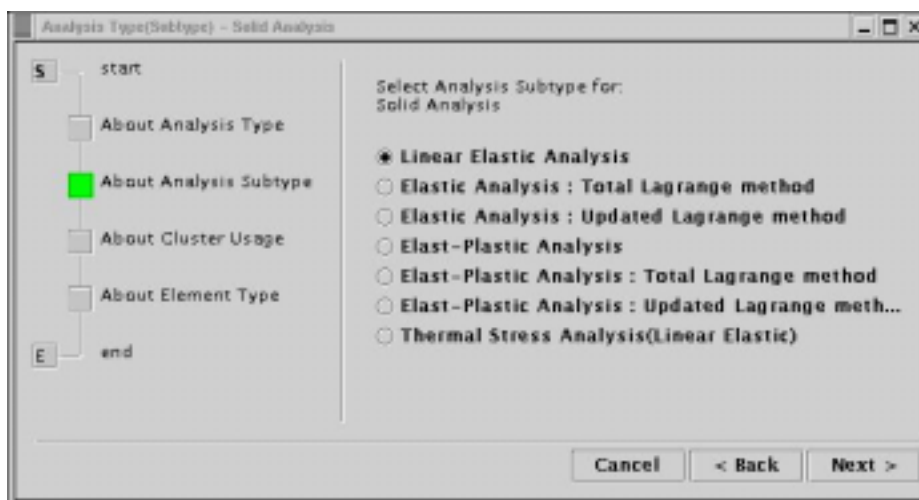


*Fig. 7*



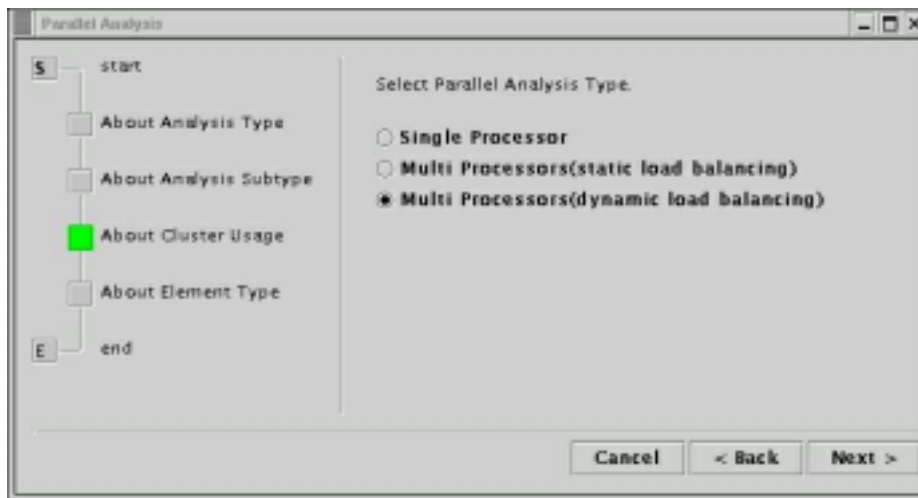
*Fig. 8*

At the beginning, select the type of analysis as it is shown in *Fig. 8*. Here, the Solid Analysis is selected. Next, select the type of Solid analysis supported by iAgent (see *Fig. 9*). Supported analyses are listed in *Fig. 9*.



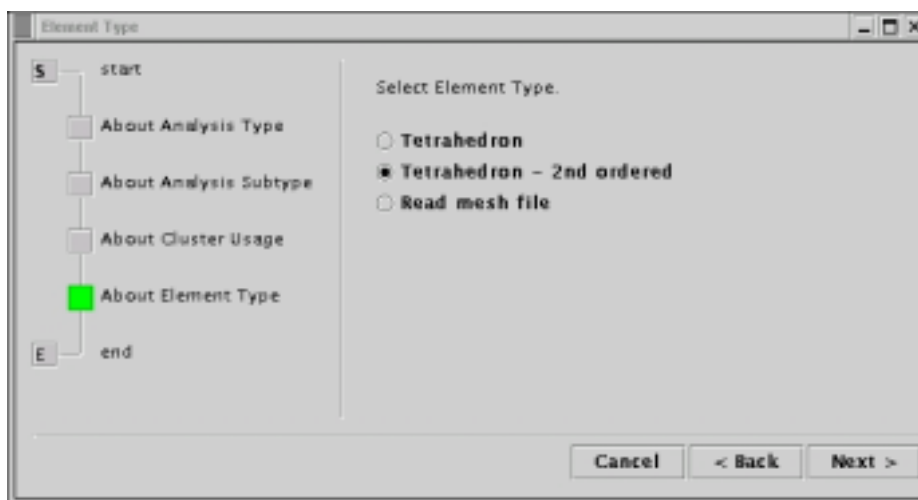
*Fig. 9*

Next, select the network nodes of cluster computer as it is shown in *Fig. 10*. Here, the load balance is done by the way of dynamic load distribution (dynamic load balancing).



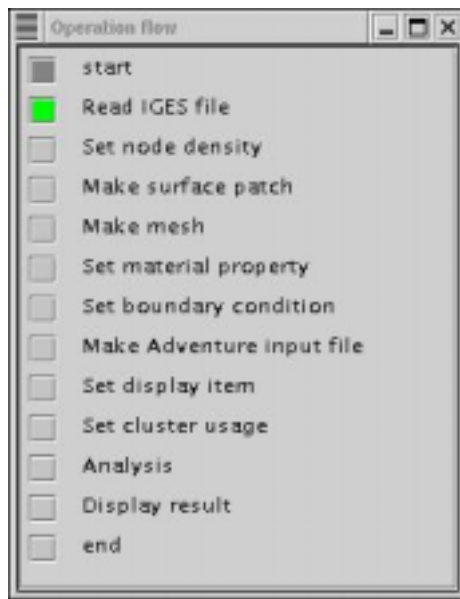
*Fig. 10*

Select the type of element that will be used for the finite element model as it is shown in *Fig. 11* (the quadratic tetrahedral element is selected).



*Fig. 11*

According to the made selections, the analysis plan will be created automatically and the sequence of necessary operations will be shown in the Flow window (*Fig. 12*). The analysis will proceed according to the created analysis plan.



*Fig. 12*

## 6.2. Creation of Mesh and Patch

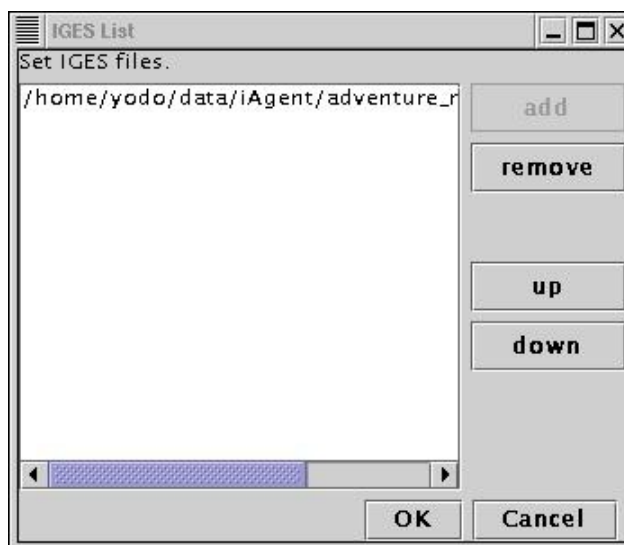
iAgent creates mesh data from CAD data using the programs ADVENTURE\_TriPatch and ADVENTURE\_TetMesh.

First, specify the CAD model. Select the submenu “Mesh” from the menu “Select IGES File” of the Main window. The window shown in *Fig. 13* will appear on the screen.



*Fig. 13*

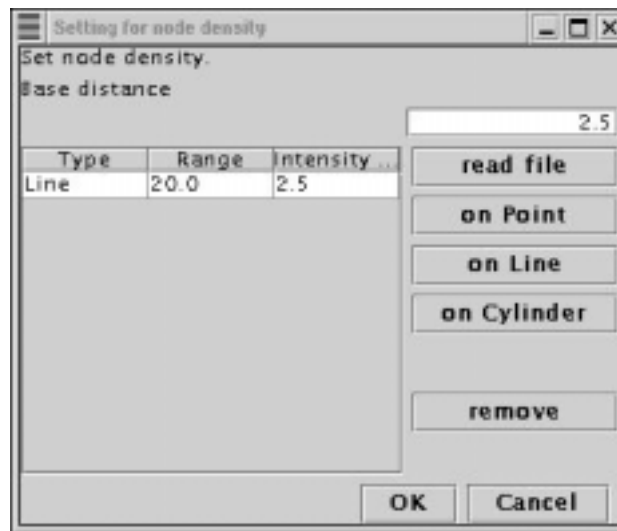
Click on the “Add” button; the dialog for file selection will appear. Select the name of the CAD model from the list as it is shown in *Fig. 14* and click on the button “OK”.



*Fig. 14*

To deselect the CAD model, select the submenu “Mesh” from the menu “Select IGES File”. The previously selected CAD data will appear in the file list. Select the model’s filename and click on the button “Remove”. You can add a new model by clicking on the button “Add” and make operations described above.

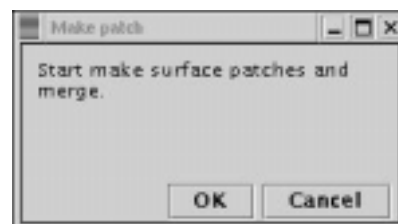
Next, set the mesh density. Choose the submenu “Set Node Density” from the menu “Mesh” as it is shown in *Fig. 15*.



*Fig. 15*

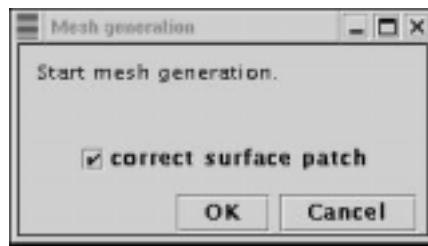
There are 3 types of settings of the node density: set by points, set by segment, and set by cylinder. Selection of the necessary parameters is explained in the User’s Manual supplied with the ADVENTURE\_TetMesh module. The necessary node density can be also read from file.

At the next stage, the surface patch data will be prepared. To create the surface patch, select the submenu “Make Surface Patch” of the menu “Mesh” and click “OK” button (Fig. 16).



*Fig. 16*

At the next stage, the mesh data will be prepared. Select the submenu “Make Mesh” of the menu “Mesh”. The window shown in *Fig. 17* will appear on the screen. The option “correct surface patch” is used to correct the surface patch before creation of the mesh (this option is set to ON by default). Click OK to start the creation of mesh.

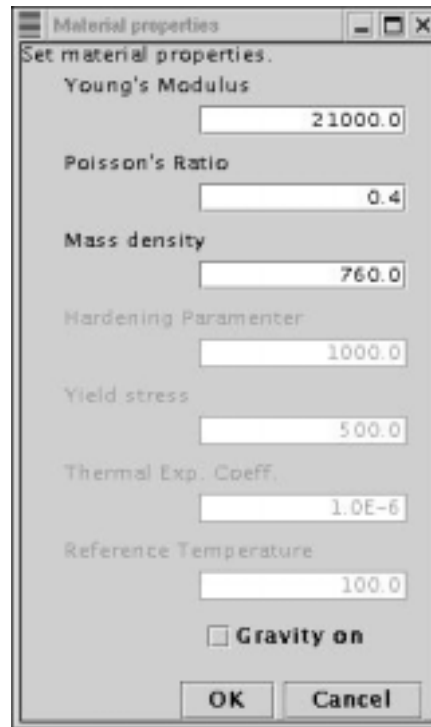


*Fig. 17*

After creation of the mesh and patch data, it is necessary to set the material properties and boundary conditions.

### 6.3. Setup of Material Properties and Boundary Conditions

To set the material properties, select the submenu “Solid” from the menus “Set material Property” and “Analysis”. The dialog box of *Fig. 18* will appear on the screen.



*Fig. 18*

The data, which are not necessary for the current analysis will be shown by dim gray color. To set the body force, check the option “Gravity on” in the bottom of the menu dialog.

The next stage is a setup of boundary conditions. The boundary conditions will be set using the module ADVENTURE\_BCtool, which will be started by iAgent. iAgent will guide you through the process.

First, select the submenu “Set Boundary Conditions” form the menu “Analysis”. The window shown in *Fig. 19* will appear on the screen.



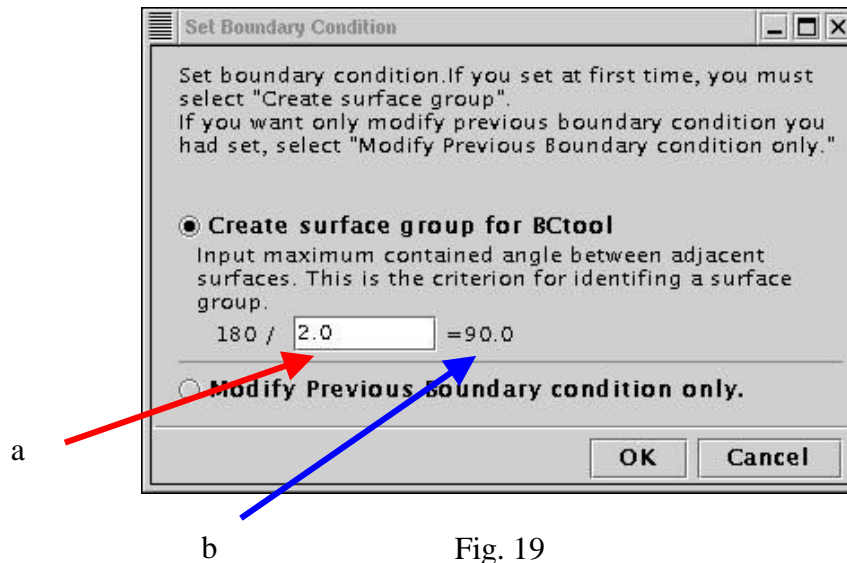


Fig. 19

BC tool will divide the mesh surface by a number of groups, and the boundary conditions can be set for these groups. The mesh data are grouped using the angle parameter between adjacent surfaces as a criterion (see the User's Manual for ADVENTURE\_BCtool for details). Fig. 19 (a) shows the input dialog for the input of adjacent angle parameter and the arrow (b) shows the angle between the adjacent surfaces.

Input the necessary parameter and click the "OK" button. The mesh will be grouped automatically, and the ADVENTURE\_BCtool module will be started (Fig. 20).

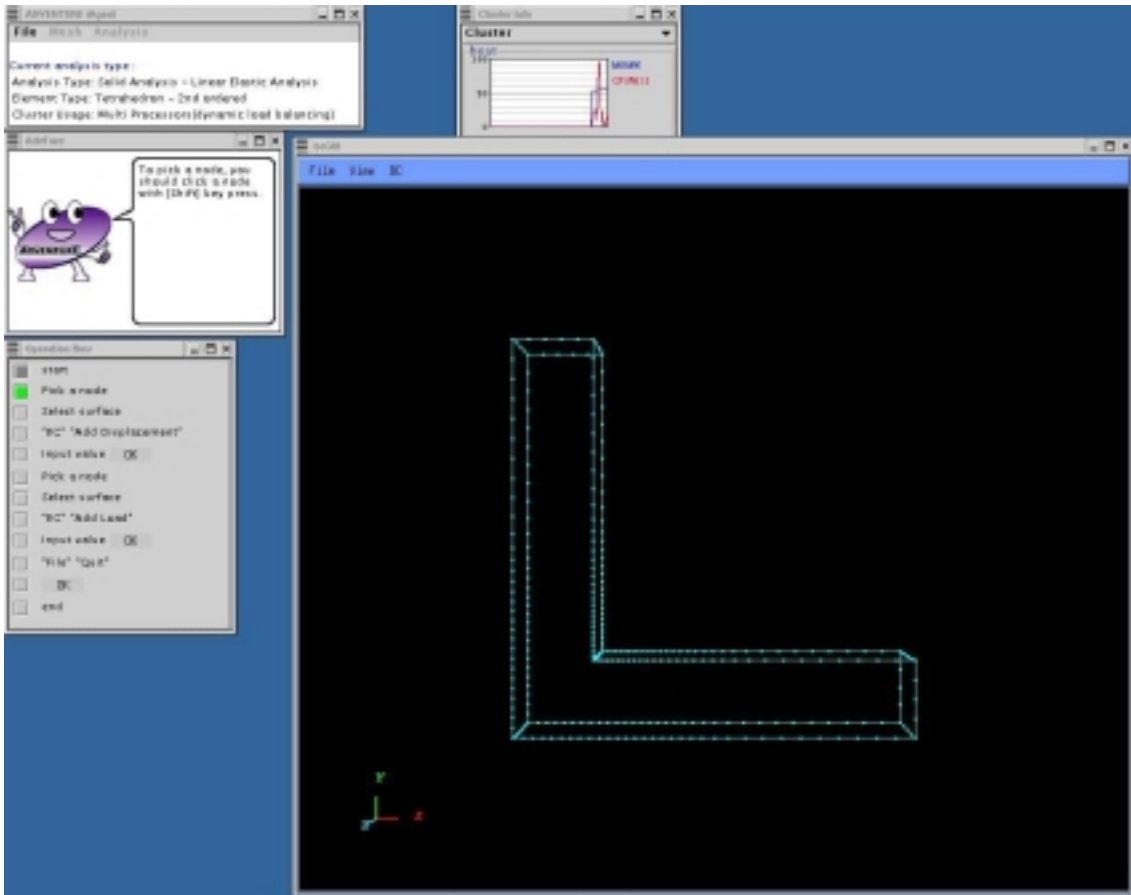


Fig. 20

At this point, the Flow window will be changed automatically to display the operation steps necessary for BCtool (Fig. 20). Refer to messages displayed in the Message window to proceed. Explanations about the usage of BCtool are provided in the ADVENTURE\_BCtool User's Manual.

