ADVENTURE_BCtool

Attachment of boundary conditions and material properties to mesh

Version: 1.0

User's Manual

March 1, 2002

ADVENTURE Project

Contents

1. Outline

The module ADVENTURE_BCtool is an interactive pre-processing program used to attach boundary conditions and material properties data to the mesh and to create finite element models for ADVENTURE system.

This manual is a reference guide to using ADVENTURE_BCtool.

2. Program Features

Types of elements supported by ADVENTURE_BCtool:

- Linear tetrahedral element
- Quadratic tetrahedral element
- Linear hexahedral element
- Quadratic hexahedral element

Types of analyses supported by ADVENTURE_BCtool:

- Elastic analysis
- Elastic-plastic analysis
- Thermal stress analysis

Boundary conditions can be set to

- Surface of grouped mesh
- Primary nodes located on surface boundaries of grouped mesh

Possible boundary conditions are:

- Load (in X, Y, and Z directions, and in the direction of normal to the surface)
- Displacement (in X, Y, and Z directions, in the direction of normal to the surface)
- Acceleration of gravity

Material properties, which can be set:

- Young's modulus
- Poisson's ratio
- Hardening parameter
- Yield stress
- Density of material

3. System Requirements

OS: UNIX, Linux
Compiler: GNU *C++* (ver. 2.95.x, 2.96)
Libraries: `Motif` (Ver. 1.2 or higher) or `LessTif` (Ver. 0.92.0 or higher)
`OpenGL` (Ver. 1.1 or higher) or `Mesa` (Ver. 3.2 or higher)
`ADVENTURE_IO` (library of the ADVENTURE system)

`LessTif` (http://www.lesstif.org/)
`Mesa` (http://www.mesa3d.org/)

4. Installation Method

To install the ADVENTURE_BC tool module, refer to the file "Install" located right under a top directory after achieve extraction.

5. Using the Program

To attach boundary conditions and material properties data to the mesh and to create finite element model using ADVENTURE_BCtool, you need to accomplish the following steps.

Step 1. Extraction of mesh surface;
Step 2. Attaching boundary conditions to mesh using the graphic user interface;
Step 3. Creation of entire-type FEA model (including settings of boundary conditions and material properties).

5.1. Step 1. Extraction of Mesh Surface

At this step, the data of extracted mesh surface groups are converted into the GUI input format. The following input and output files are used.

Input:
Mesh data file (extension: `.msh`)

Output:
Mesh surface data file (extension: `.fgr`)
Extracted surface mesh data file (extension: `.pch`)
Patch group data file (extension: `.pcg`)
Global index file (extension: `.trn`)

The shell script `msh2pch` is used for conversion. There are two arguments in `msh2pch` command line.

```
% msh2pch mshFile div_n
```

mshFile is the name of mesh file, and
div_n is the dihedral angle which becomes a base for mesh surface grouping. The dihedral angle should be specified as a fraction (segment) of 180 degrees. The surface, which dihedral angle is more than specified, will be assigned to another group.

Example 1. The filename of mesh data is `Model.msh` and the dihedral angle is 60^o (60^o=180^o/3).

```
% msh2pch Model.msh 3
```

Example 2. The filename of mesh is `Model.msh` and the dihedral angle is 45^o (45^o=180^o/4).

```
% msh2pch Model.msh 4
```

An appropriate dihedral angle varies form model to model and should be adjusted by the user after checking of a condition whether the grouping is enough detailed or not by displaying the object with GUI. If the filename of mesh data is `Model.msh` and the dihedral angle is 180/*N*, the following output will be created:

`Model_N.fgr` ← Mesh surface data file
`Model_N.pch` ← Extracted surface mesh data file
`Model_N.pcg` ← Surface patch group data file
`Model_N.trn` ← Global index file

5.2. Step 2. Setting of Boundary Conditions

The boundary conditions can be set to mesh using the executable module `bcGUI`. The following input and output files are used.

<u>Input files:</u>	Extracted surface mesh data file (extension: `.pch`)
	Surface patch group data file (extension: `.pcg`)
<u>Output file:</u>	File with analysis conditions (extension: `.cnd`)

Executing **`bcGUI`**

`% bcGUI` *pchFile* *pcgFile* `[-icnd` *cndFile*`]` `[-ocnd` *outFile*`]`

pchFile	is the extracted surface mesh data file.
pcgFile	is the surface patch group data file.
cndFile	is the file with analysis conditions which is read automatically at the moment of `bcGUI` startup.
outFile	is the file with analysis conditions which is created automatically at the end of `bcGUI` execution.
`-icnd`	is the option used to set the file with conditions for analysis, which will be read automatically at the start of `bcGUI` execution.
`-ocnd`	is the option used to set the file, which will be written automatically at the end of `bcGUI` execution.
`[....]`	the settings in parenthesis can be omitted.

After startup of `bcGUI`, a default window will appear on the screen. The model will be plotted as it is shown in *Fig. 1* (example).

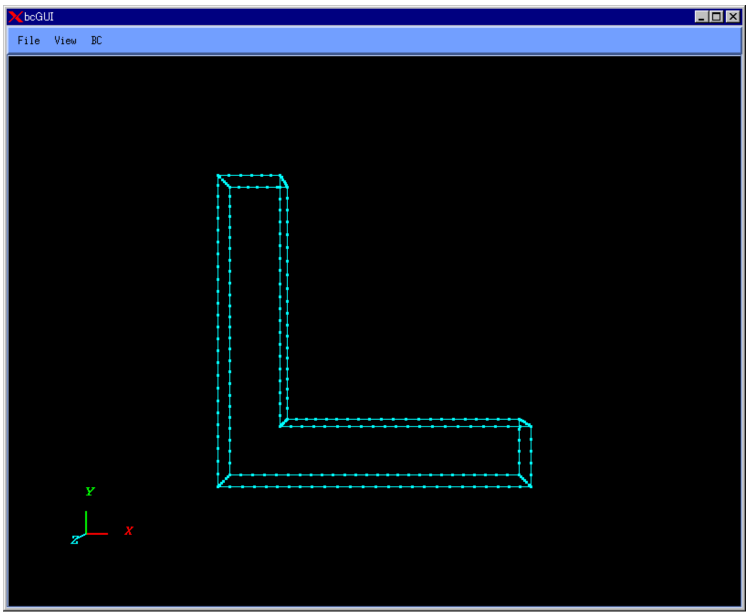


Fig. 1. Initial Window with Model

Changing of Observer Viewpoint

The user is able to use the mouse for panning, zooming, and rotating of the displayed object (model). By default, the left mouse button controls panning, the right mouse button controls zooming, and the center mouse button controls rotating. The user can press the mouse buttons at any time to activate these features.

Panning

To pan the model in the plane of the screen, press the left mouse button and move the mouse. The model will be translated in the moving direction.

Zooming

To zoom the model about the center of the screen, press the right mouse button and move the mouse. Moving the mouse upward the screen causes the model to increase in size. Moving the mouse downward the screen causes the model to decrease in size.

Rotating

To rotate the model, press the center mouse button and move the mouse. The model will rotate about its center along the axis, which is perpendicular to the motion of the mouse (cursor).

Selection of Nodes and Surface Groups

The primary nodes of surface group boundaries are displayed on the screen by round dots. To select the primary node, press the Shift keyboard key with clicking on the node using the left mouse button. The selected node will be shown by vivid color. To deselect the node, press the Shift keyboard key with clicking on the background (place without node) using the left mouse button. If the node is selected, its number will be extracted from the extracted surface mesh data file (extension `.pch`) and displayed on the screen.

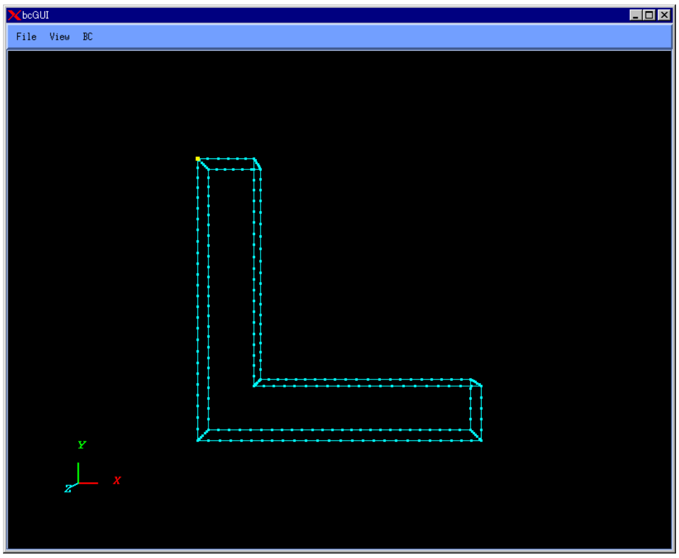


Fig. 2. Selected Node

To select the surface group, which contains the primary node, press the Shift keyboard key with clicking on the node using the left mouse button. Other adjacent surfaces can be selected one by one by pressing the right mouse button. If the surface is selected, its number will be extracted from the surface patch group data file (extension `.pcg`) and displayed on the screen.

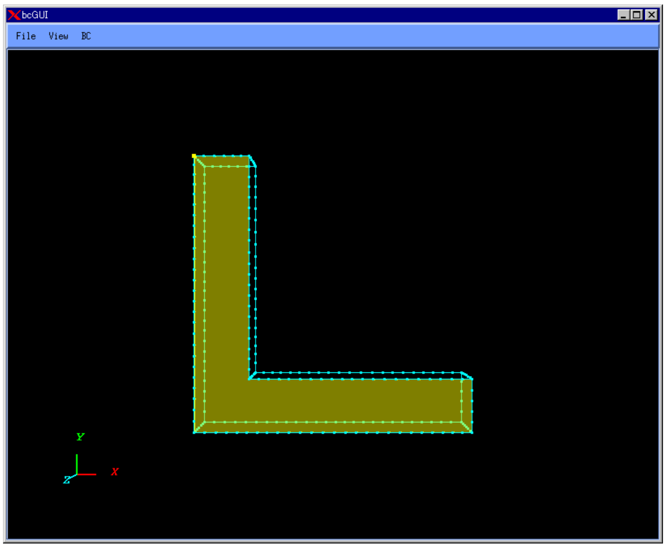


Fig. 3. 1st Selected Surface

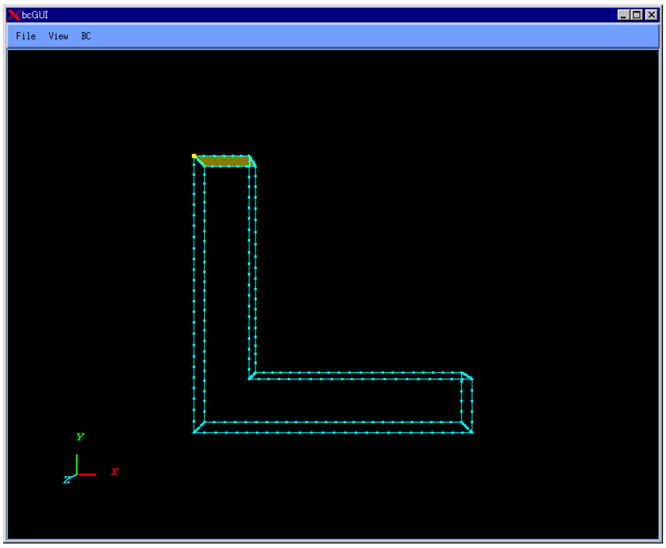


Fig. 4. 2nd Selected Surface

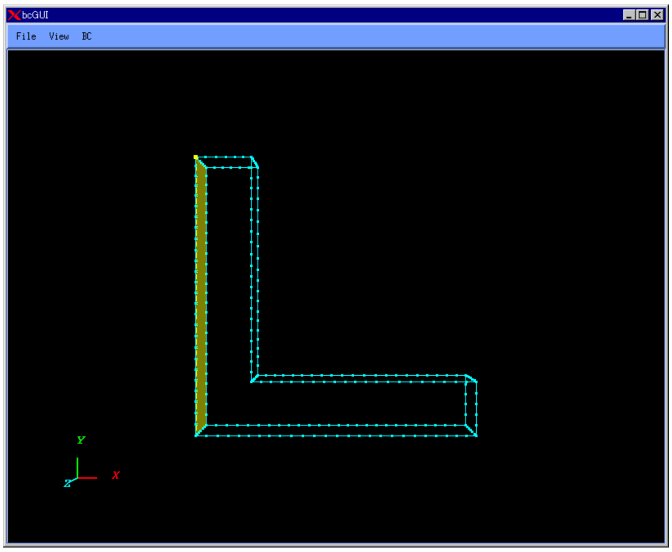


Fig. 5. 3rd Selected Surface

Setup of Load and Displacement Boundary Conditions

To attach the boundary conditions to a node (or surface group):

- Select the node (or surface group) as it was described above.
- Select the menu *Add Load* or *Add Displacement* from the main menu *BC* (*Fig. 6*). The dialog shown in *Fig. 7* will appear on the screen.
- Set the boundary condition by pressing the toggle button against the necessary component (*Fig. 8*).
- The values of components (X, Y, and Z) can be set into the text boxes on the right.
- If the toggle buttons on the left are not pressed, the values set in text boxes on the right will be ignored.
- *Normal* represents the direction, which is perpendicular to the selected surface (triangle or quadrangle of mesh surface). The direction is assumed to be "positive" if the normal to the face is pointed into the object (body), and "Negative" if the normal to a face is pointed outside the object (body).
- The toggle button against *Normal* can be selected only when a face group is selected.
- The settings for "X component", "Y component", and "Z component" cannot be changed if the toggle button *Normal* is pressed.
- Zero value of load and zero value of displacement normal to the surface cannot be set.
- The load on the face group should be specified as a load per unit area.

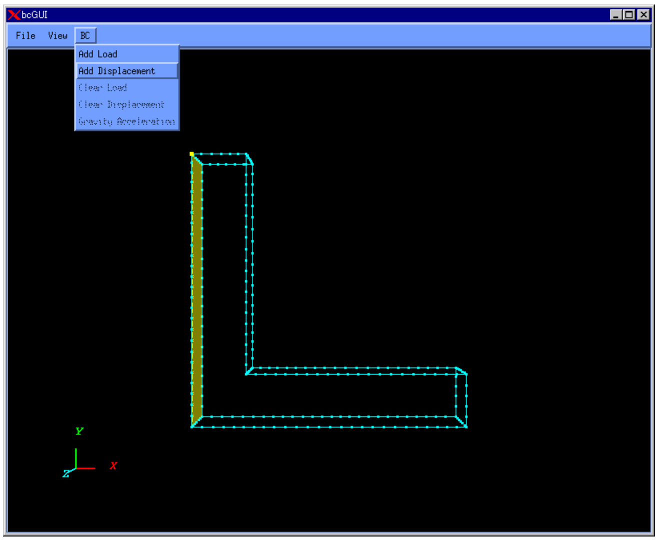


Fig. 6. Menu for Boundary Conditions Setup

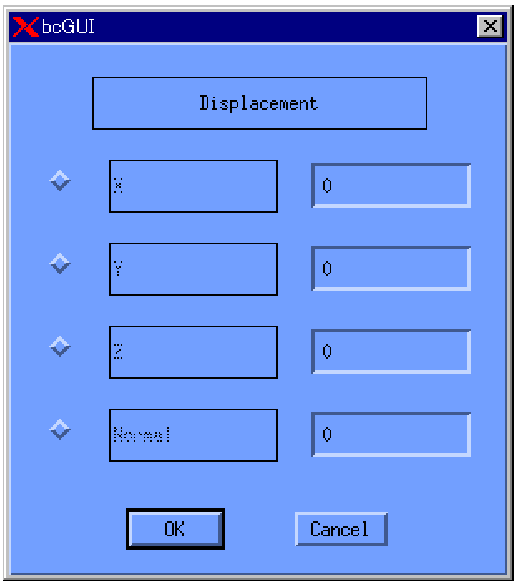


Fig. 7. Dialog for Displacement Boundary Conditions

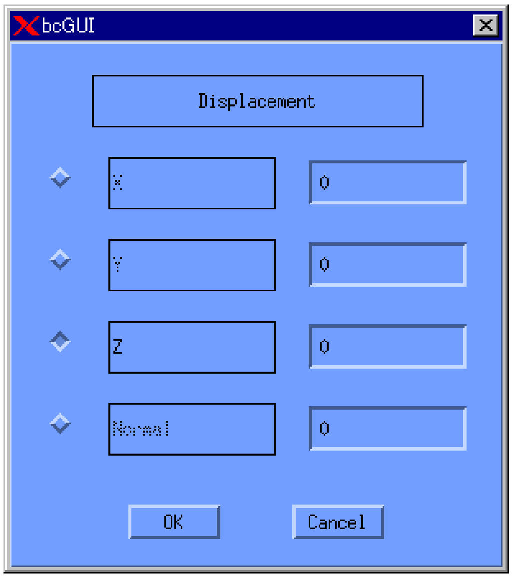


Fig. 8. Displacement in Z Direction Is Fixed

Setup of Gravity

To set up the gravity acceleration, select the option *Gravity Acceleration* in the menu *BC*, as it is shown in *Fig 9*. The dialog shown in *Fig. 10* will be displayed. *X*, *Y*, and *Z* in the dialog menu represent the "X component", "Y component", and "Z component" of gravity, respectively.

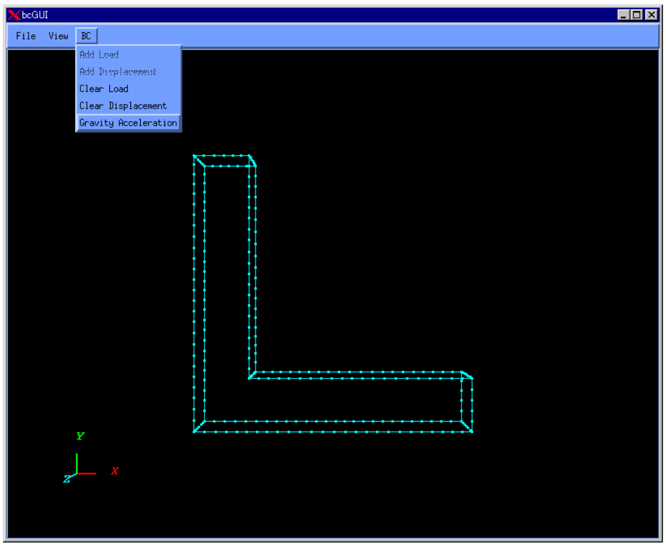


Fig. 9. Setup Menu for Gravity Acceleration

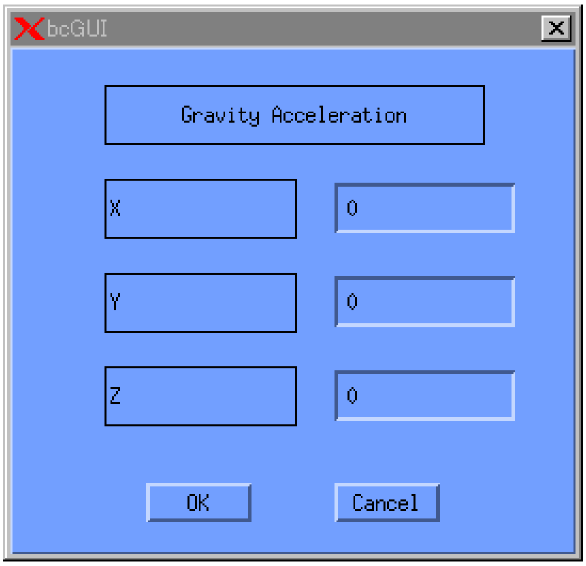


Fig. 10. Dialog Window for Gravity Acceleration Setup

Display of Boundary Conditions

To view the boundary conditions, select the option *Boundary Condition* in the menu *View*. Three options will appear: *None*, *View Load*, and *View Displacement*. To view the load, select the option *View Load* as it is shown in *Fig. 11*. To view the displacement, select the option *View Displacement* as it is shown in *Fig. 12*.