After setting of the boundary conditions, exit from BCtool. The boundary conditions will be automatically saved and checked by iAgent (to save the boundary conditions by user is unnecessary).

The operation of setting of the boundary conditions and material properties will be finished after saving of the data into the entire-type input data file. To save the data, select the submenu "Convert to Input File" of the menu "Analysis".

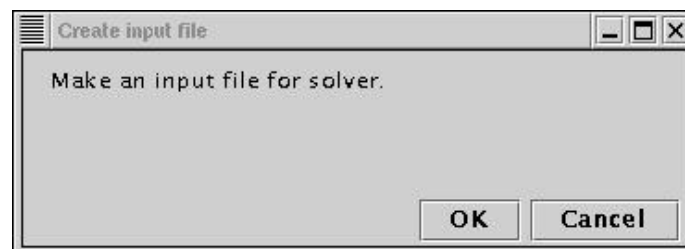
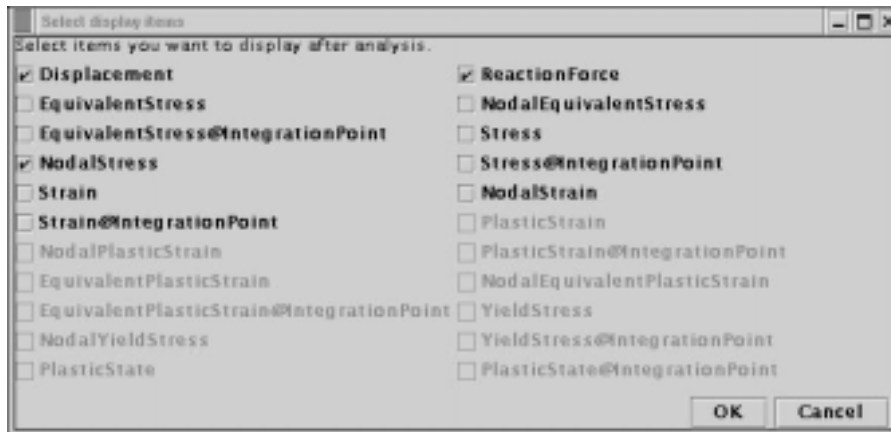


Fig. 21

## 6.4. Execution of Analysis Solver

Prior to execution of the analysis solver, it is necessary to select the parameters for output results, the names of network hosts, which will be used by solver for parallel calculations, and the incremental steps, which will be applied in case of non-linear analyses.

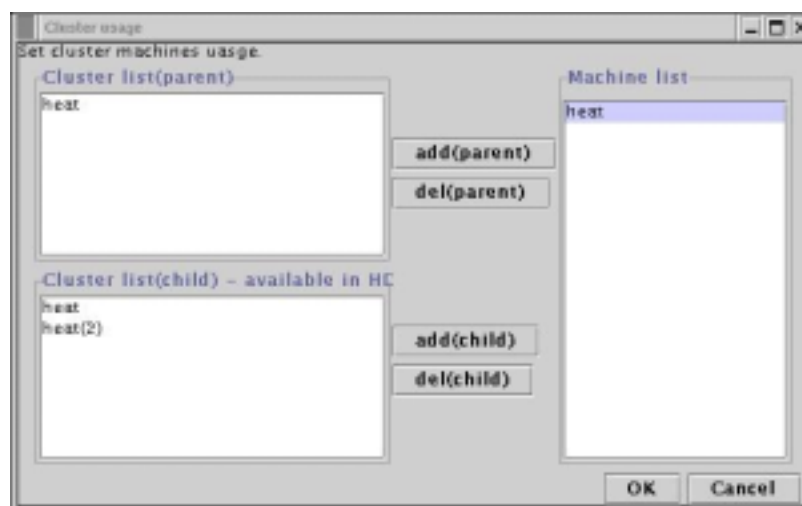
To set the output parameters, select the submenu “Solid”, the submenu “Set Display Item” of the menu “Analysis”. The window shown in *Fig. 22* will appear on the screen.



*Fig. 22*

The items possible for output in the current analysis are displayed by black font. Click on the checkboxes next to the items, which will be printed out after analysis. If the checkbox is checked, the item will be printed out.

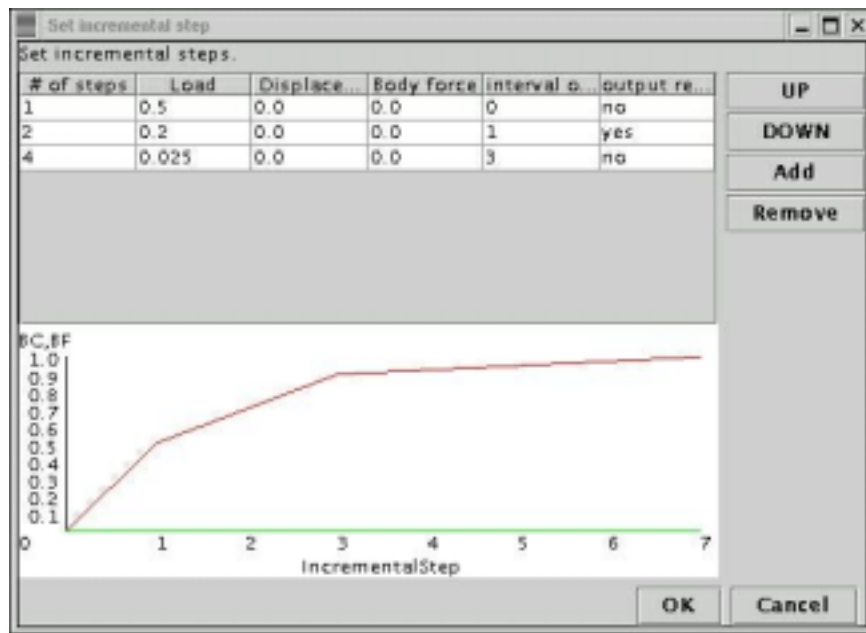
To set the usage of network nodes (only in case of parallel computing), select the submenu “Set Cluster Usage” of the menu “Analysis”. The window shown in *Fig. 23* will appear on the screen.



*Fig. 23*

Here, the *Parent* and the *Child* nodes should be specified for the case of dynamic load distribution, and only the *Parent* node should be specified for the static load distribution. Refer to the ADVENTURE\_Solid User's Manual for details.

In case of non-linear analysis, the incremental step should be set. To set the incremental step, select the submenu "Solid" of the submenu "Set Incremental Step" of the menu "Analysis". The window shown in *Fig. 24* will appear on the screen.



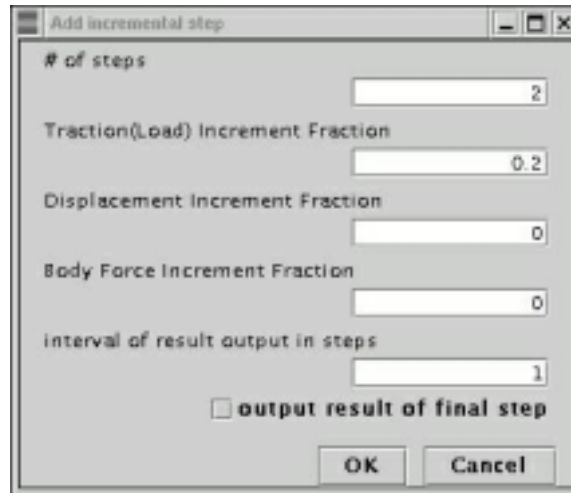
*Fig. 24*

The values of the incremental steps will be shown in the upper part of the window, and its graphical interpretation will be shown in the lower part of the window. The graph shows the multiplication factor for:

- Red : Load
- Yellow : Displacement
- Green : Body force

The BC is corresponded to the boundary conditions and the BF is corresponded to the body force.

To add new steps, click on the "Add" button. The window shown in *Fig. 25* will appear on the screen.



*Fig. 25*

In this window,

- The number of steps to be added
- The multiplication factor for load
- The multiplication factor for displacement
- The multiplication factor for body force
- The interval for intermediate output
- The number of the last step after which the output will be done

The input example is shown in Figs. 24 and 25. The value set by BCtool is assumed to be 1 and the multiplication factor shows the part of unity to be added at one step. Note that the total value of multiplication factor for one step should be less than 1. Here, it is for control of load only.

**Note** that “# of steps” is not the step number, but the number of steps added by the current setup. For example, if you need to add 500 steps, specify 500 for the “# of steps”. If the real number is inputted, the decimal numbers will be cut off and only integer numbers will be stored.

You can change the order of steps using the “UP” and “DOWN” buttons and, using the button “Remove”, you can delete the selected step.

After finishing the above-described procedures, the analysis by solver can be started. To start the solver, select the submenu “Start Analysis” of the menu “Analysis”. The window shown in *Fig. 26* will appear on the screen. The options, which can be used with the solver, can be set in this window. If no options have been specially specified, default solver options will be used.

The number of *Subdomains*, which will be treated by 1 CPU can be set in the tab menu “Domain Decompose” (Fig. 26) using 2 methods.

- The number of elements in one *Subdomain* (click on the radio button “Input “of elements/subdomain”)
- The number of *Subdomains* (click on the radio button “Input # of subdomains/part”)

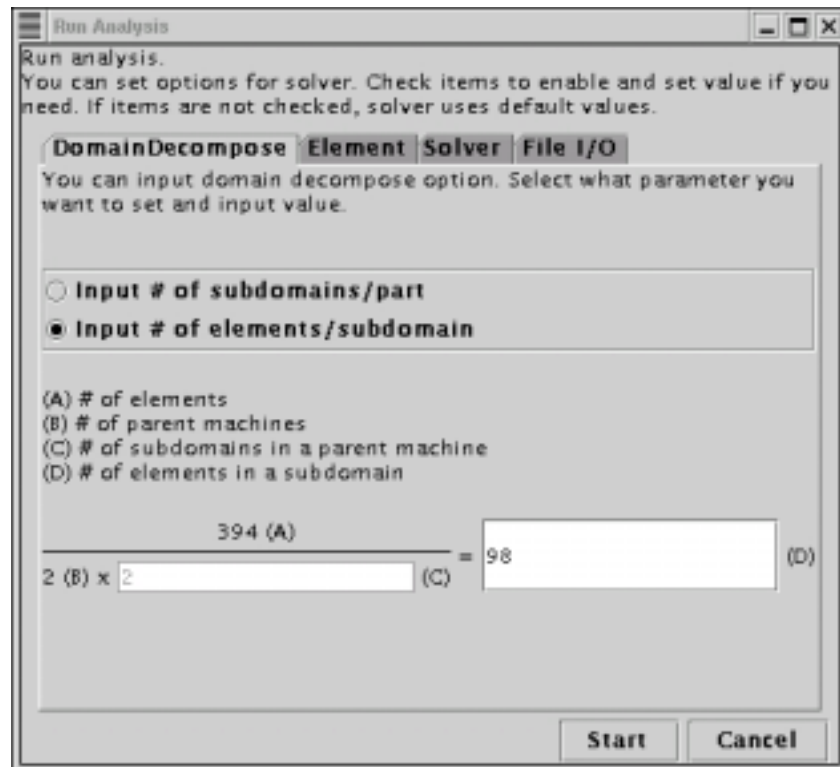
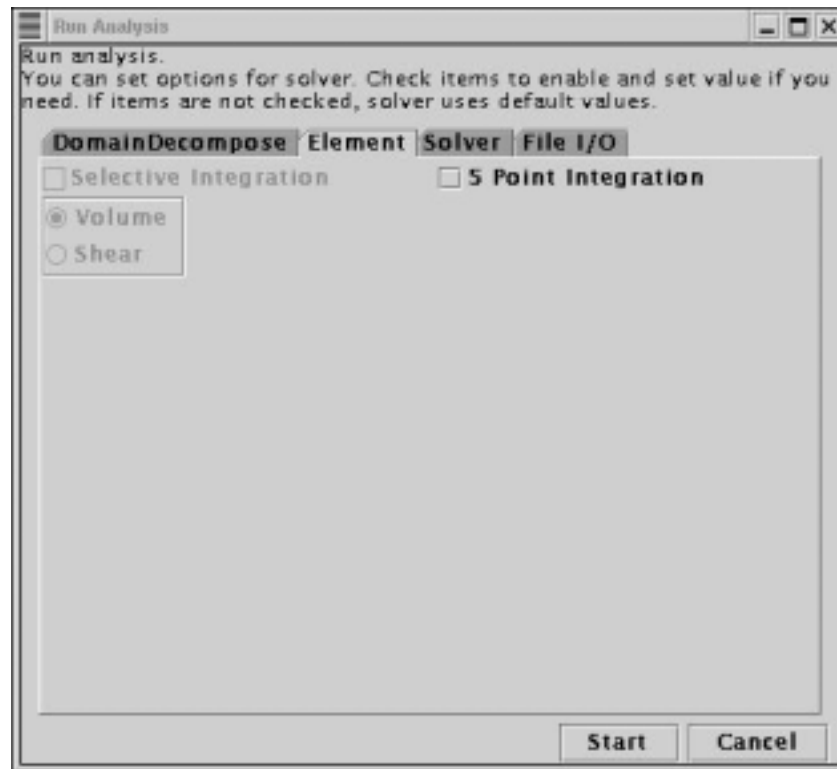


Fig. 26

The options related to elements can be set in the tab menu “Elements” (*Fig. 27*). The following 2 options cannot be used together at once. To apply these options, click on the corresponded checkboxes.

- Selective integration (volumetric strain, shear strain) (it can be set only for linear hexahedral elements)
- 5 point integration (it can be set only for quadratic tetrahedral elements)



*Fig. 27*

The solver-related options can be set in the tab menu “Solver”. To apply these options, click on the corresponded checkboxes.