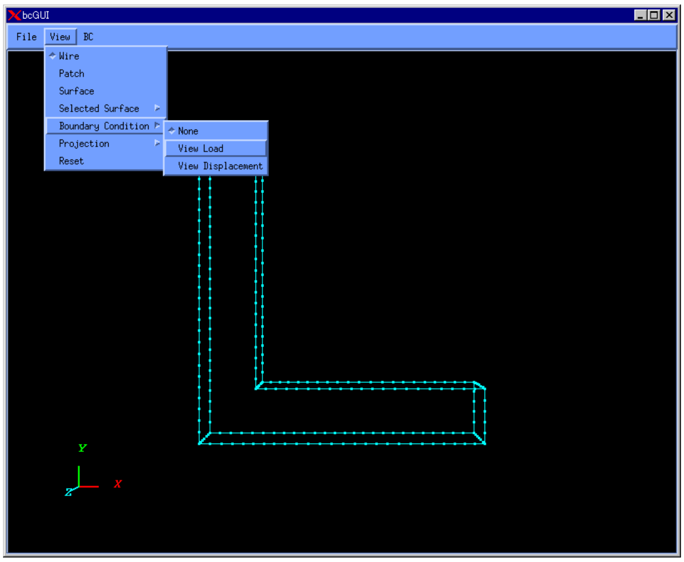


Fig. 11. Menu for Displaying of Load Boundary Conditions

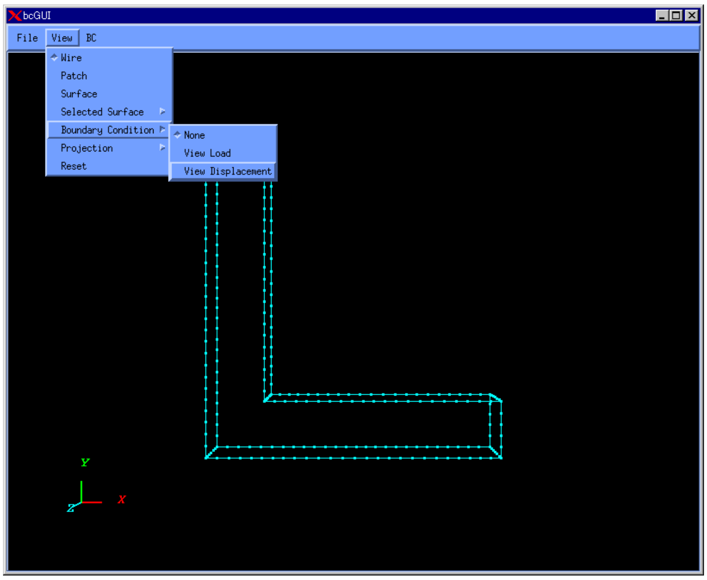


Fig. 12. Menu for Displaying of Displacement Boundary Conditions

Notes

- The load and the displacement cannot be displayed at the same time.
- The surface groups, where the boundary conditions are set, are displayed by lucent colors.
- The boundary conditions for nodes are displayed using vectors.
- Two vectors in opposite directions display each restrained degree-of-freedom of the restrained node.

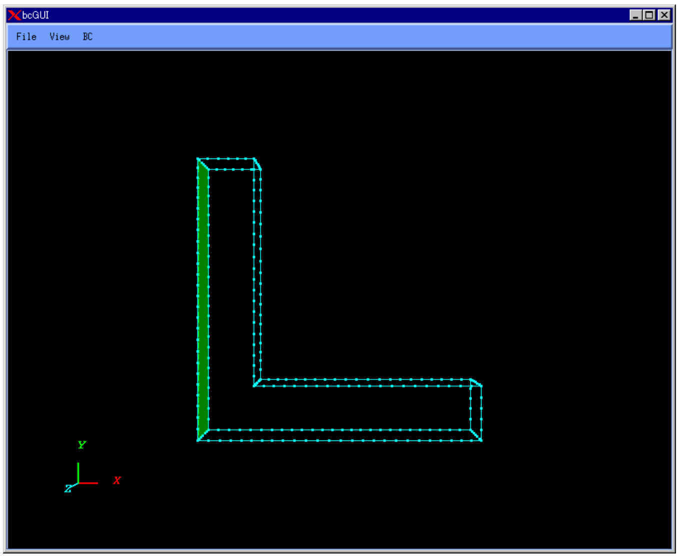


Fig. 13. Displaying of Boundary Conditions for Surface Group

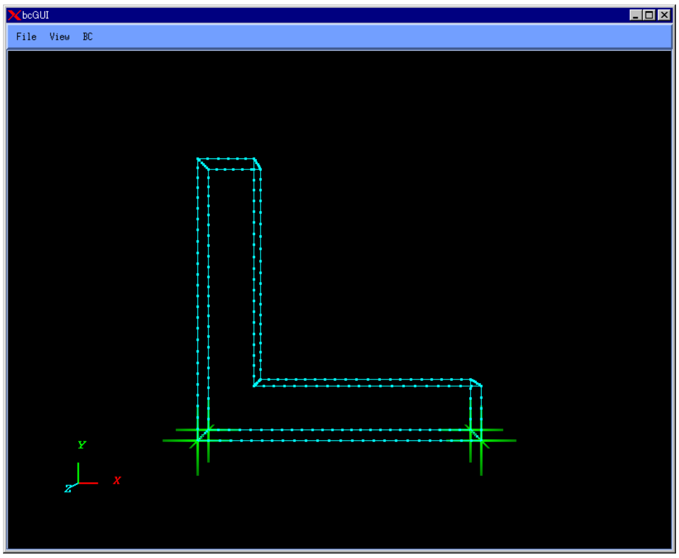


Fig. 14. Displaying of Boundary Conditions for Nodes

Removing of Boundary Conditions

To remove the set boundary conditions, select the option *Clear Load* or *Clear Displacement* in the *BC* menu. To clear all set loads, select the option *Clear Load* as it is shown in *Fig. 15*. To clear all set displacements, select the option *Clear Displacement* as it is shown in *Fig. 16*.

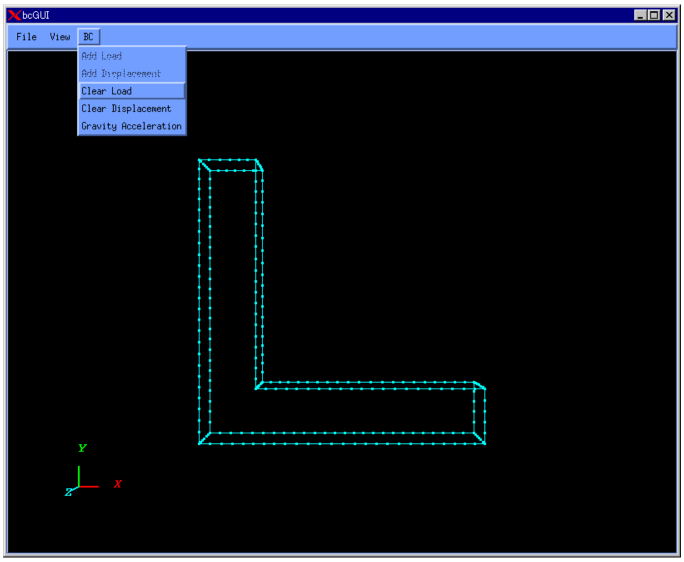


Fig. 15. Removing of All Set Loads

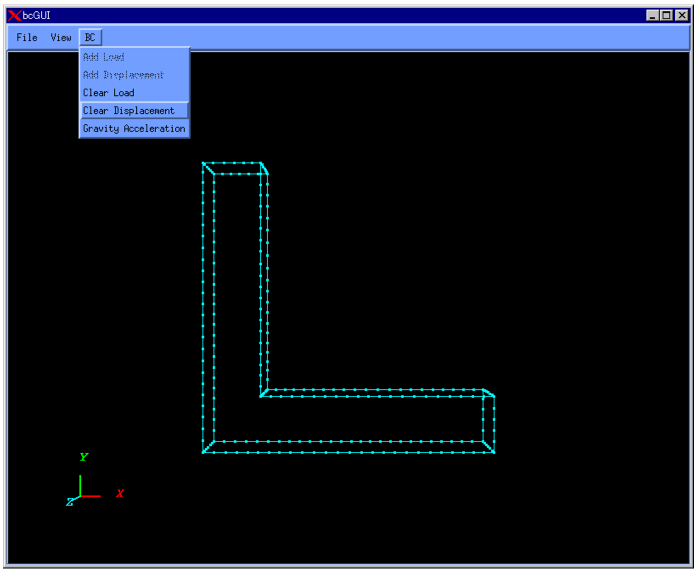


Fig. 16. Removing of All Set Displacements

Output of Boundary Conditions to File

To save the boundary conditions as an analytical conditions file (extension: `.cnd`), select the option *Save Condition* in the menu *File* (*Fig. 17*). The file extension `*.cnd` will be added to the filename automatically (*Fig. 18*).

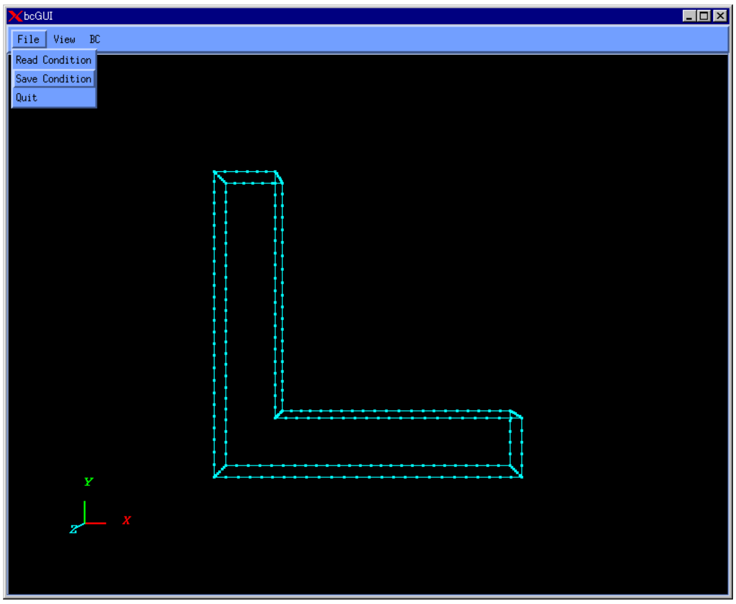


Fig. 17. Output of Boundary Conditions to File

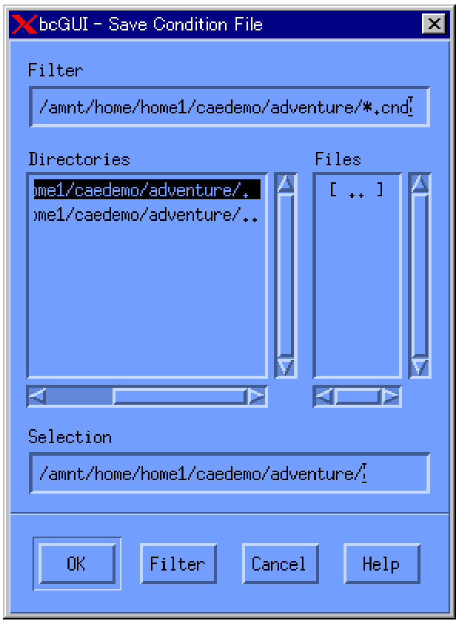


Fig. 18. Specification of Boundary Conditions Output File

Reading of Boundary Conditions from File

To input the boundary conditions from a file, select the option *Read Condition* in the menu *File*. In this case, all boundary conditions set previously will be annulled, and the boundary conditions specified in the file will be set.

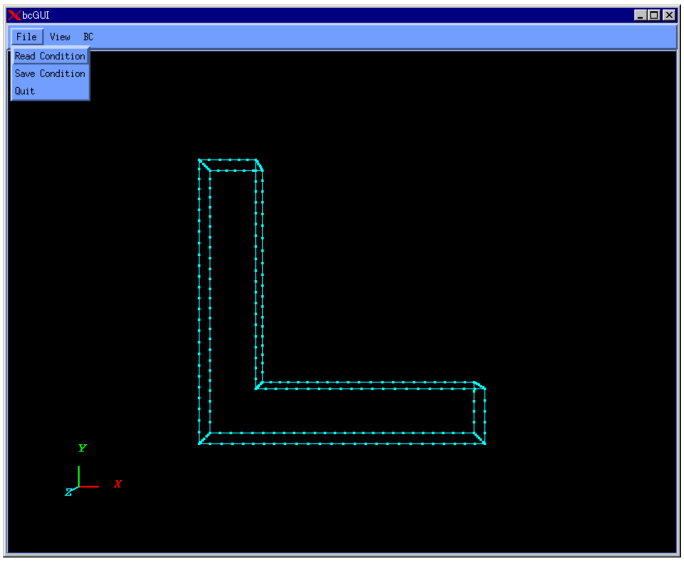


Fig. 19. Reading of Boundary Conditions from File

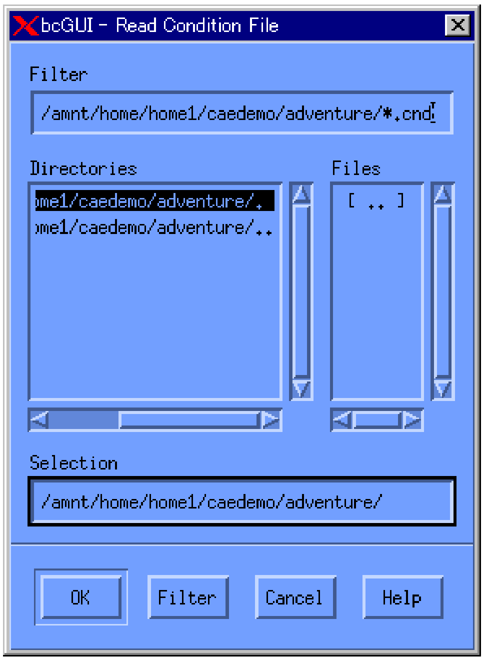


Fig. 20. Menu for Specification of File with Boundary Conditions

Display of Mesh Surface and Surface Layer

To display a mesh surface, select the option *Patch* in the menu *View* (*Fig. 21*).

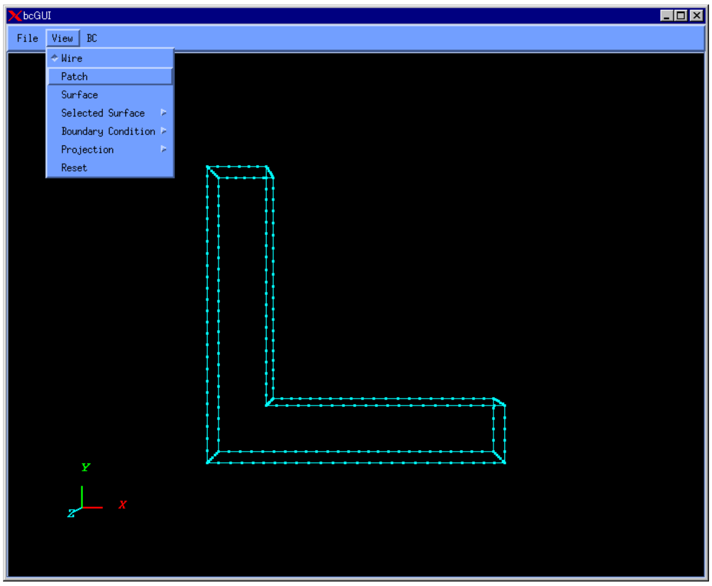


Fig. 21. Switch to Mesh Surface View

Even if the mesh surface is displayed, only primary nodes of face group boundaries can be selected.

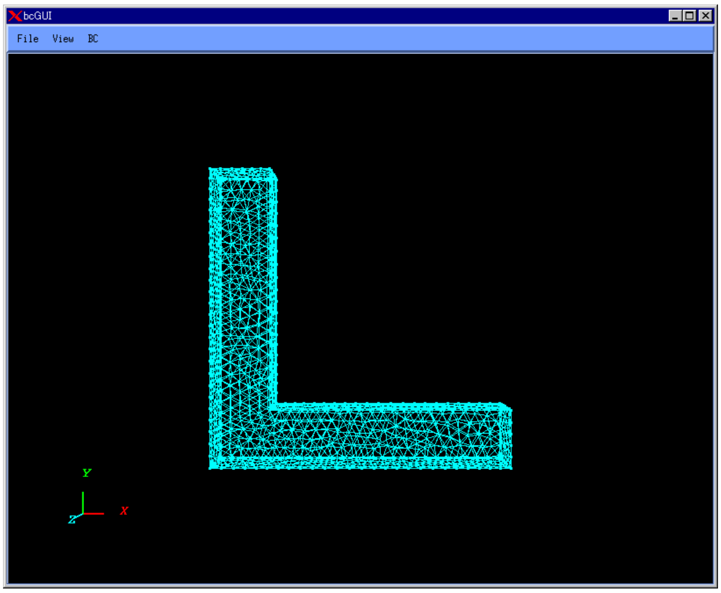


Fig. 22. Mesh surface View

To display surfaces, select the option *Surface* in the menu *View* (*Fig. 23*).

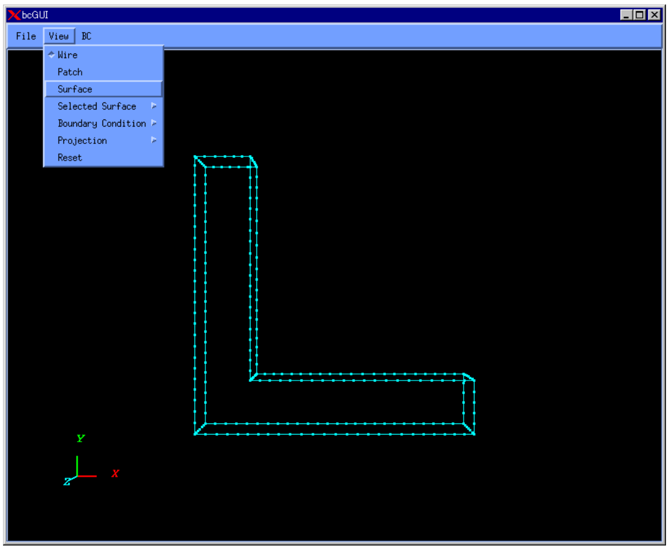


Fig. 23. Switch to Surface Layer View

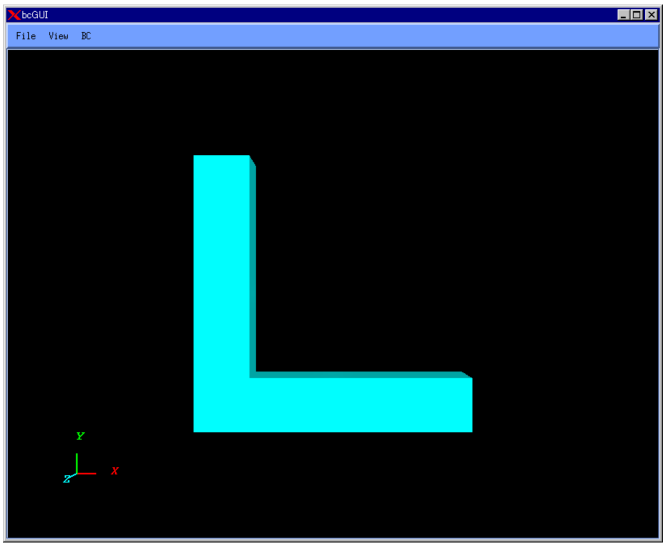


Fig. 24. Surface Layer View

Note

The nodes cannot be selected in the surface layer view.