- The solver type
- The tolerance for CG method
- The maximum number of loops for CG iterations
- The option for saving of the K-matrix
- The parameter for Neumann-Neumann preconditioning (only for BDD solver)
- The tolerance for iterations (Newton Tolerance) (applied only for non-linear analyses)
- The maximum number of iterative loops (applied only for non-linear analyses)

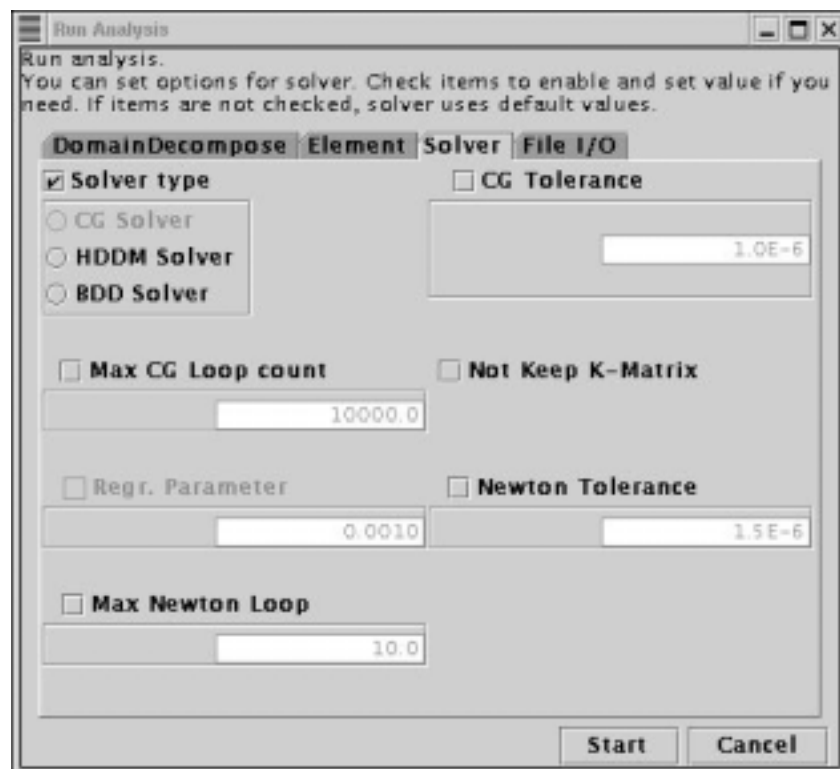
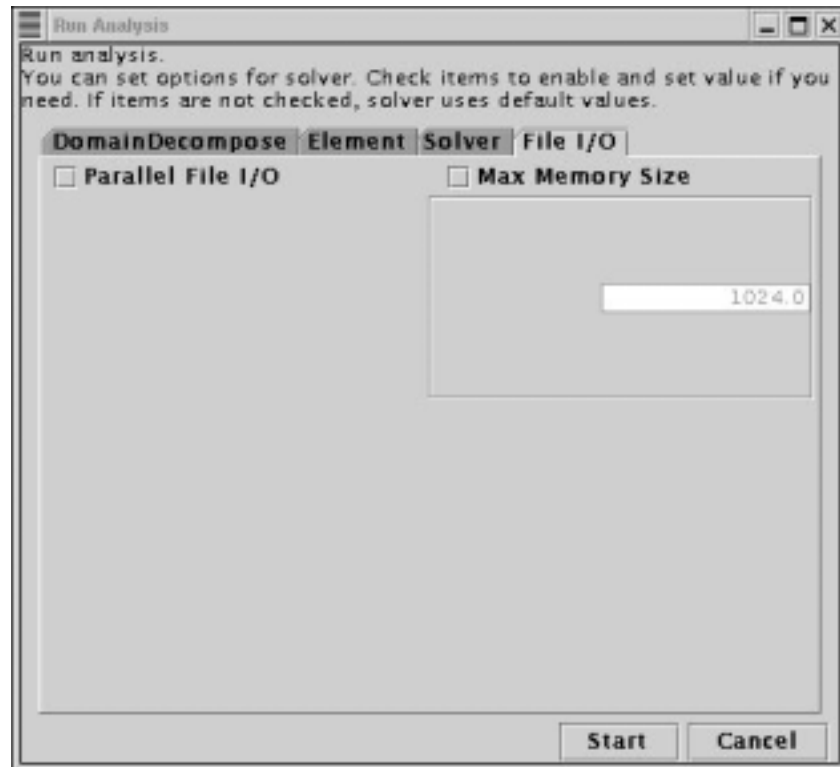


Fig. 28



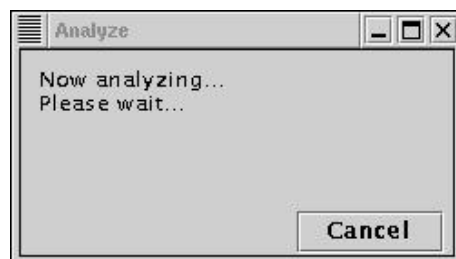
The options related to input/output can be set in the tab menu “File I/O” (*Fig. 29*). To apply these options, click on the corresponded checkboxes.

- The parallel data processing (Parallel File I/O)
- The maximum memory size used by one process (Max Memory Size)



*Fig. 29*

When the setup of all necessary options is finished, click on the “Start” button. The solver will analyze the problem after making the necessary domain decomposition. The window shown in *Fig. 30* will be displayed.

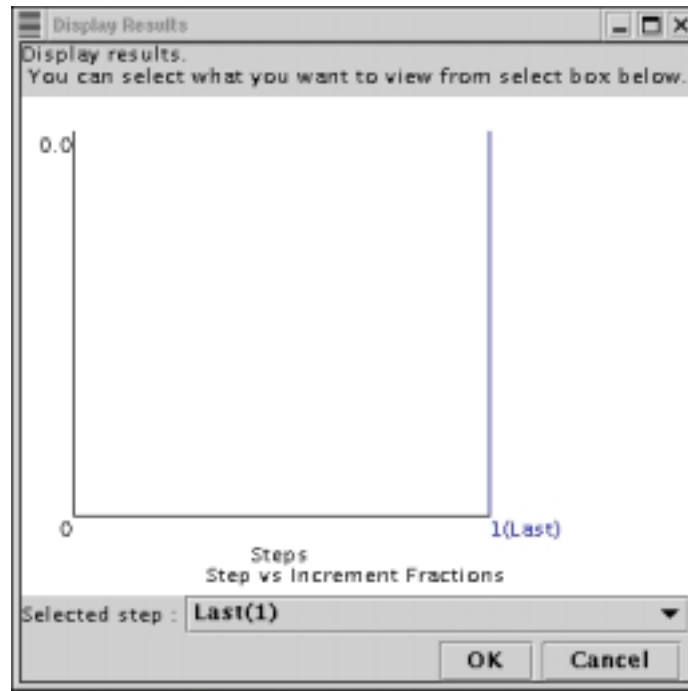


*Fig. 30*

In the current version of iAgent, the button “Cancel” does not support the function of solver termination (it will be supported in the future version of iAgent).

## 6.5. Displaying of Analysis Results

The results of analysis can be visualized using the module ADVENTURE\_Visual. To display the results, select the submenu “View Result” of the menu “Analysis”. The window shown in *Fig. 31* will appear on the screen.



*Fig. 31*

Since, the linear analysis was selected here, ignore the scale of the graph and click “OK” button. The module AdvVisual will be started automatically (*Fig. 34*).

In case of non-linear analysis, the window shown in *Fig. 32* will appear on the screen. The graph shows the interval steps, which have been applied during analysis. The horizontal axis corresponds to the step numbers. The step number, which results should be visualized, can be selected using the menu “Selected step” at the lower part of the window. As it is shown in *Fig. 33*, the selected step, in this case it is step No.3, is shown by the vertical blue line.

After selection of the step for visualization, click the “OK” button that will start the AdvVisual module (*Fig. 34*).

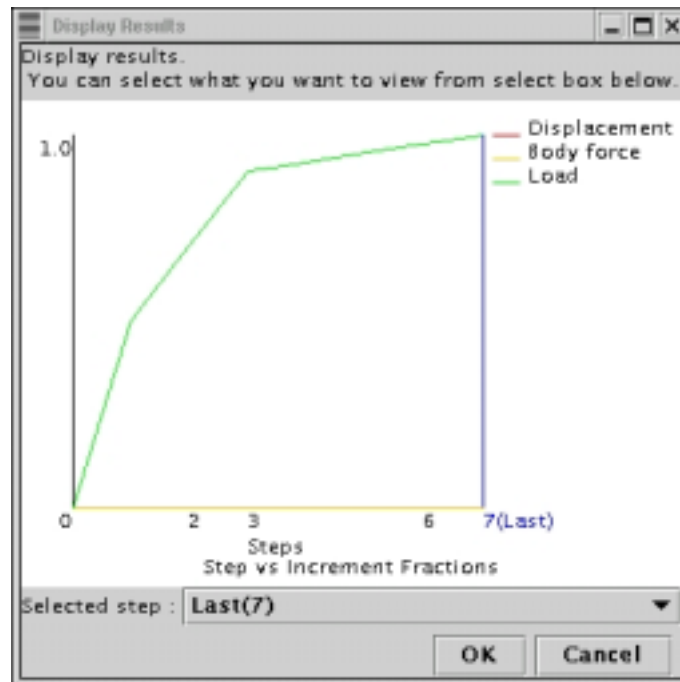


Fig. 32

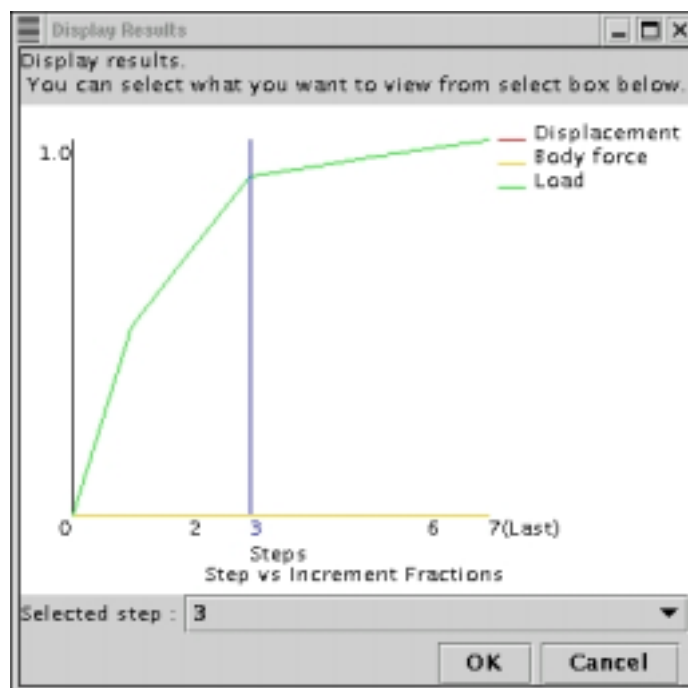


Fig. 33

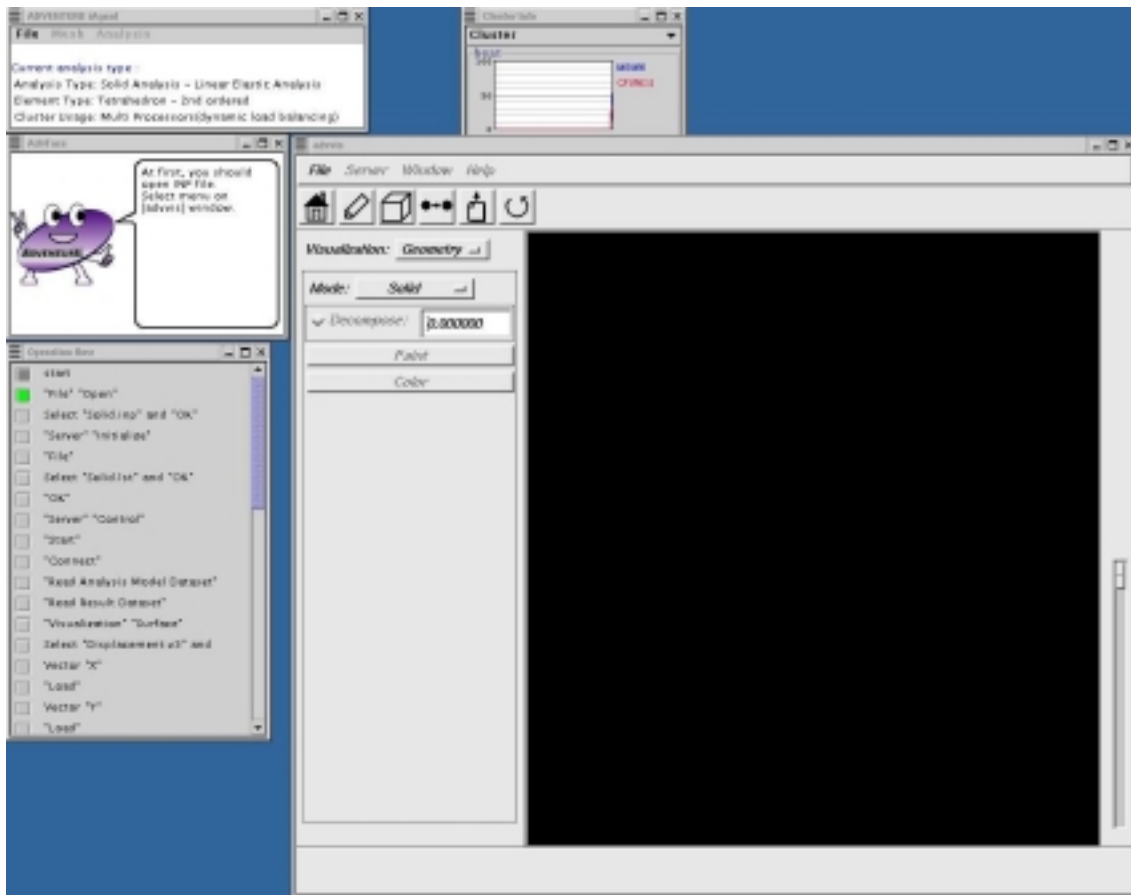


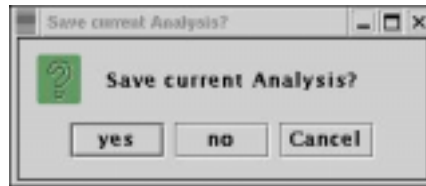
Fig. 34

After start-up of the AdvVisual module, the Flow window will be changed to display AdvVisual (as it was done for AdvBCtool). You can use the navigation messages provided in the Message window to display the visualized results.

## 7. Thermal Analysis

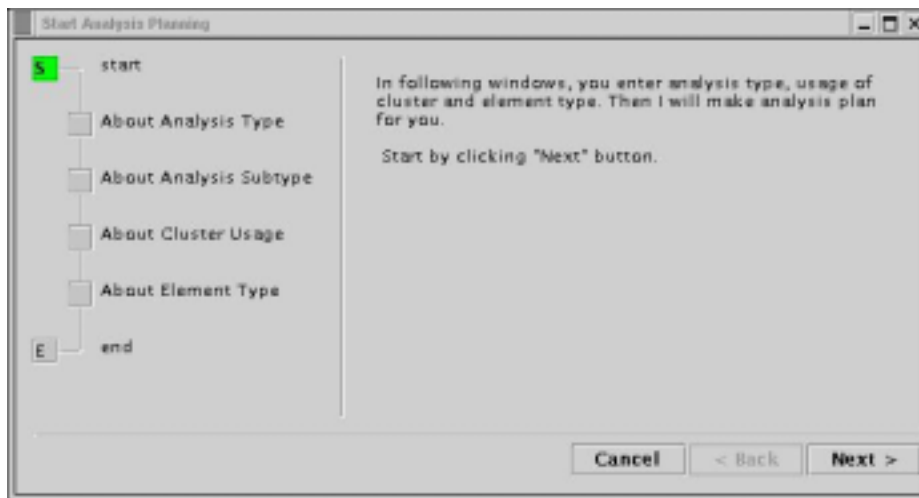
### 7.1. Confirmation of User's Objectives

To start new analysis, select the submenu “Create New Analysis Case” in the menu “File”. The iAgent will ask you whether it is necessary to save the current analysis or not (see *Fig 35*). If you need to proceed with the current analysis or if you need to save the current data, select the button “Yes”. Otherwise, select “No”.



*Fig. 35*

To create new “Analysis Case”, iAgent will start to ask the user some questions to navigate him through analysis. The window shown in *Fig. 36* will appear on the screen. Go to the next analysis stage by clicking on the “Next” button.



*Fig. 36*

At the beginning, select the type of analysis as it is shown in *Fig. 37*. Select the option “Thermal Analysis”. Two kinds of thermal analyses can be performed: the steady-state analyses, and the transient analyses (*Fig. 38*).