To display the surface group boundaries (default display), select the option *Wire* in the menu *View* (*Fig. 25*).

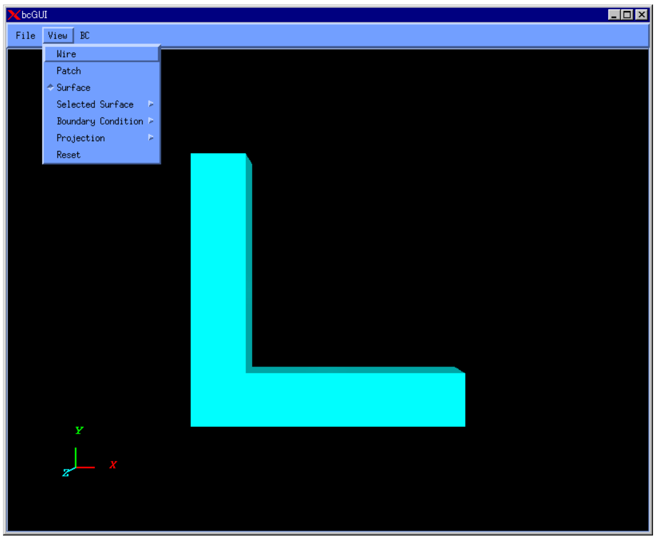


Fig. 25. Switch to Surface Group Boundaries View

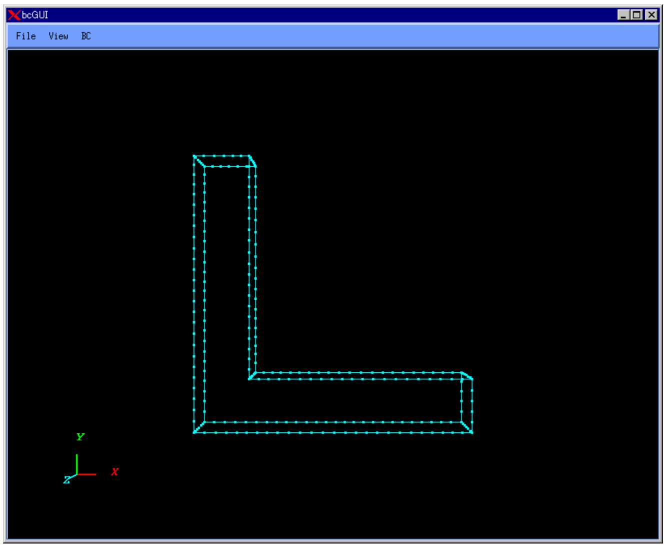


Fig. 26. Surface Group Boundaries View

Changing of Projection Method

To display the object in parallel projection, select the option *Orthographic* in the submenu *Projection* of the menu *View* (*Fig. 27*).

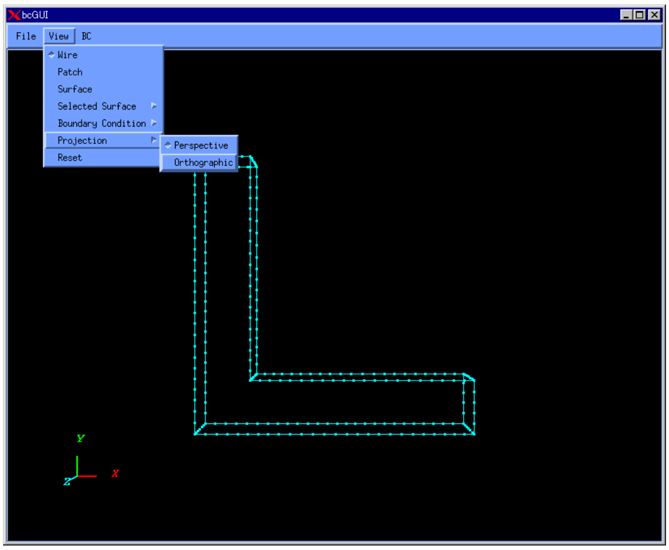


Fig. 27. Switch to Orthographical Projection View

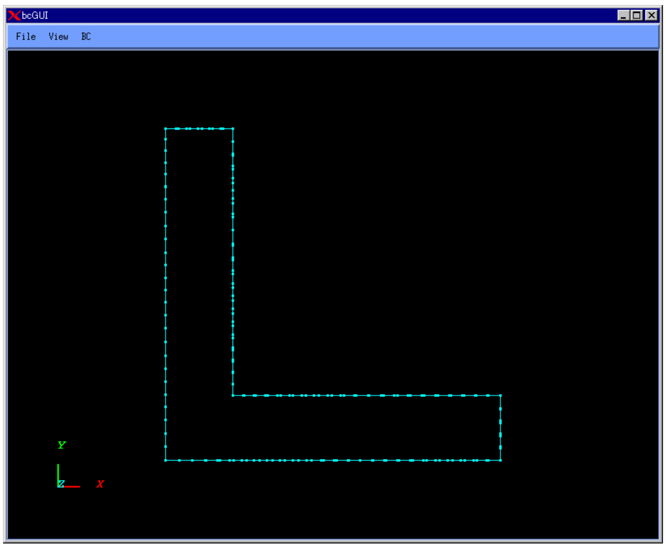


Fig. 28. Orthographical Projection View

To display the object in the perspective projection (default setting), select the option *Perspective* in the submenu *Projection* of the menu *View* (*Fig. 29*).

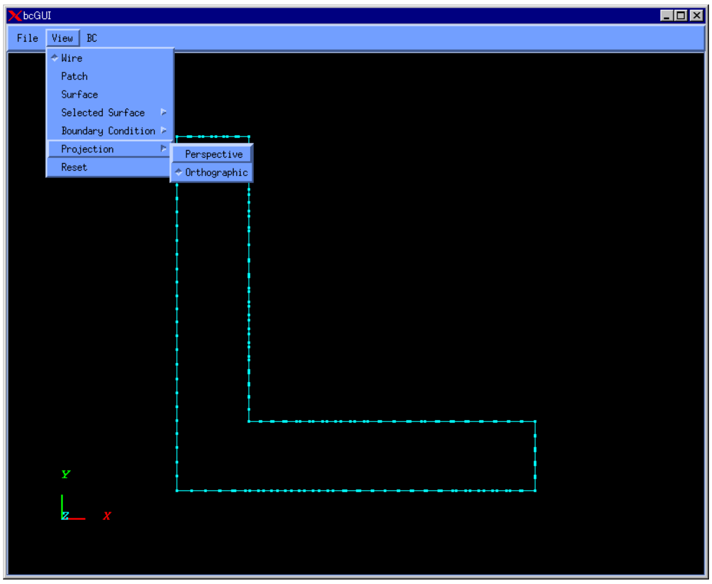


Fig. 29. Switch to Perspective Projection View

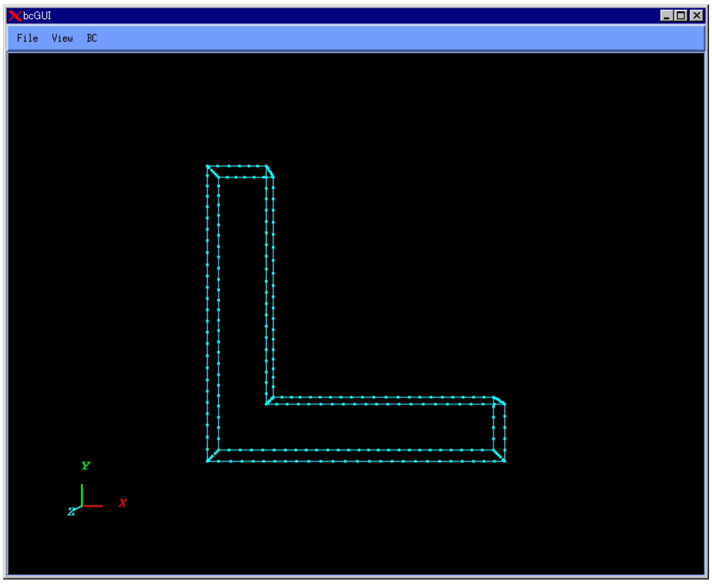


Fig. 30. Perspective Projection View

Displaying of Selected Face Group

To display the selected face group with the patch, choose the option *Patch* in the submenu *Selected Surface* of the menu *View* (*Fig. 31*).

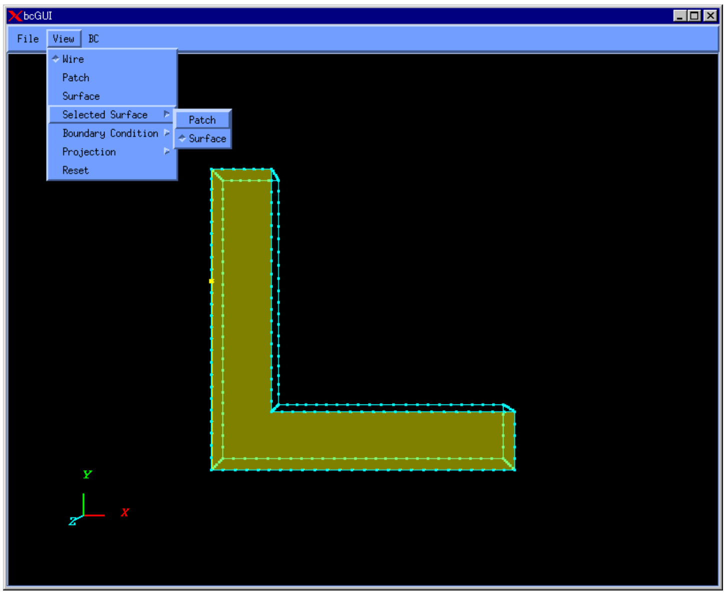


Fig. 31. Switching to Patch View

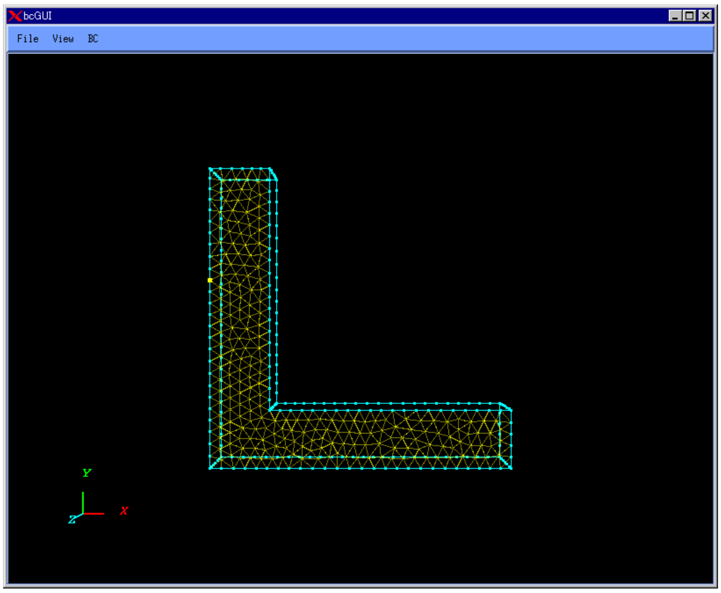


Fig. 32. Patch View of Selected Surface Group

To display the selected surfaces, choose the option *Surface* in submenu *Selected Surface* of the menu *View*. The selected surface group will be displayed by lucent color. This is default display.

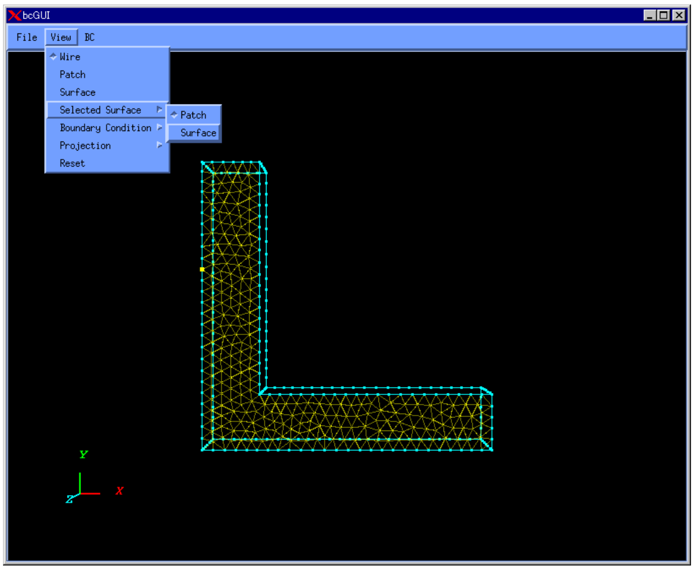


Fig. 33. Switching to Surface View

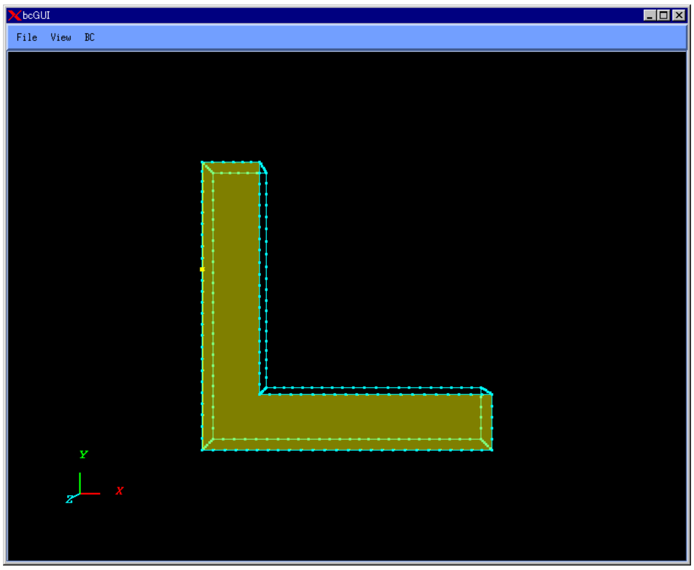


Fig. 34. Selected Surface Group View (Displayed by Lucent Color)

Returning of Observer Viewpoint to Default Settings

To return the observer viewpoint to default settings, choose the option *Reset* in the menu *View*. The observer viewpoint will be returned to its original settings (default view).

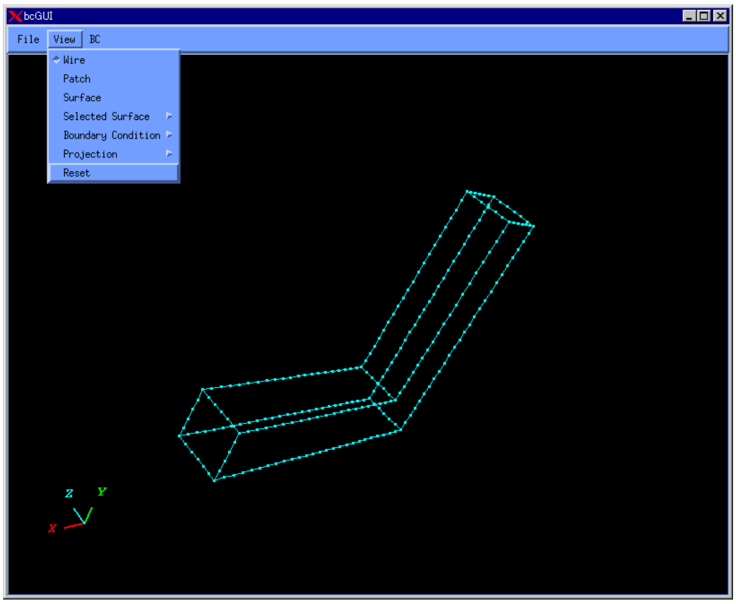


Fig. 35. Resetting of Observer Viewpoint

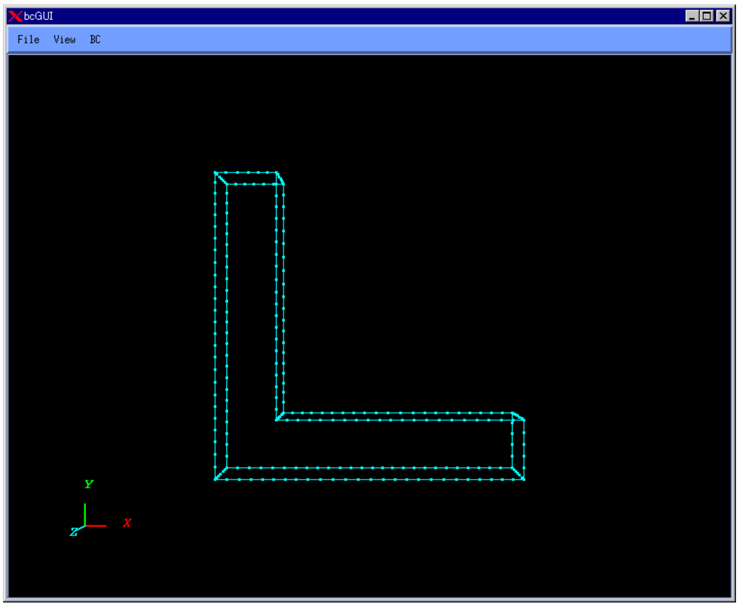


Fig. 36. Default Observer Viewpoint of the Sample Model

5.3. Step 3. Creation of Entire-type FEA Model File

The boundary conditions and material properties attached to mesh can be saved in an entire-type FEA model file of ADVENTURE binary format. At this step, the following input and output files are used.

Input:

Mesh data file (extension is `msh`)
Mesh surface data file (extension is `fgr`)
File with analytical conditions (extension is `cnd`)
Global index file (extension is `trn`)
Material properties data file (extension is `dat`)
Nodal temperature data file (extension is `tmp`)

Output:

Entire-type FEA model file (extension is `adv`)
Entire-type FEA model file (in *ASCII* format) (extension is `fem`)

In order to perform this operation, the command `makefem` is used. The following arguments should be specified with `makefem` in the command line.

`% makefem` *mshFile fgrFile cndFile matFile advFile* [-t *trnFile*]
[*-temperature tmpFile*] [-f *femFile*]

mshFile	:	the name of mesh data file
fgrFile	:	the name of mesh surface data file
cndFile	:	the name of analytical conditions data file
matFile	:	the name of material properties data file
advFile	:	the name of entire-type FEA model file
trnFile	:	the name of global index file*
tmpFile	:	the name of nodal temperature data file**
femFile	:	the name of entire-type FEA model file in *ASCII* format***

Notes

Arguments in the parentheses [………] can be omitted.
* Specify *trnFile* when you create *cndFile* by `bcGUI`.
** The file *tmpFile* is needed only for thermal stress FEA.
*** The file *femFile* is used for debugging and usually not necessary being specified by the user.