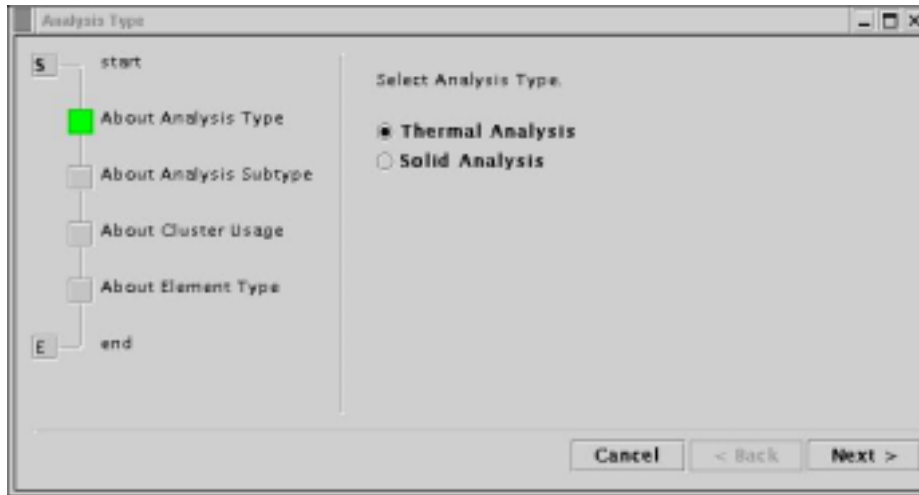


Fig. 37

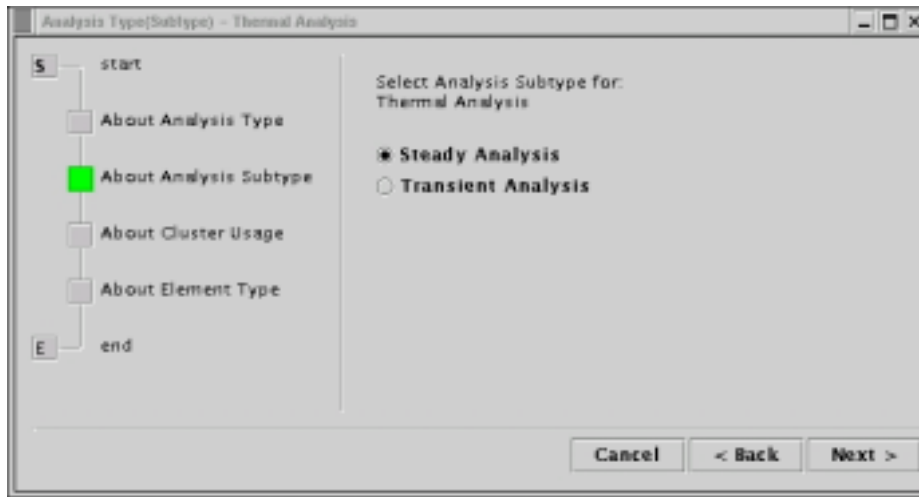
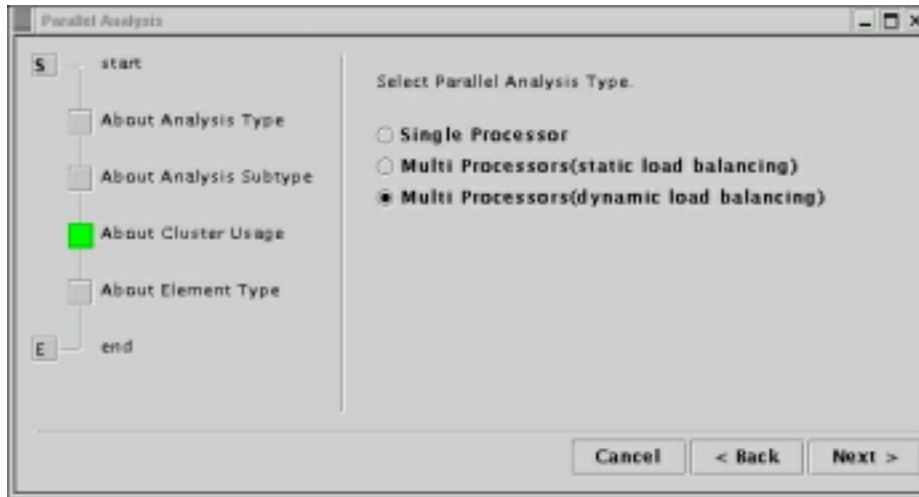


Fig. 38

Next, select the network nodes of cluster computer as it is shown in *Fig. 39*. Here, the load balance is done by the way of dynamic load distribution (dynamic load balancing).



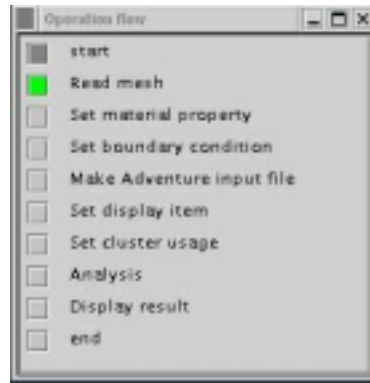
*Fig. 39*

Select the type of element that will be used for the finite element model as it is shown in *Fig. 40* (the quadratic tetrahedral element is selected).



*Fig. 40*

According to the made selections, the analysis plan will be created automatically and the sequence of necessary operations will be shown in the Flow window (*Fig. 41*). The analysis will proceed according to the created analysis plan.



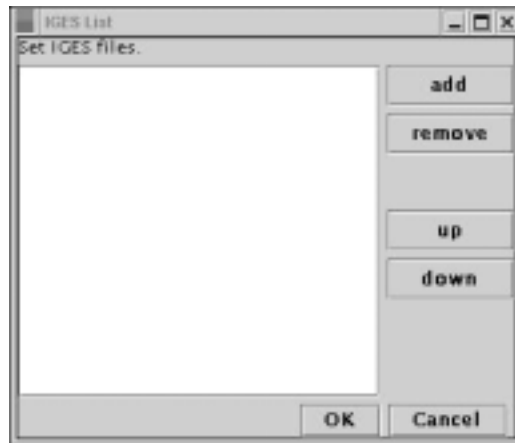
*Fig. 41*



## 7.2. Creation of Patch and Mesh

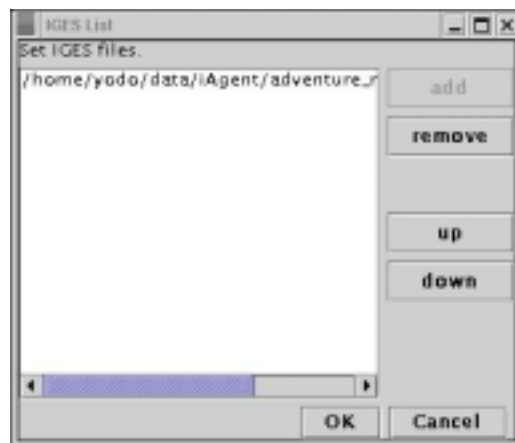
iAgent creates mesh data from CAD data using the programs ADVENTURE\_TriPatch and ADVENTURE\_TetMesh.

First, specify the CAD model. Select the submenu “Mesh” from the menu “Select IGES File” of the Main window. The window shown in *Fig. 42* will appear on the screen.



*Fig. 42*

Click on the “Add” button; the dialog for file selection will appear. Select the name of the CAD model from the list as it is shown in *Fig. 43* and click on the button “OK”.



*Fig. 43*

To deselect the CAD model, select the submenu “Mesh” from the menu “Select IGES File”. The previously selected CAD data will appear in the file list. Select the model’s filename and click on the button “Remove”. You can add a new model by clicking on the button “Add” and make operations described above.

Next, set the mesh density. Choose the submenu “Set Node Density” from the menu “Mesh” as it is shown in *Fig. 44*.

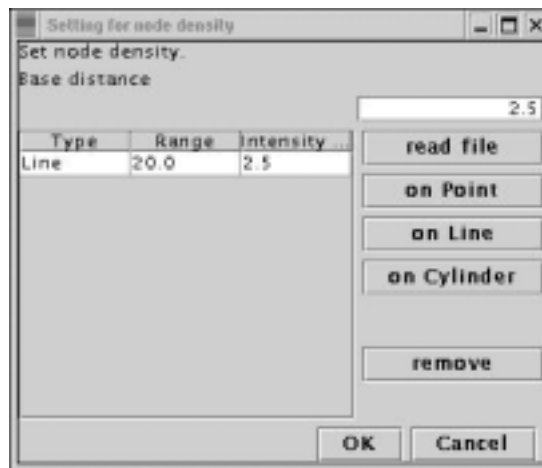


Fig. 44

There are 3 types of settings of the node density: set by points, set by segment, and set by cylinder. Selection of the necessary parameters is explained in the User's Manual supplied with the ADVENTURE\_TetMesh module. The necessary node density can be also read from file.

At the next stage, the mesh data will be prepared. Select the submenu "Make Surface Patch" of the menu "Mesh". The window shown in Fig. 45 will appear on the screen. Click "OK" to start the creation of surface patch.

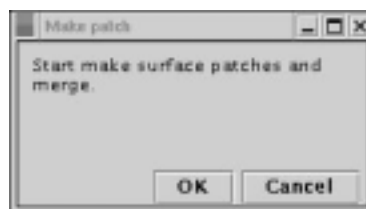


Fig. 45

Select the submenu "Make Mesh" of the menu "Mesh". The window shown in Fig. 46 will appear on the screen. The option "correct surface patch" is used to correct the surface patch before creation of the mesh (this option is set to ON by default). Click "OK" to start the creation of mesh.

After creation of the mesh and patch data, it is necessary to set the material properties and boundary conditions.

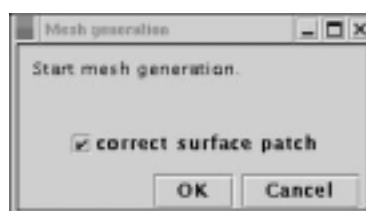
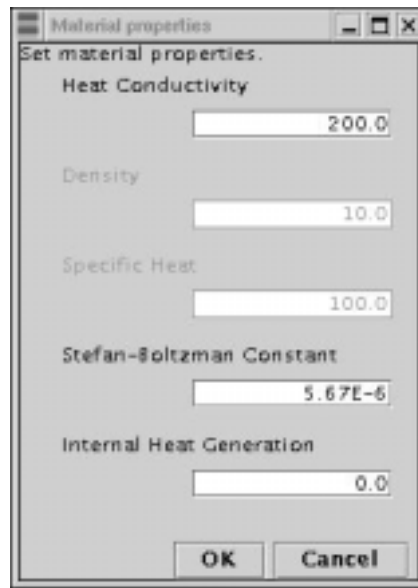


Fig. 46

### 7.3. Setup of Material Properties and Boundary Conditions

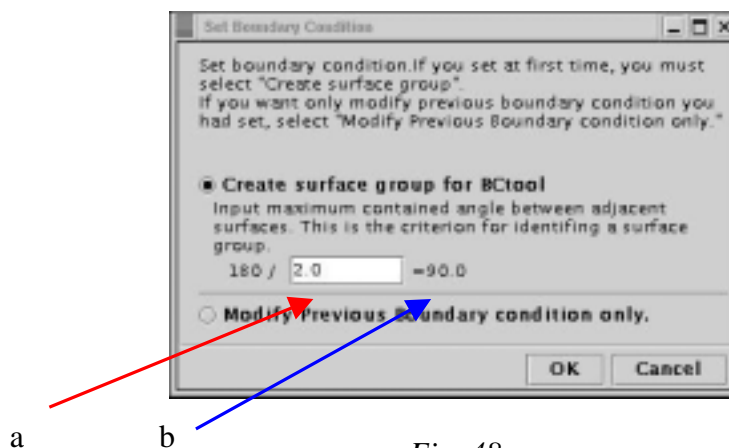
To set the material properties, select the submenu “Thermal” from the menus “Set material Property” and “Analysis”. The dialog box of *Fig. 47* will appear on the screen. The data, which are not necessary for the current analysis will be shown by dim gray color.



*Fig. 47*

The next stage is a setup of boundary conditions. The boundary conditions will be set using the module ADVENTURE\_BCtool, which will be started by iAgent. iAgent will guide you through the process.

First, select the submenu “Set Boundary Conditions” form the menu “Analysis”. The window shown in *Fig. 48* will appear on the screen.



*Fig. 48*

BC tool will divide the mesh surface by a number of groups, and the boundary conditions can be set for these groups. The mesh data are grouped using the angle parameter between adjacent surfaces as a criterion (see the User's Manual for ADVENTURE\_BCtool for details). Fig. 48 (a) shows the input dialog for the input of adjacent angle parameter and the arrow (b) shows the angle between the adjacent surfaces.

Input the necessary parameter and click the "OK" button. The mesh will be grouped automatically, and the ADVENTURE\_BCtool module will be started (Fig. 49).

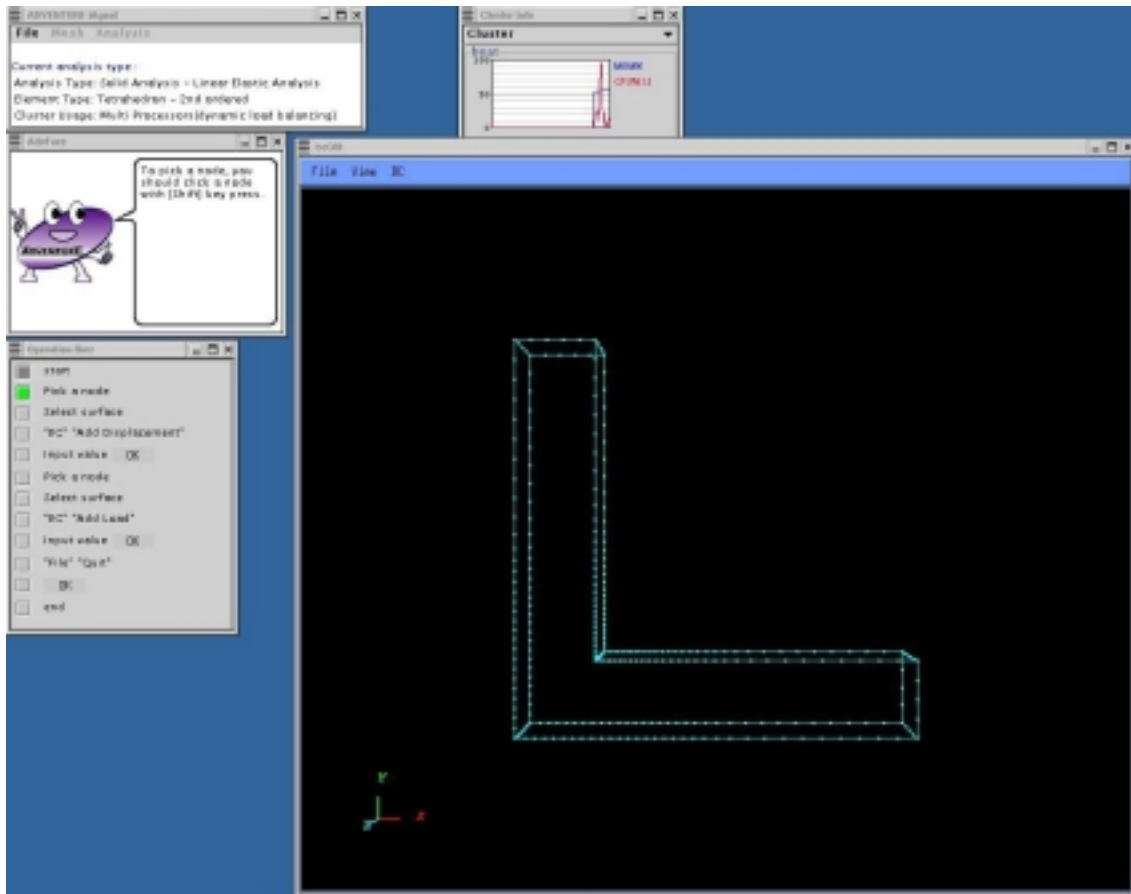


Fig. 49

At this point, the Flow window will be changed automatically to display the operation steps necessary for BCtool. Refer to messages displayed in the Message window to proceed. Explanations about the usage of BCtool are provided in the ADVENTURE\_BCtool User's Manual.

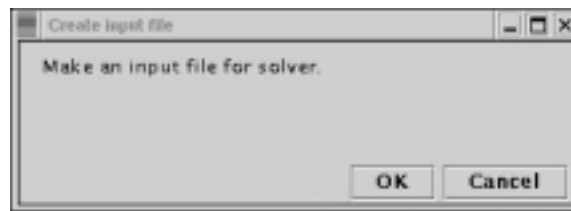
In case of heat conduction analysis, the boundary conditions can be set using a special window provided by the BCtool module. The input values later will be automatically converted to the values necessary for the heat conduction analysis. The following conversion is done:

- The X-displacement is converted to temperature
- The load is converted to the heat flux

Heat convection and heat radiation are not supported by the current version of the program.

After setting of the boundary conditions, exit from BCtool. The boundary conditions will be automatically saved and checked by iAgent (to save the boundary conditions by user is unnecessary).

The operation of setting of the boundary conditions and material properties will be finished after saving of the data into the entire-type input data file. To save the data, select the submenu "Convert to Input File" of the menu "Analysis". The message shown in *Fig. 50* will appear on the screen.



*Fig. 50*

## 7.4. Execution of Analysis Solver

To set the usage of network nodes (only in case of parallel computing), select the submenu “Set Cluster Usage” of the menu “Analysis”. The window shown in *Fig. 51* will appear on the screen. The options, which can be used with the solver, can be set in this window. If no options have been specially specified, default solver options will be used.



*Fig. 51*

The number of *Subdomains*, which will be treated by 1 CPU can be set in the tab menu “Domain Decompose” (Fig. 52) using 2 methods.

- The number of elements in one *Subdomain* (click on the radio button “Input # of elements/subdomain”)
- The number of *Subdomains* (click on the radio button “Input # of subdomains/part”)

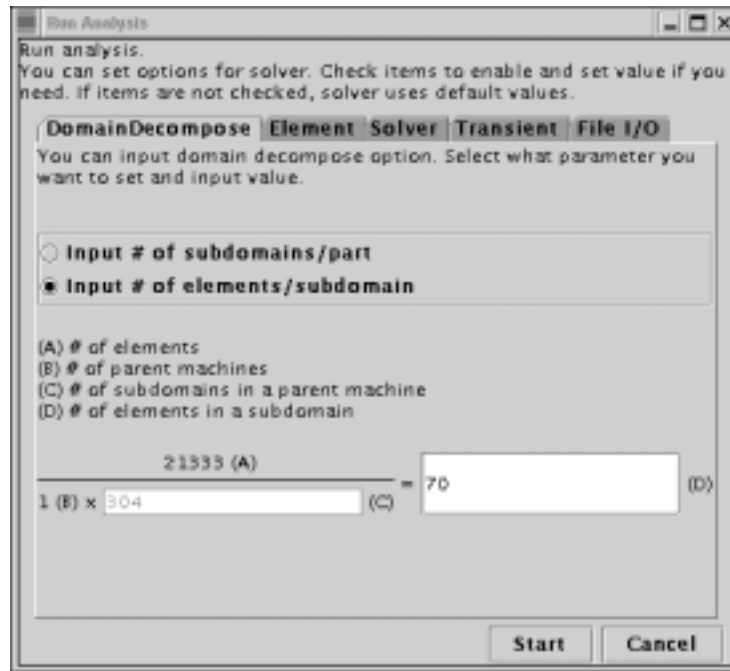
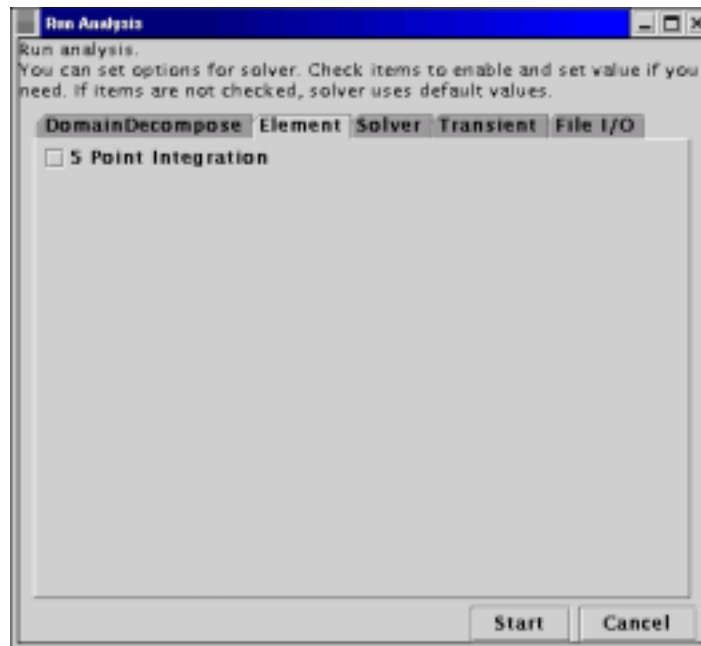


Fig. 52

The options related to elements can be set in the tab menu “Elements” (*Fig. 53*). To apply the option, click on the corresponded checkbox.

- 5 point integration (it can be set only for quadratic tetrahedral elements)

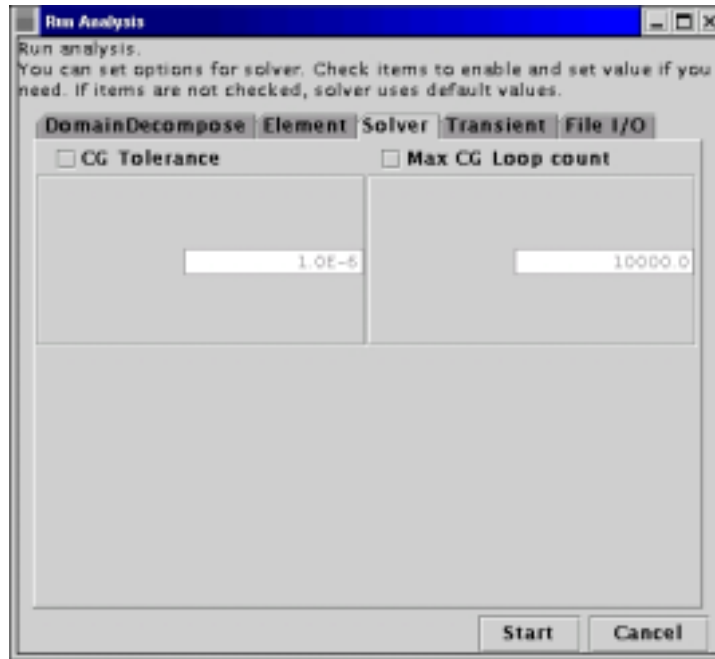


*Fig. 53*



The solver-related options can be set in the tab menu “Solver”. To apply these options, click on the corresponded checkboxes.

- The tolerance for CG method
- The maximum number of loops for CG iterations



*Fig. 54*

The options related to transient analyses can be set in the tab menu “Transient” (Fig. 55). To apply these options, click on the corresponded checkboxes.

- The time integration according to the Crank-Nicolson method (Crank-Nicolson Int.)
- The maximum number of iterations for time integration (Max Time Step Count)
- The interval for step output. (Time Step Output Interval)
- The interval of time step (Time Step Width [sec])
- Initial temperature for all nodes (Initial Temperature)

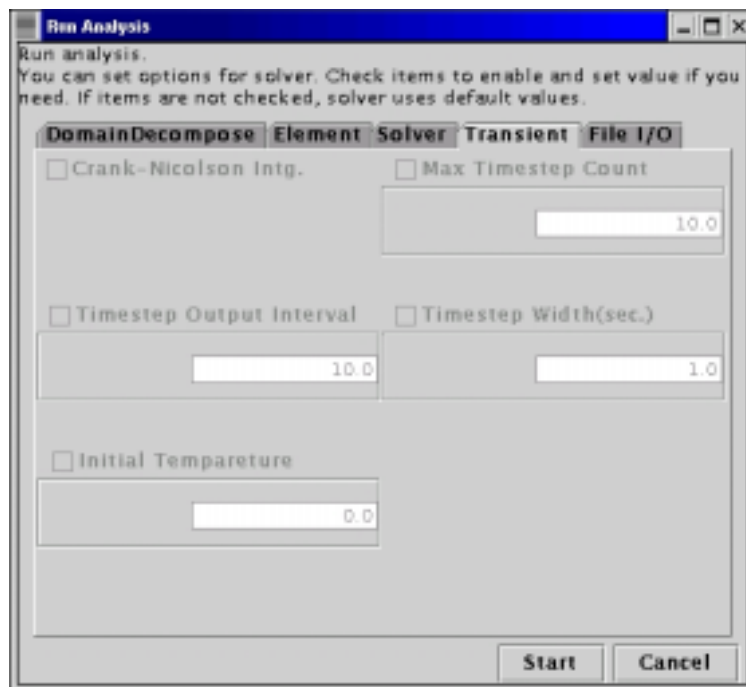
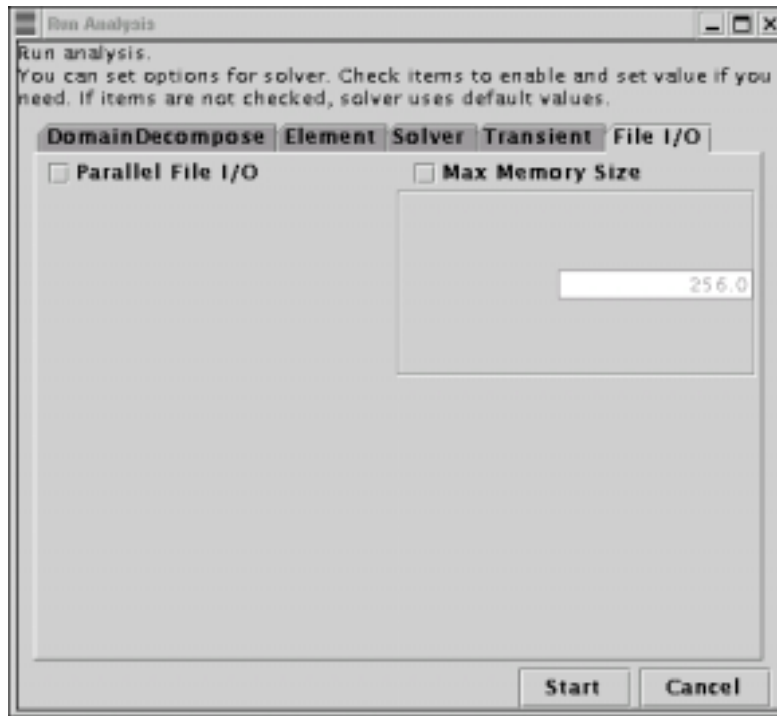


Fig. 55

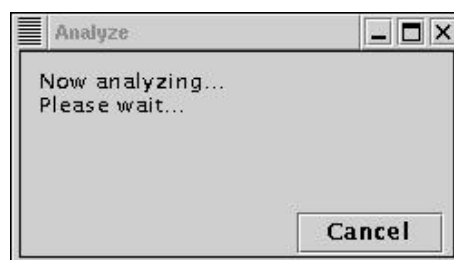
The options related to input/output can be set in the tab menu “File I/O” (*Fig. 56*). To apply these options, click on the corresponded checkboxes.

- The parallel data processing (Parallel File I/O)
- The maximum memory size used by one process (Max Memory Size)



*Fig. 56*

When the setup of all necessary options is finished, click on the “Start” button. The solver will analyze the problem after making the necessary domain decomposition. The window shown in *Fig. 57* will be displayed.

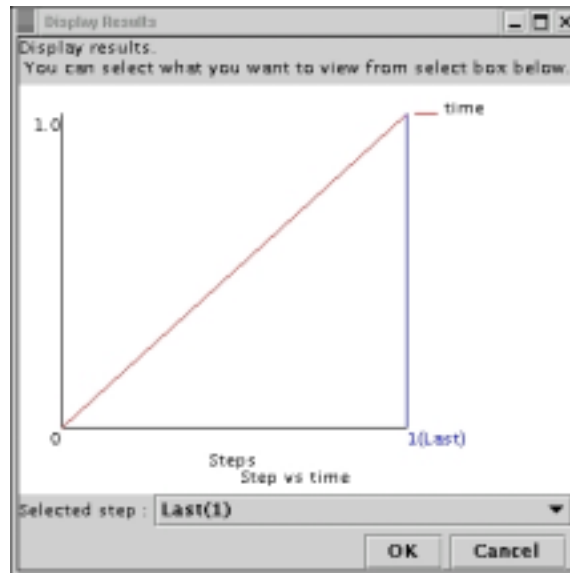


*Fig. 57*

In the current version of iAgent, the button “Cancel” does not support the function of solver termination (it will be supported in the future version of iAgent).

## 7.5. Displaying of Analysis Results

The results of analysis can be visualized using the module ADVENTURE\_Visual. To display the results, select the submenu “View Result” of the menu “Analysis”. The window shown in *Fig. 58* will appear on the screen.



*Fig. 58*

Since, the linear analysis was selected here, ignore the scale of the graph and click “OK” button. The module AdvVisual will be started automatically (*Fig. 61*).

In case of non-linear analysis, the window shown in *Fig. 59* will appear on the screen. The graph shows the interval steps, which have been applied during analysis. The horizontal axis corresponds to the step numbers. The step number, which results should be visualized, can be selected using the menu “Selected step” at the lower part of the window. As it is shown in *Fig. 60*, the selected step, in this case it is step No.6, is shown by the vertical blue line.

After selection of the step for visualization, click the “OK” button that will start the AdvVisual module (*Fig. 61*).