- The files with material properties data and nodal temperature data can be created using any available text editor.
- If the model consists of several parts with different material properties, the correspondence between the material properties data and volumes (domains) should be checked using the procedures, which will be described below.

Displaying of Multi-Material Models

(1). If the model consists of several volumes with different material properties, use shell script `msh2pcm` to convert the model.

`% msh2pcm` *mshFile*

where *mshFile* is the name of mesh data file.
If the model data file is `Model.msh`, two files will be created:

`Model_V.pcm`: the surface patch data file
`Model_V.pcg`: the surface patch group data file

(2). Start the `bcGUI` module to display the volumes.

`% bcGUI` *pcmFile* *pcgFile*

where *pcmFile* is the surface patch data file and *pcg*File is the surface patch group data file.

If the first argument (*pcmFile*) is set, `bcGUI` will be started in the *Volume View* mode. In this mode, the nodes cannot be selected and the boundary conditions cannot be set. The volumes can be selected using keyboard keys. Use *n* to display the next volume and *p* to display the previous volume. The volume number will be shown in the left upper part of the screen.

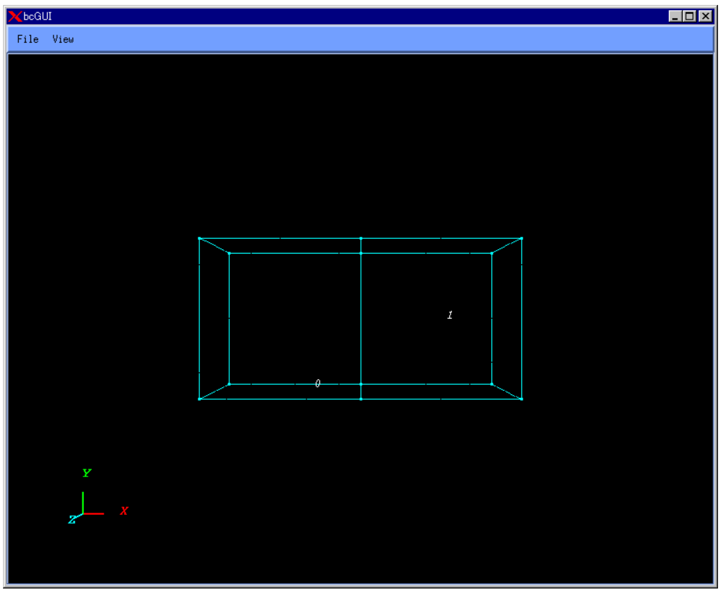
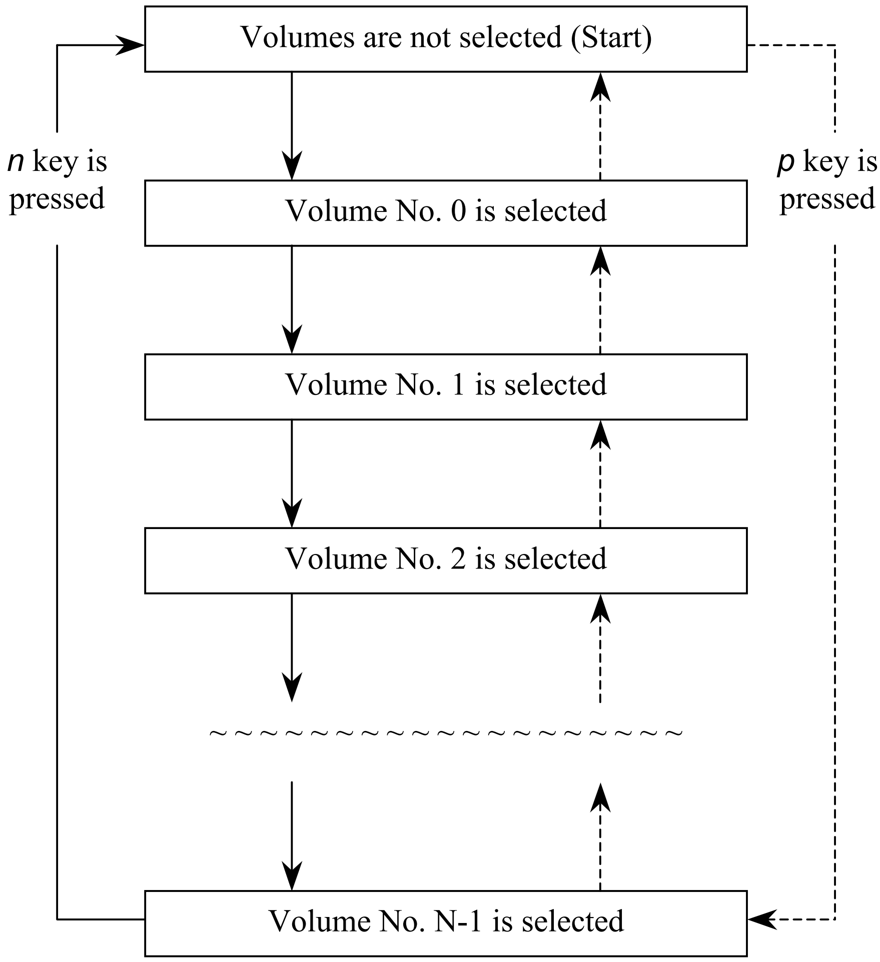


Fig. 37. View Mode for Volumes


Volumes are not selected (Start)
n key is pressed
p key is pressed
Volume No. 0 is selected
Volume No. 1 is selected
Volume No. 2 is selected
~ ~
Volume No. N-1 is selected

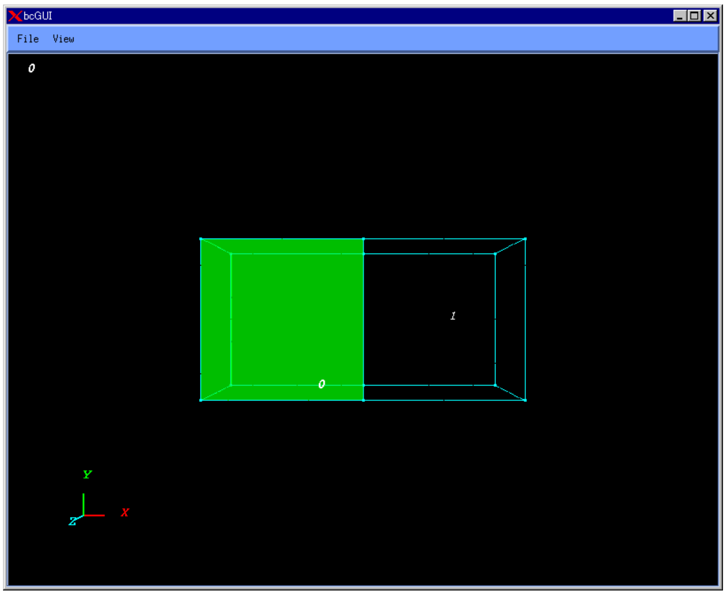


Fig. 38. Volume No. 0 Is Selected

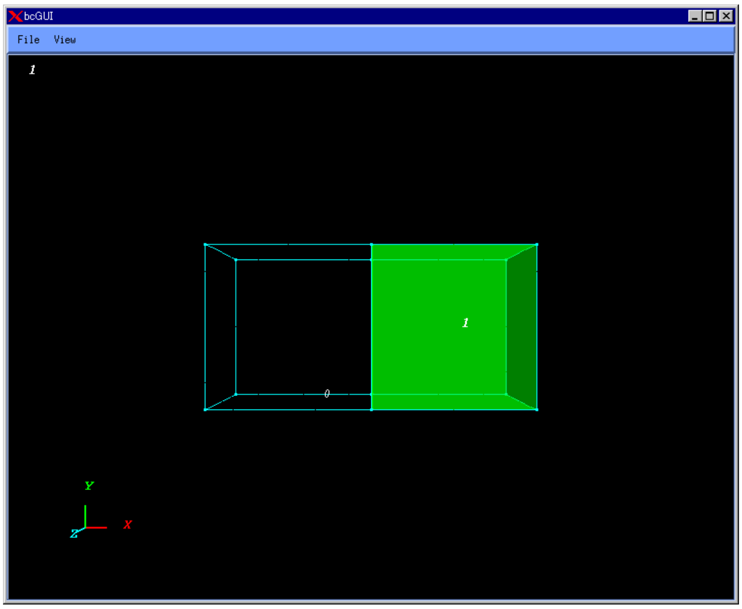
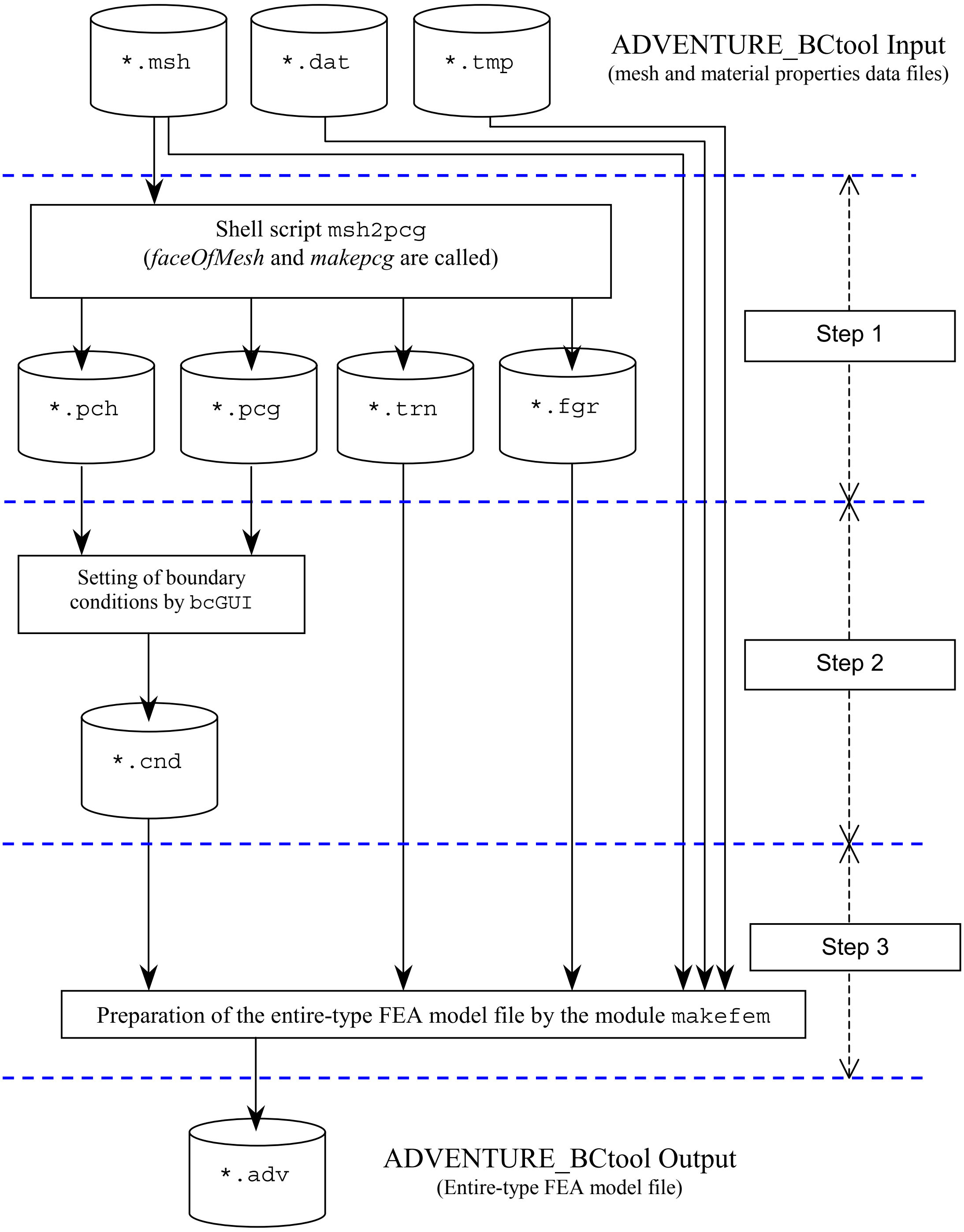