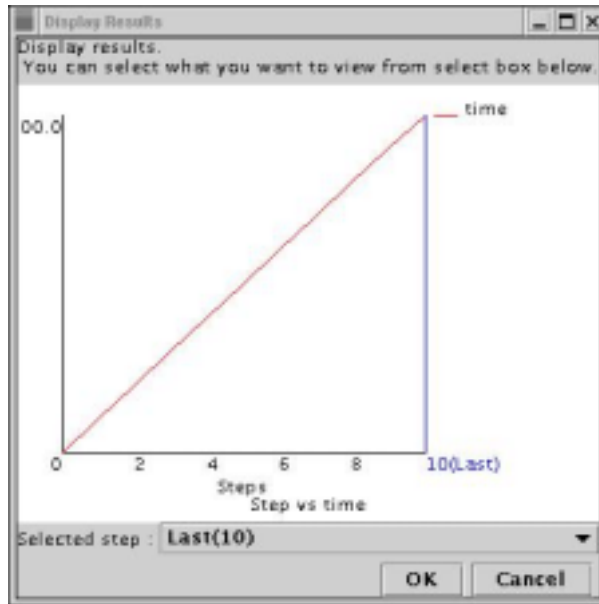


Fig. 59

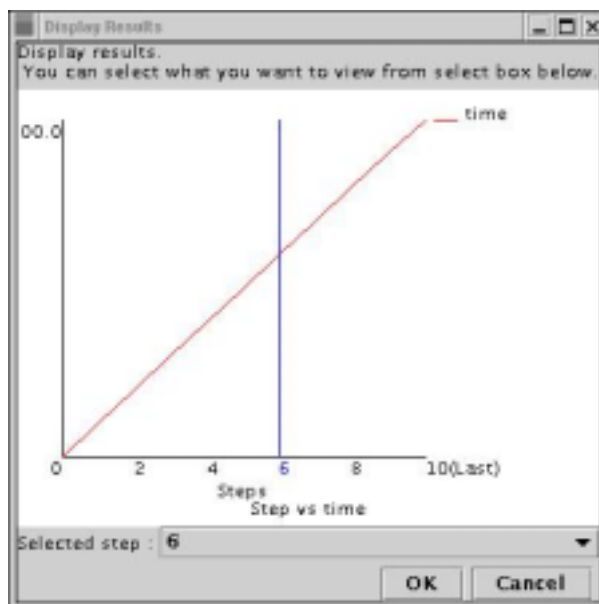
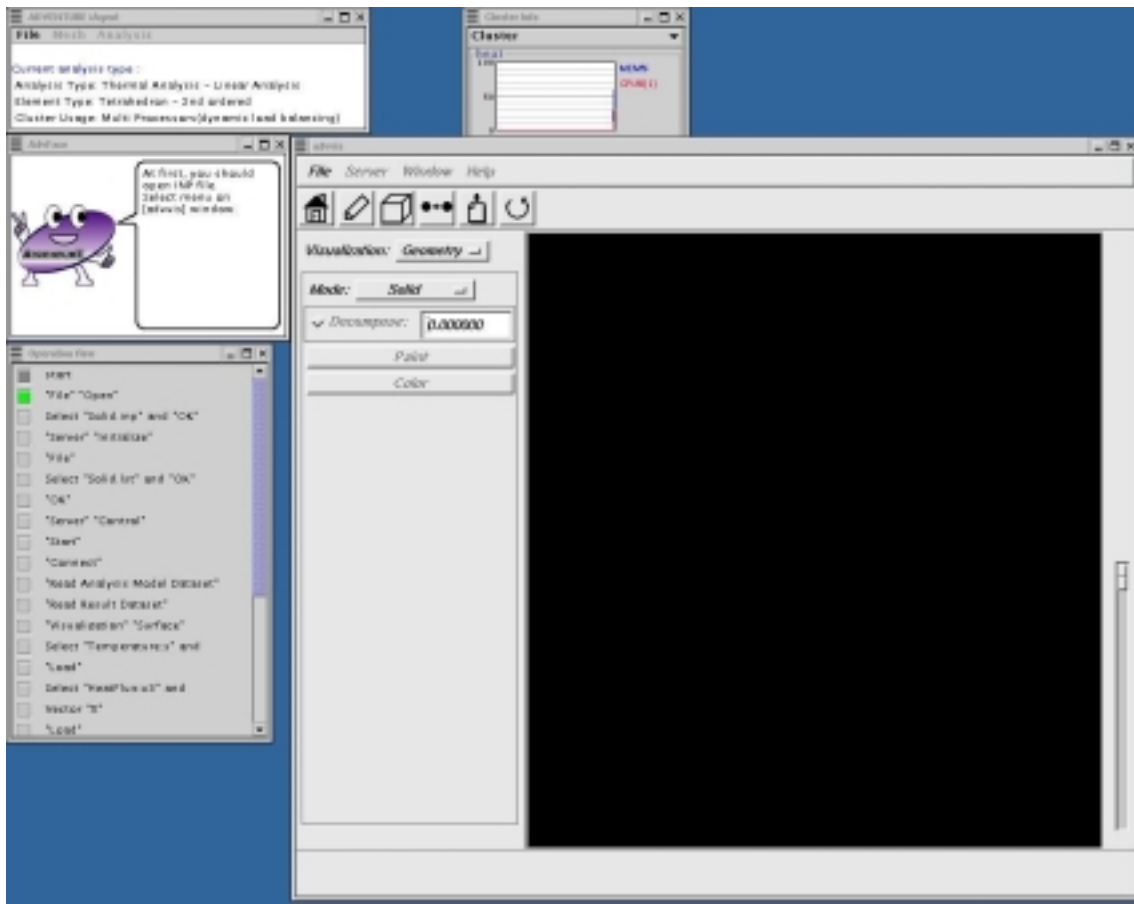


Fig. 60



*Fig. 61*

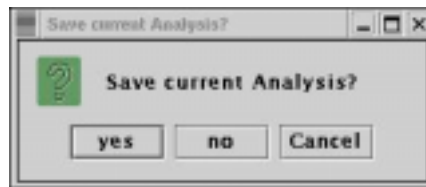
After start-up of the AdvVisual module, the Flow window will be changed to display AdvVisual (as it was done for AdvBCtool). You can use the navigation messages provided in the Message window to display the visualized results.

## 8. Thermal Stress Analysis

The results of thermal analysis can be used as an input for the thermal stress analysis. To use this function, the thermal analysis should be done before the thermal stress analysis.

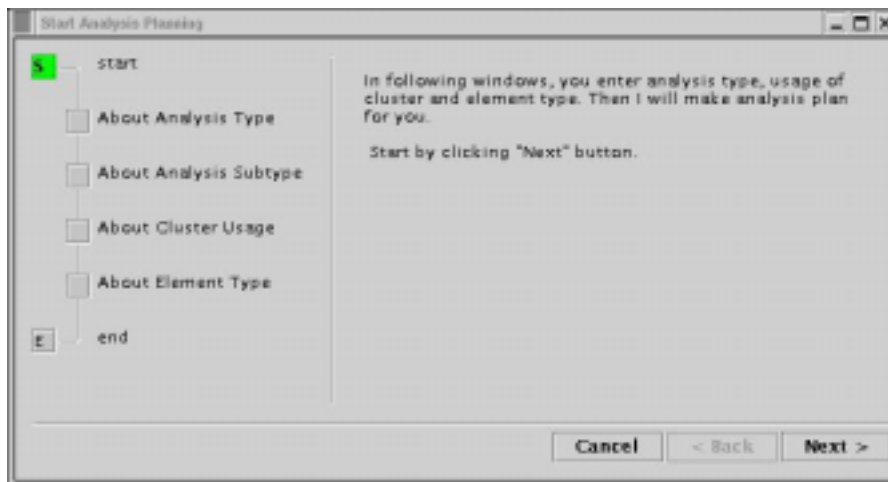
### 8.1. Confirmation of User's Objectives

To start new analysis, select the submenu "Create New Analysis Case" in the menu "File". The iAgent will ask you whether it is necessary to save the current analysis or not (see *Fig 62*). If you need to proceed with the current analysis or if you need to save the current data, select the button "Yes". Otherwise, select "No".



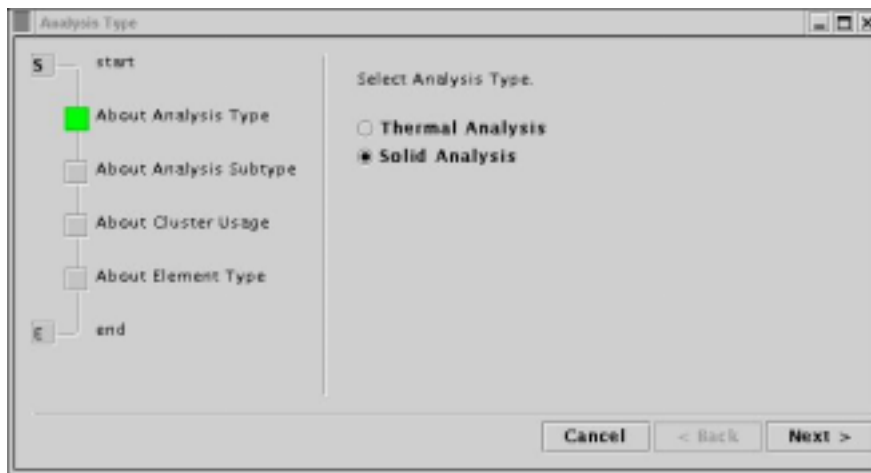
*Fig. 62*

To create new "Analysis Case", iAgent will start to ask the user some questions to navigate him through analysis. The window shown in *Fig. 63* will appear on the screen. Go to the next analysis stage by clicking on the "Next" button.



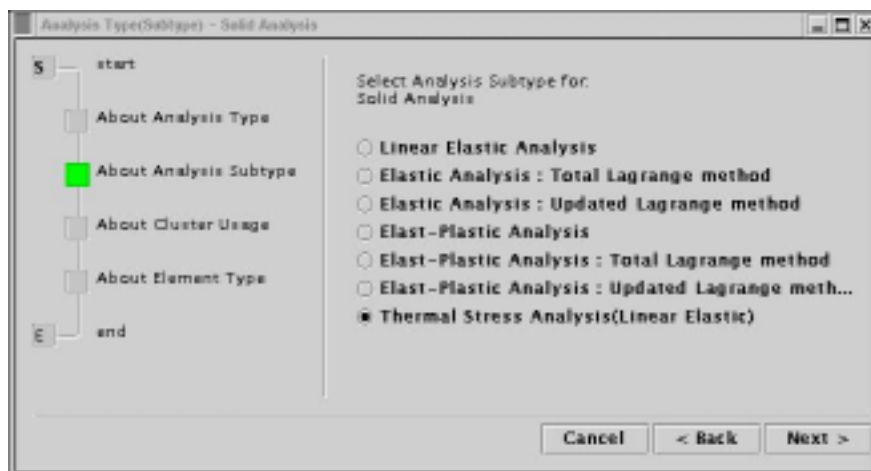
*Fig. 63*

At the beginning, select the type of analysis as it is shown in *Fig. 64* (the option “Solid Analysis” is selected).



*Fig. 64*

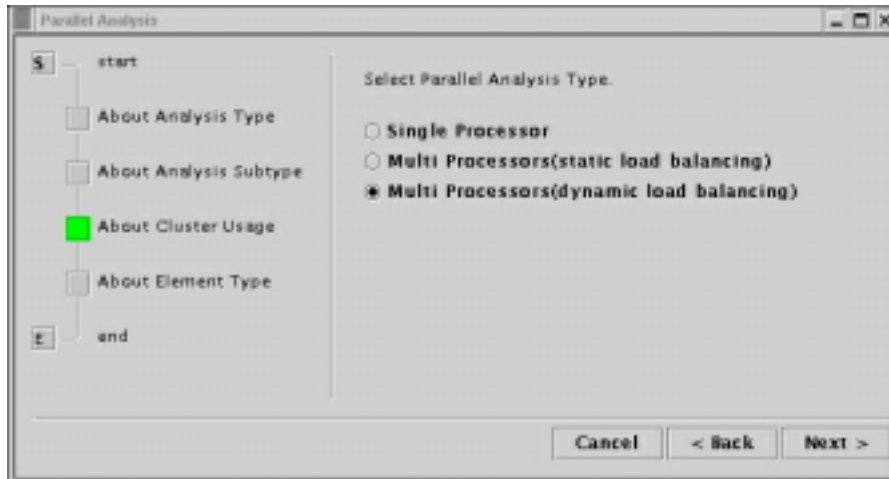
Select the option “Thermal Stress Analysis (linear elastic)” as it is shown in *Fig. 65*.



*Fig. 65*

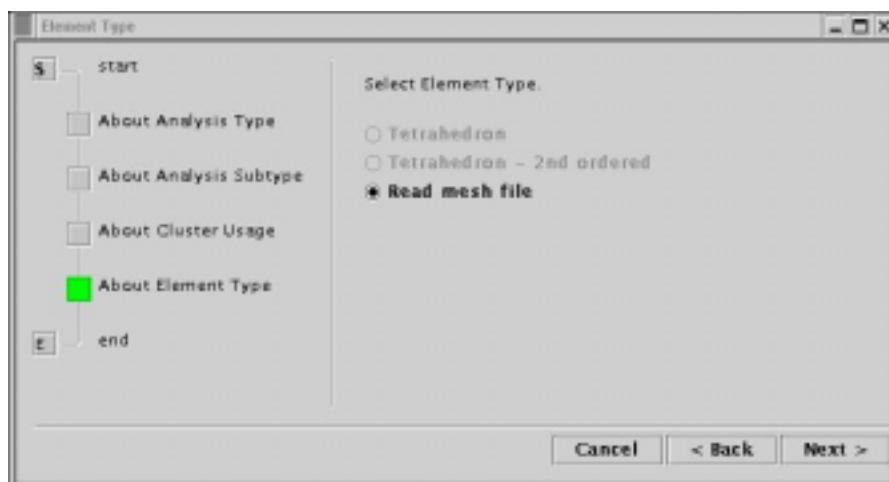
Next, select the network nodes of cluster computer as it is shown in *Fig. 66*. Here, the load balance is done by the way of dynamic load distribution (dynamic load balancing).





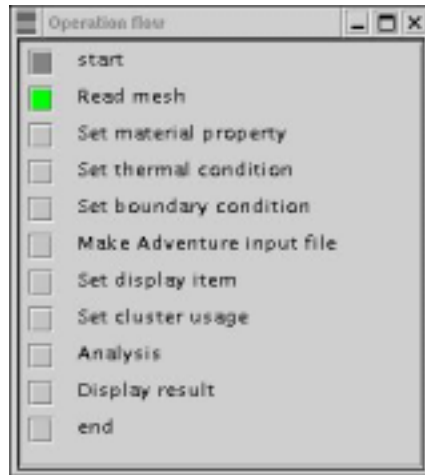
*Fig. 66*

Select the type of element that will be used for the finite element model as it is shown in *Fig. 67*. For the thermal stress analysis, it is necessary to use the mesh that was used for the thermal analysis. This mesh data should be read from file prepared at the step of thermal analysis. Select the option "Read mesh file".



*Fig. 67*

According to the made selections, the analysis plan will be created automatically and the sequence of necessary operations will be shown in the Flow window (*Fig. 68*). The analysis will proceed according to the created analysis plan.

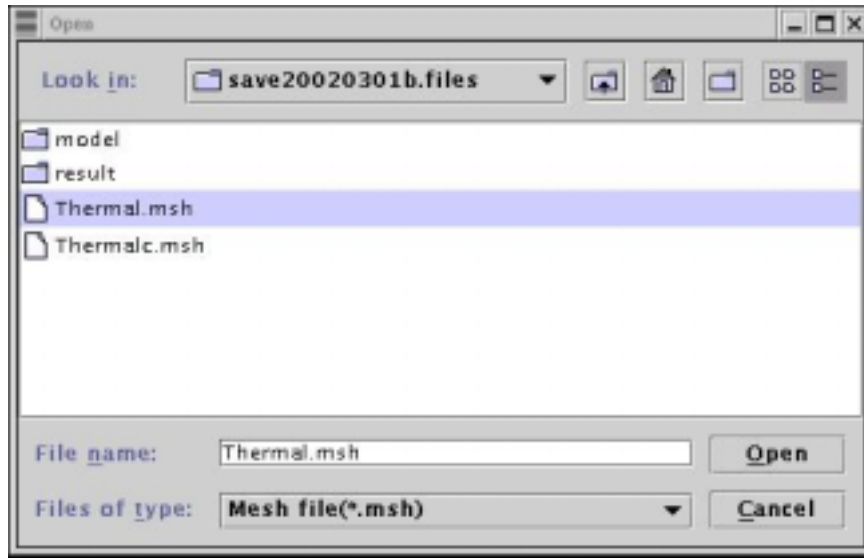


*Fig. 68*

## 8.2. Reading of Mesh Data File

As it was mentioned, the thermal stress analysis uses the mesh data that was used for the thermal analysis. This mesh data should be read from file prepared at the step of thermal analysis.

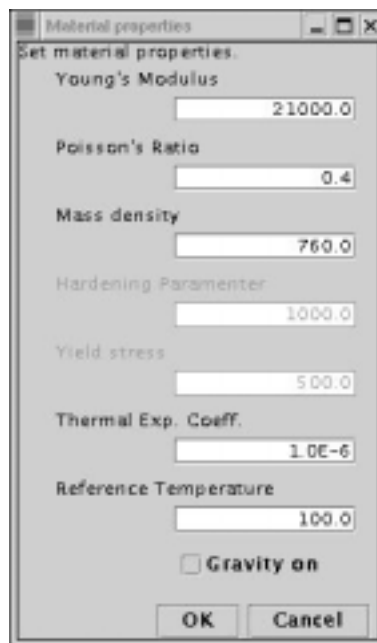
Select the submenu “Load Mesh” of the menu “Mesh”. The window shown in *Fig. 69* will appear on the screen. Select the file to be read and click the button “Open”.



*Fig. 69*

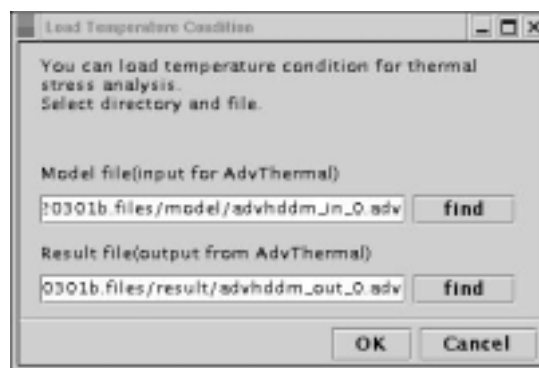
### 8.3. Setup of Material Properties and Boundary Conditions

To set the material properties, select the submenu “Solid” from the menus “Set material Property” and “Analysis”. The dialog box of *Fig. 70* will appear on the screen. The data, which are not necessary for the current analysis will be shown by dim gray color. To set the body force, check the option “Gravity on” in the bottom of the menu dialog.



*Fig. 70*

At the next step, the temperature distribution, which was obtained from the thermal analysis, should be read. To read the temperature distribution from file, select the submenu “Load Temperature Condition” form the menu “Analysis”. The window shown in *Fig. 71* will appear on the screen.



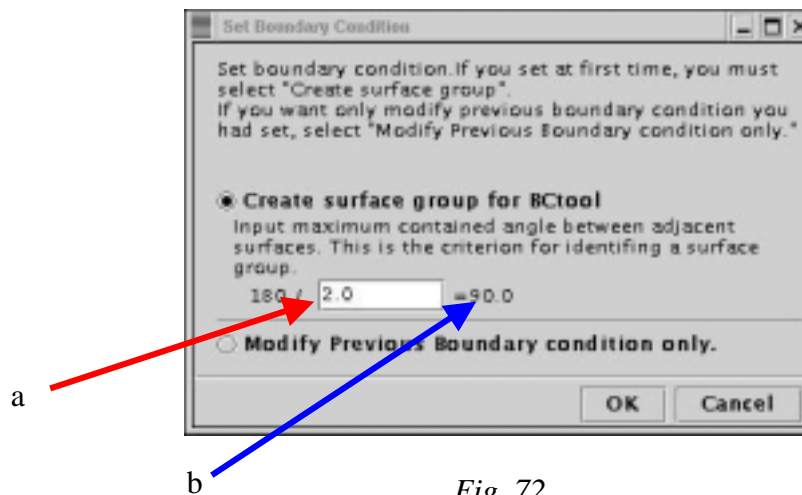
*Fig. 71*

To read the temperature data from file, it is necessary to set the input and output files used for the thermal analysis. Input the filename directly or find the necessary file by using the button “find” of the window. If the model was decomposed into several domains (for example, advhddm\_in\_0.adv ~ advhddm\_in\_6.adv), select one file (any) from the file list of the decomposed model.

If you used iAgent to perform the thermal analysis, and the “Analysis Case” is named /home/user/data/thermal.iag , set the model filename as /home/user/data/thermal.files/model/advhddm\_in\_0.adv and the output results file as /home/user/data/thermal.files/result/advhddm\_out\_0.adv

The next step is to set the boundary conditions. The boundary conditions will be set using the module ADVENTURE\_BCtool, which will be started by iAgent. iAgent will guide you through the process.

First, select the submenu “Set Boundary Conditions” form the menu “Analysis”. The window shown in *Fig. 72* will appear on the screen.



*Fig. 72*

BC tool will divide the mesh surface by a number of groups, and the boundary conditions can be set for these groups. The mesh data are grouped using the angle parameter between adjacent surfaces as a criterion (see the User’s Manual for ADVENTURE\_BCtool for details). *Fig. 72* (a) shows the input dialog for the input of adjacent angle parameter and the arrow (b) shows the angle between the adjacent surfaces.

Input the necessary parameter and click the “OK” button. The mesh will be grouped automatically, and the ADVENTURE\_BCtool module will be started (*Fig. 73*).

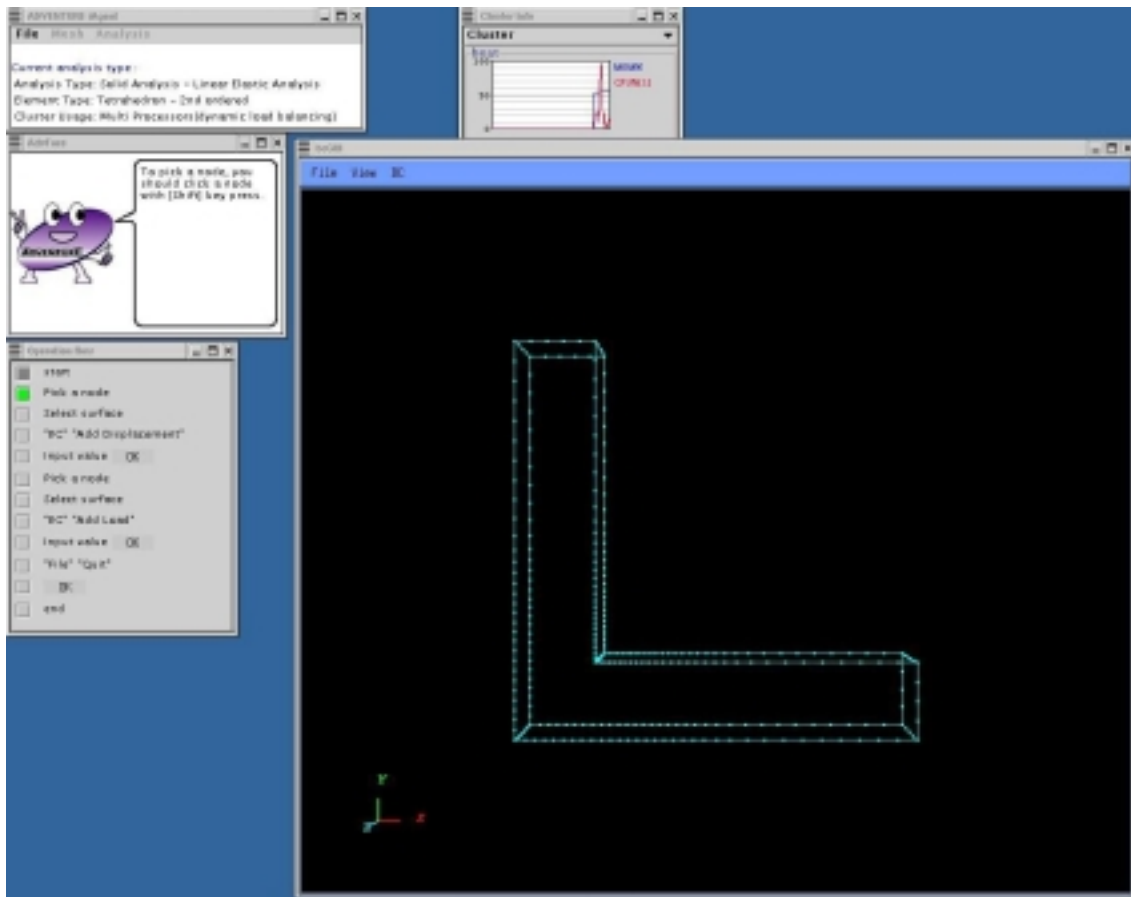


Fig. 73

At this point, the Flow window will be changed automatically to display the operation steps necessary for BCtool. Refer to messages displayed in the Message window to proceed. Explanations about the usage of BCtool are provided in the ADVENTURE\_BCtool User's Manual.

After setting of the boundary conditions, exit from BCtool. The boundary conditions will be automatically saved and checked by iAgent (to save the boundary conditions by user is unnecessary).

The operation of setting of the boundary conditions and material properties will be finished after saving of the data into the entire-type input data file. To save the data, select the submenu "Convert to Input File" of the menu "Analysis". The message shown in Fig. 74 will appear on the screen.

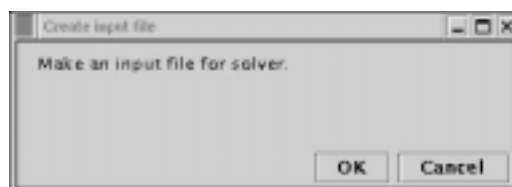
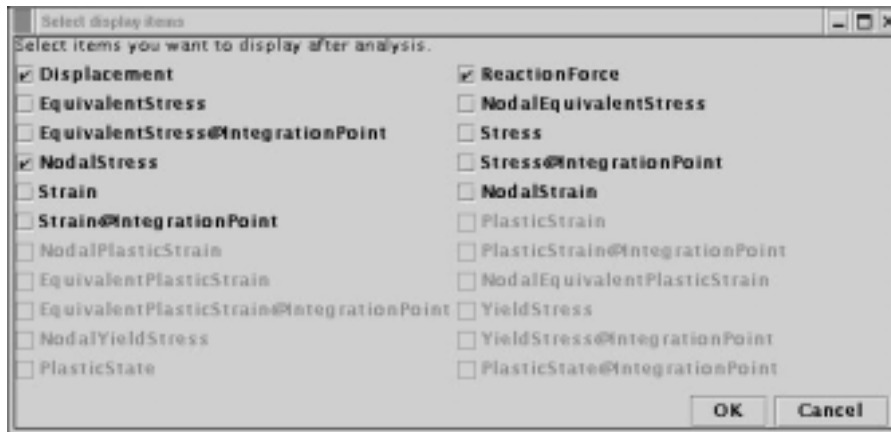


Fig. 74

## 8.4. Execution of Analysis Solver

Prior to execution of the analysis solver, it is necessary to select the parameters for output results, the names of network hosts, which will be used by solver for parallel calculations, and the incremental steps, which will be applied in case of non-linear analyses.

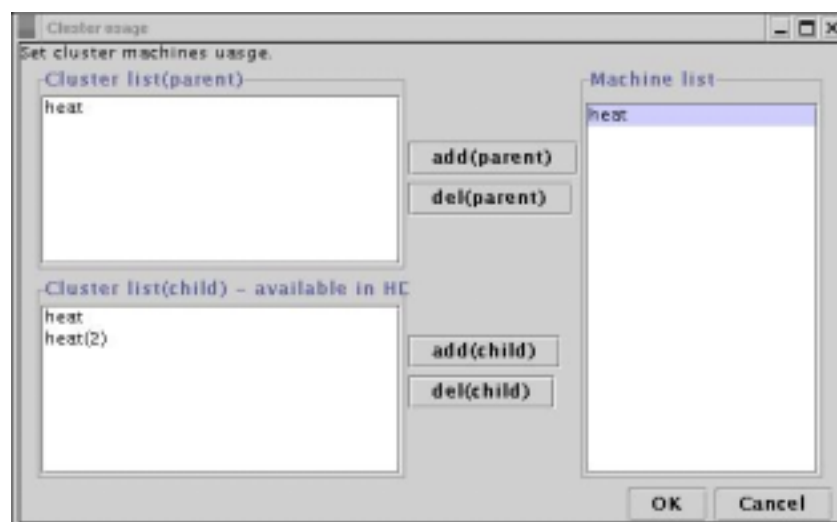
To set the output parameters, select the submenu “Solid”, the submenu “Set Display Item” of the menu “Analysis”. The window shown in *Fig. 75* will appear on the screen.



*Fig. 75*

The items possible for output in the current analysis are displayed by black font. Click on the checkboxes next to the items, which will be printed out after analysis. If the checkbox is checked, the item will be printed out.

To set the usage of network nodes (only in case of parallel computing), select the submenu “Set Cluster Usage” of the menu “Analysis”. The window shown in *Fig. 76* will appear on the screen.



*Fig. 76*

Here, the *Parent* and the *Child* nodes should be specified for the case of dynamic load distribution, and only the *Parent* node should be specified for the case of static load distribution. Refer to the ADVENTURE\_Solid User's Manual for details.

To start the solver, select the submenu "Start Analysis" of the menu "Analysis". The window shown in Fig. 77 will appear on the screen. The options, which can be used with the solver, can be set in this window. If no options have been specially specified, default solver options will be used.

The number of *Subdomains*, which will be treated by 1 CPU can be set in the tab menu "Domain Decompose" (Fig. 77) using 2 methods.

- The number of elements in one *Subdomain* (click on the radio button "Input # of elements/subdomain")
- The number of *Subdomains* (click on the radio button "Input # of subdomains/part")

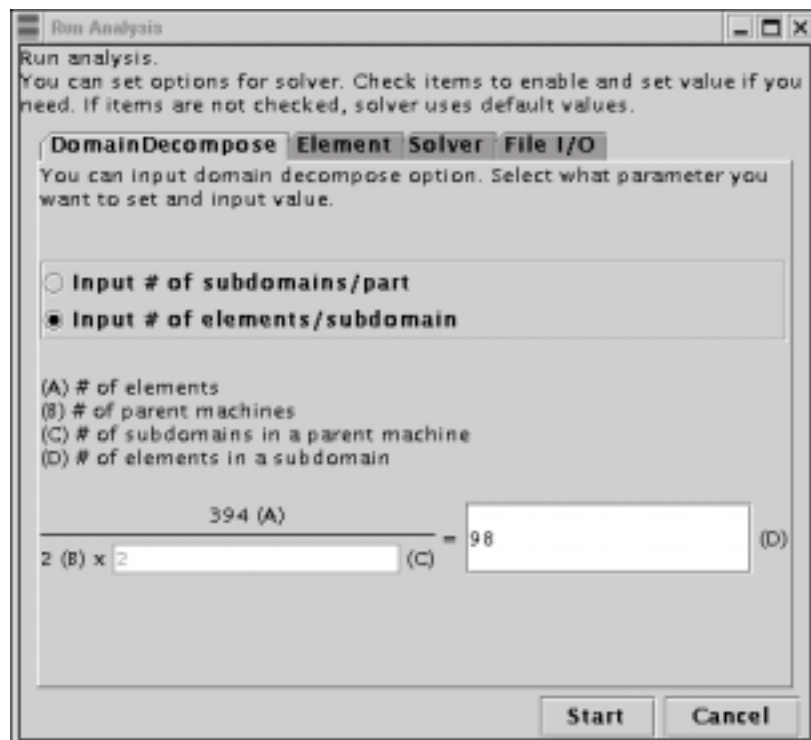
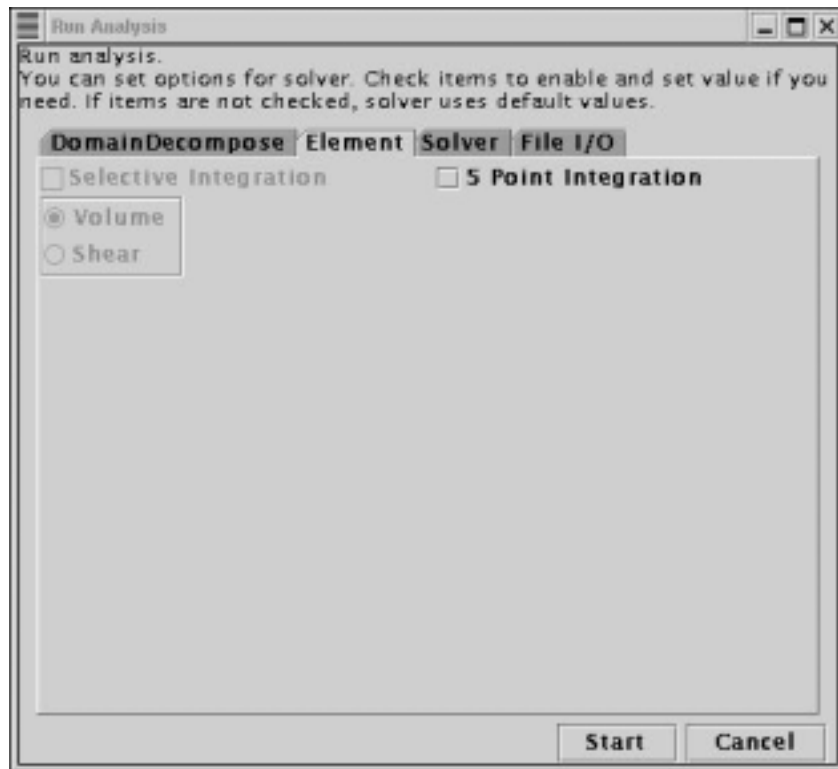


Fig. 77



The options related to elements can be set in the tab menu “Elements” (*Fig. 78*). The following 2 options cannot be used together at once. To apply these options, click on the corresponded checkboxes.

- Selective integration (volumetric strain, shear strain) (it can be set only for linear hexahedral elements)
- 5 point integration (it can be set only for quadratic tetrahedral elements)



*Fig. 78*

The solver-related options can be set in the tab menu “Solver” (Fig. 79). To apply these options, click on the corresponded checkboxes.

- The solver type
- The tolerance for CG method
- The maximum number of loops for CG iterations
- The option for saving of the K-matrix
- The parameter for Neumann-Neumann preconditioning (only for BDD solver)
- The tolerance for iterations (Newton Tolerance) (applied only for non-linear analyses)
- The maximum number of iterative loops (applied only for non-linear analyses)

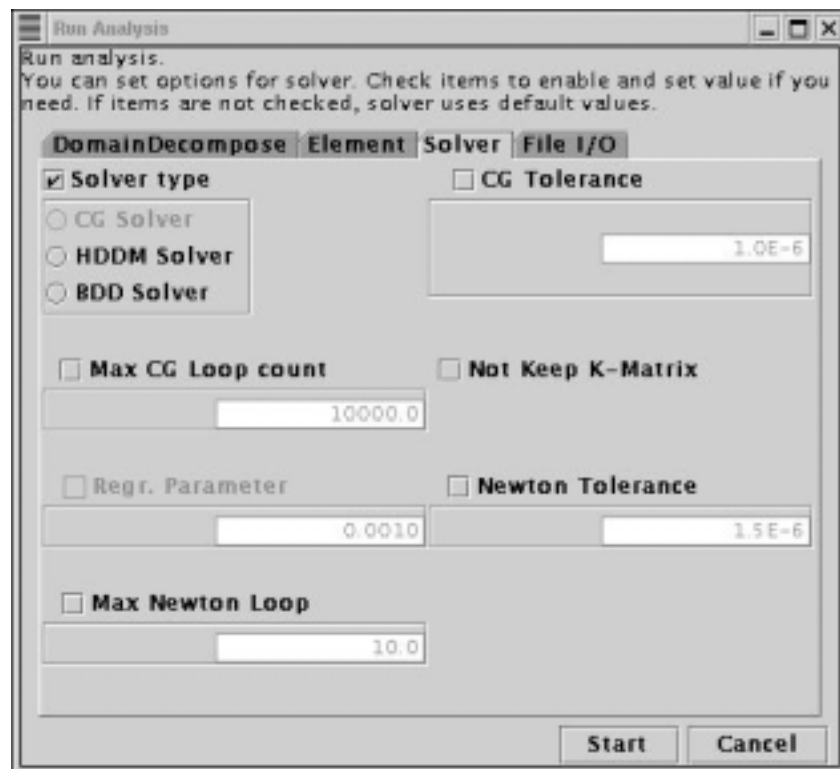
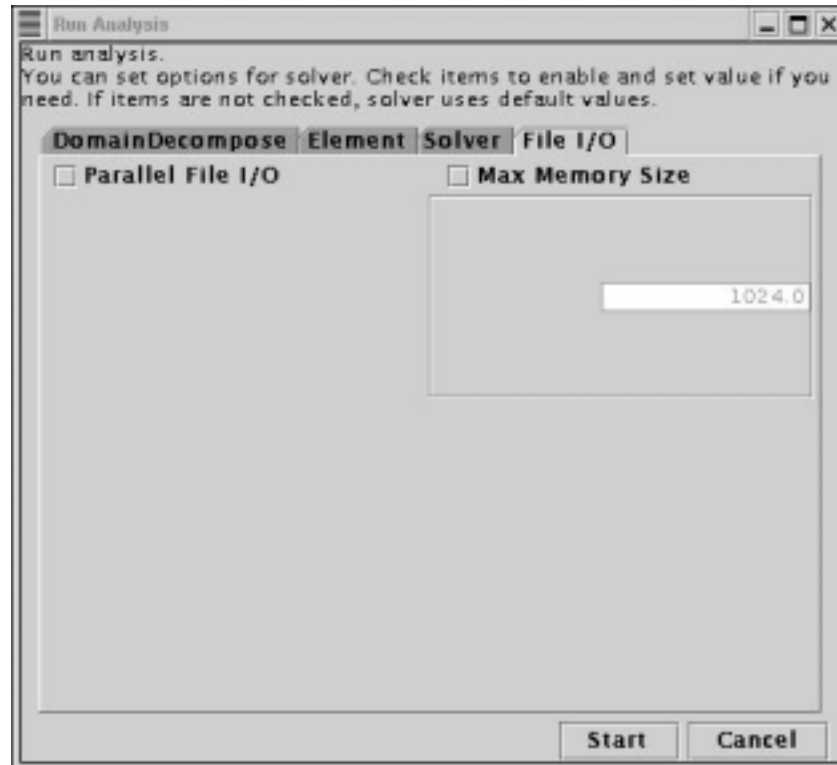


Fig. 79

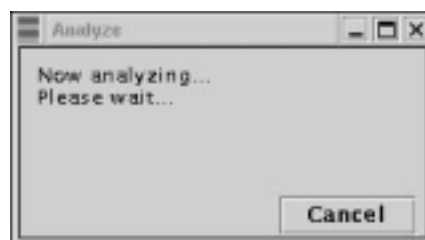
The options related to input/output can be set in the tab menu “File I/O” (*Fig. 80*). To apply these options, click on the corresponded checkboxes.

- The parallel data processing (Parallel File I/O)
- The maximum memory size used by one process (Max Memory Size)



*Fig. 80*

When the setup of all necessary options is finished, click on the “Start” button. The solver will analyze the problem after making the necessary domain decomposition. The window shown in *Fig. 81* will be displayed.

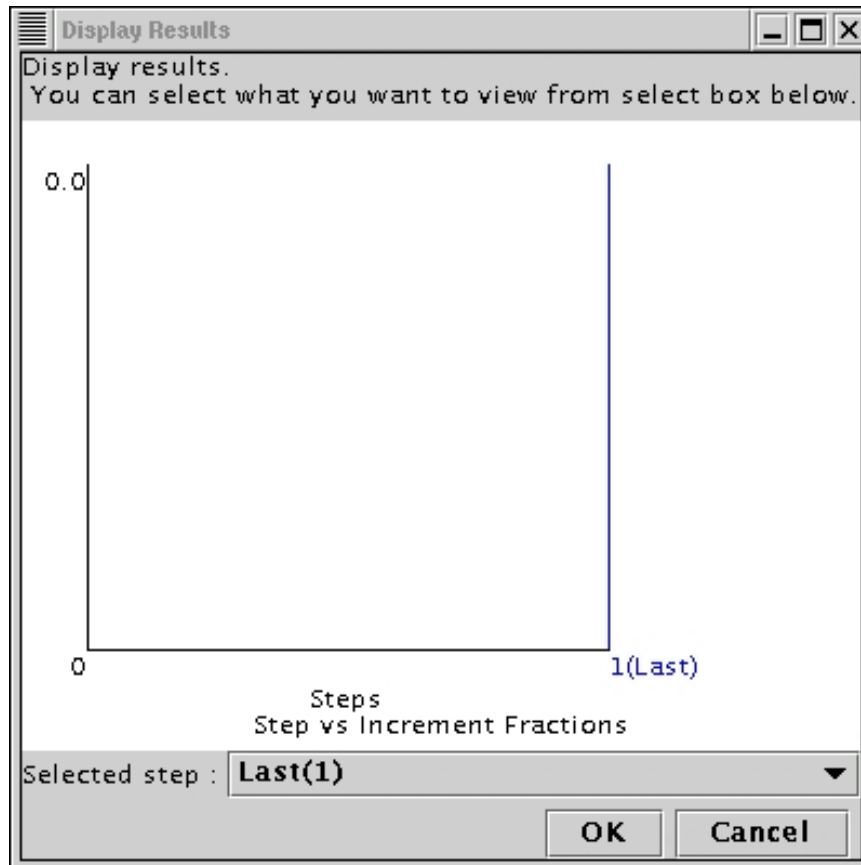


*Fig. 81*

In the current version of iAgent, the button “Cancel” does not support the function of solver termination (it will be supported in the future version of iAgent).

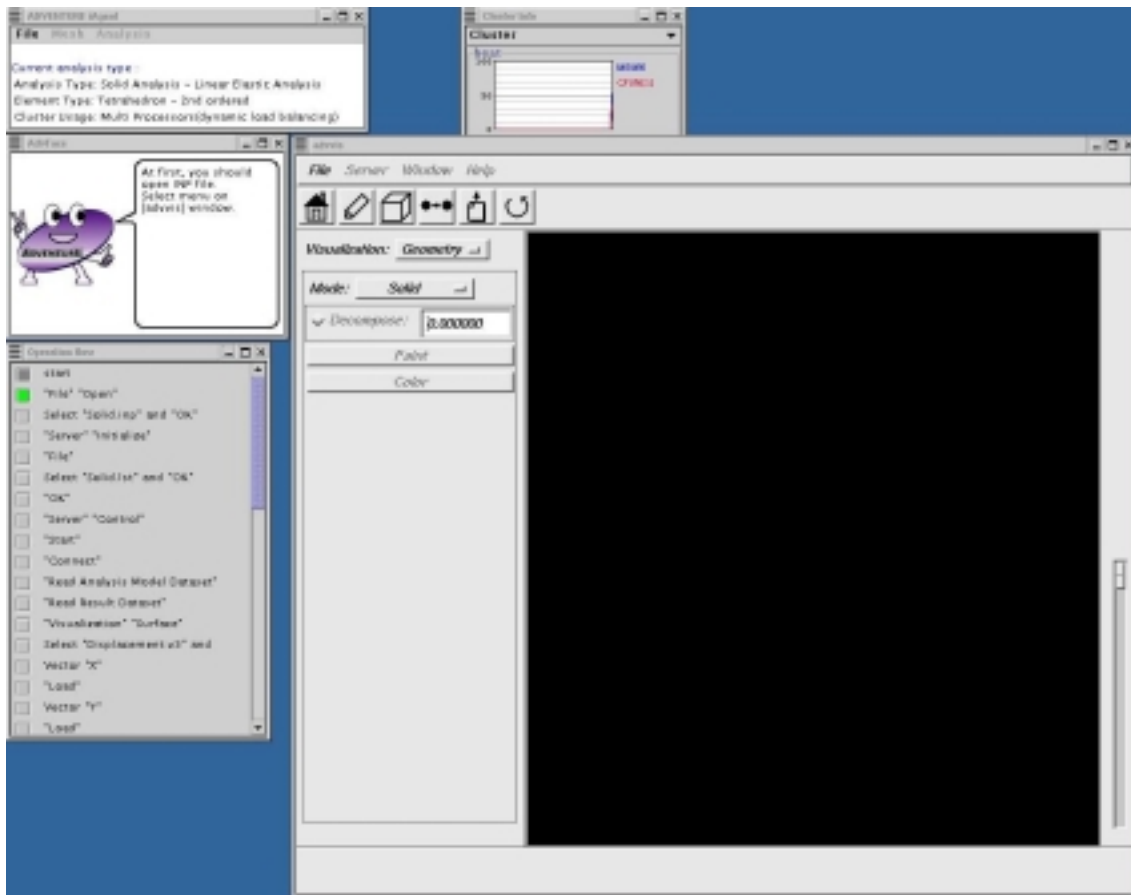
## 8.5. Displaying of Analysis Results

The results of analysis can be visualized using the module ADVENTURE\_Visual. To display the results, select the submenu “View Result” of the menu “Analysis”. The window shown in *Fig. 82* will appear on the screen.



*Fig. 82*

Since, the linear analysis was selected here, ignore the scale of the graph and click “OK” button. The module AdvVisual will be started automatically (*Fig. 83*).



*Fig. 83*

After start-up of the AdvVisual module, the Flow window will be changed to display AdvVisual (as it was done for AdvBCtool). You can use the navigation messages provided in the Message window to display the visualized results.

## Appendix 1. Format of Setup File

### A1.1. Common Setup File

The common setup file is a file, which includes the common settings for iAgent shared for all users. In this version of iAgent, the common setup file should be changed using any available text editor. The filename is:

`${the directory where iAgent is installed}bin/iAgent/iAgent.conf`

The file contents are shown below.

```
iAgent = ${the directory where iAgent is installed}bin/iAgent

*DefaultAnalysis = (Default type of analysis)
*DefaultParallel = (Default method of cluster usage)
*DefaultMesh = (Default method for creation of analysis model)

MPI.mpirun = (Full path to the MPI executable file)
TriPatch.PatchGenerator = (Full path to ADVENTURE_TriPatch)
TetMesh.PatchCorrector = (Full path to the advtmesh8p program)
TetMesh.MeshGenerator = (Full path to the advtmesh8m program)
BCTool.SurfaceGroup = (Full path to the msh2pcg program)
BCTool.BCTool = (Full path to the bcGUI program)
BCTool.MakeFEMInput = (Full path to the makefem program)
BCTool.Region = (Full path to the msh2pcm program)
Metis.DomainDecomposer = (Full path to the adventure_metis domain decomposer)
Metis.DomainDecomposer.singleAnalysis = (Filename of adventure_metis)
Solid.Solver = (Full path to advsoild)
Solid.Solver.singleAnalysis = (The filename of advsolid)
Thermal.Converter = (Full path to the so2th converter)
Thermal.Solver = (Full path to the advthermal solver)
Tool.ResultMerger = (Full path to the hddmrg program)

Visual.Server = (Full path to the advvis_server program)
Visual.Client = (Full path to the advvis program)
VRMLViewer = (Full path to the VRML viewer program)

*localhost = (The address of host for the iAgent module startup)
*clusterList = (The file with cluster hostnames' list)

nPart.Solid.singleAnalysis = (The number of decomposed domains for the Single CPU
                             version)
nPart.Solid.staticParallel = (The number of decomposed domains for the Static load
                              distribution version)
nPart.Solid.dynamicParallel = (The number of decomposed domains for the Dynamic
                               load distribution version)

The other rows should not be changed (they are omitted here)
```

\* Do not change items, which are not listed here.

\*\* If the line is started with “#” mark, the information will be ignored (will be treated as comments).

This version of iAgent supports the following types of analyses.

Types of analyses:

- `Solid.LinearElastic` (Linear elastic analysis / a-1)
- `Solid.ElasticTotalLagrange` (Elastic analysis with large displacement and infinitesimal strain / a-2)
- `Solid.ElasticUpdatedLagrange` (Elastic analysis with large displacement and large strain / a-3)
- `Solid.Plastic` (Elastic-plastic analysis / a-4)
- `Solid.PlasticTotalLagrange` (Elastic-plastic analysis with large displacement and infinitesimal strain / a-5)
- `Solid.PlasticUpdatedLagrange` (Elastic-plastic analysis with large displacement and large strain / a-6)
- `Solid.LinearElasticThermal` (Linear thermal stress analysis / a-7)
- `Thermal.Linear` (Linear thermal analysis / a-8)
- `Thermal.Nonlinear` (Non-linear thermal analysis / a-9)

Methods of job distributions between CPUs:

- `SingleAnalysis` (Analysis using a single CPU / c-1)
- `StaticParallel` (Analysis using parallel static load distribution / c-2)
- `DynamicParallel` (Analysis using parallel dynamic load distribution / c-3)

Methods for model creations

- `Tetra` (Linear tetrahedral element / m-1)
- `Tetra20` (Quadratic tetrahedral element / m-2)
- `ReadMeshFile` (Reading of mesh data from file / m-3)

However, there are limitations imposed on using of the analysis model types in some combinations of the analyses (the analysis model cannot be created). Possible combinations of analyses are presented in *Table 1* (the rows and columns are corresponded to the numbers given in the parentheses of listed items).

*Table 1*

	c-1	c-2	c-3	m-1	m-2	m-3
a-1	OK	OK	OK	OK	OK	OK
a-2	OK	OK	OK	OK	OK	OK
a-3	OK	OK	OK	OK	OK	OK
a-4	OK	OK	OK	OK	OK	OK
a-5	OK	OK	OK	OK	OK	OK
a-6	OK	OK	OK	OK	OK	OK
a-7	OK	OK	OK	Not supported	Not supported	OK
a-8	OK	OK	OK	OK	OK	OK
a-9	OK	OK	OK	OK	OK	OK

## **A1.2. User Setup File**

The user setup file is used to save default settings for each user (for example, default type of analysis). It has the same format with the common setup file, but higher priority for reading of settings. After startup, iAgent reads the user setup file and, if the user setup file cannot be found, iAgent reads the common setup file. The items, which are not set in the user setup file, are read from the common setup file.

The name of the user setup file is `${HOME}/.iAgent/iAgent.conf`. iAgent does not create this file automatically; each user should create it by any available text editor.