Fig. 39. Volume No. 1 Is Selected

5.4. Flowchart of Data Processing

The data processing flow occurring from Step 1 to Step 3 discussed in *Chapters 5.1*, *5.2*, and *5.3* is shown below.



6. File Formats

6.1. Mesh Data File (extension .msh)

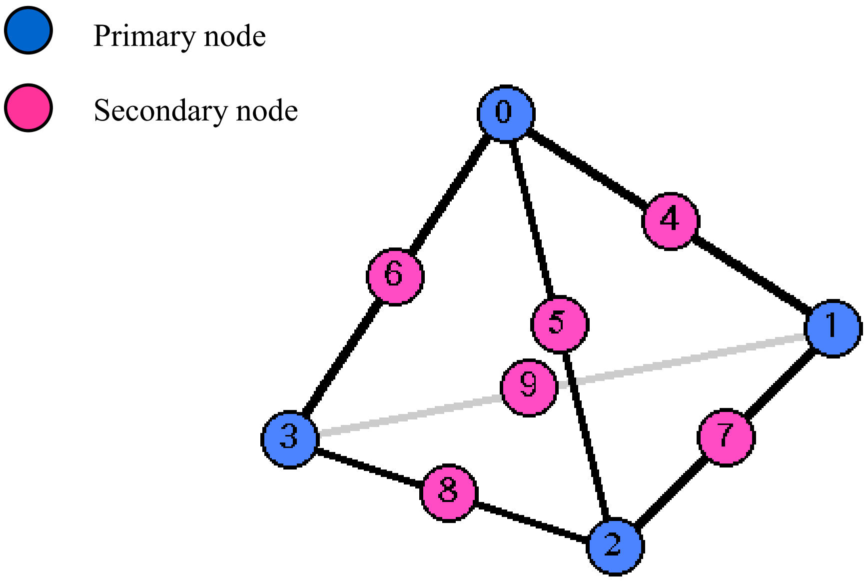
Format of mesh data file (for linear tetrahedron)

```
23441                                        ← Number of elements
2770    2610    2600    3480                 ← Nodes, which compose 0 element*
3480    2770    2610    3490                 ← Nodes, which compose 1st element
23      22      310     24
22      21      310     24
21      5       310     24

       ===== Omitted =====

4703    4696    4702    4830
4707    4703    4702    4830
4727    4703    4708    4830
4708    4703    4707    4830
4732    4726    4727    4830                 ← Nodes that compose 23441-1th element
4831                                         ← Number of nodes
-71.576560    -1.614198    0.000000          ← X, Y, and Z coordinates of 0 node
-71.576560    -1.614198    1.381617          ← X, Y, and Z coordinates of 1st node
-71.576560    -1.614198    2.722493
-71.576560    -1.614198    4.030211
-71.576560    -1.614198    5.300060

       ===== Omitted =====

-55.506172    3.848006    9.440700
-53.050143    0.879802    4.377307
-54.576493    4.474329    8.997737
-53.896503    7.741545    9.582692
-30.521942    6.716313    7.342463           ← Coordinates of X, Y, and Z of 4381-1th node
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
 2        ← Number of volumes                               ⎫
 11720    ← Number of elements in Volume No. 0              ⎪
 0        ← 0 element of Volume No. 0                       ⎪
 1        ← 1st element of Volume No. 0                     ⎪
                                                            ⎪
       ===== Omitted =====                                  ⎪   Only for cases with
                                                            ⎬   multi-material data
 11719    ← 11720-1th element of Volume No. 0               ⎪
 11721    ← number of elements in Volume No. 1.             ⎪
 11720    ← 0 element of Volume No. 1                       ⎪
                                                            ⎪
       ===== Omitted =====                                  ⎪
                                                            ⎪
 23440    ← 11720-1th element of Volume No. 1               ⎭
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
```

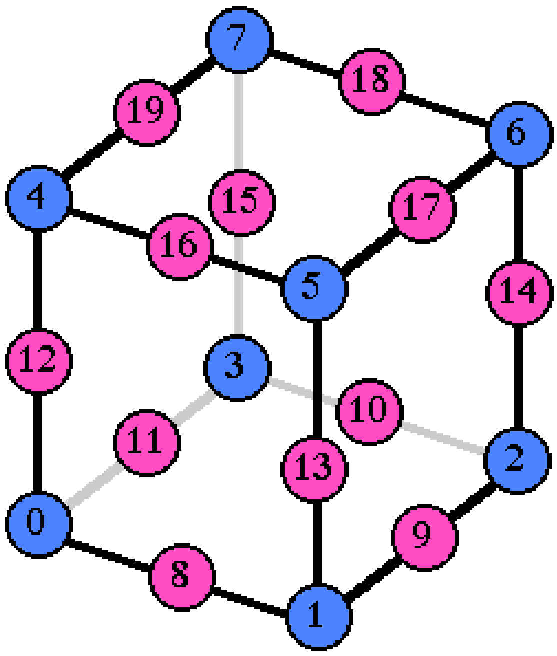
* Number of nodes, which compose elements:

- Linear tetrahedral element consists of 4 nodes.
- Quadratic tetrahedral element consists of 10 nodes.
- Linear hexahedral element consists of 8 nodes.
- Quadratic hexahedral element consists of 20 nodes.

The following figures show the connectivity of nodes in elements.



Tetrahedral Element



Hexahedral Element

6.2. Mesh Surface Data File (extension .fgr)

Format of mesh surface data file (for linear tetrahedral element)

```
4                          ← Type of element*
8                          ← Number of surface groups
470                        ← Number of surfaces, which compose 0 surface group
2    23    22    24        ← Information on 0 surface of 0 surface group**
3    22    21    24        ← Information on 1st surface of 0 surface group

===== Omitted =====

2313 268   267   263
2314 269   268   264       ← Information on 470-1th surface of 0 surface group
1115                       ← Number of surfaces, which compose 1st surface group
19    79    440    80      ← Information on 0 surface of 1st surface group
838   435   80     440     ← Information on 1st surface of 1st surface group

===== Omitted =====

22998 4798 4792 4772
22963 4792 4787 4772       ← Information on 1115-1th surface of 1st surface group

===== Omitted =====

39                         ← Number of surfaces, which compose 8-1th surface group
22942 4785 4790 4786       ← Information on 0 surface of 8-1th surface group
22962 4786 4790 4791       ← Information on 1st surface of 8-1th surface group

===== Omitted =====

23032 4806 4807 4800
23027 4798 4805 4800       ← Information on 39-1th surface of 8-1th surface group
```

* Type of element:

Linear tetrahedron	:	4
Quadratic tetrahedron	:	10
Linear hexahedron	:	8
Quadratic hexahedron	:	20

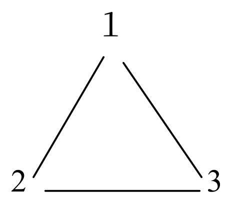
** Information on surface

Here, "surface" means a triangular surface or a quadrangular surface composed by nodes. The first numeric value is corresponded to the element's number that belongs to the surface. The remained are the node numbers, which compose the surface. The number of nodes, which compose the surfaces, is:

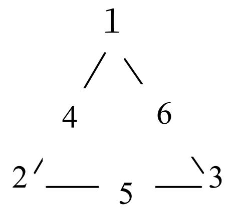
Linear tetrahedron	:	3
Quadratic tetrahedron	:	6
Linear hexahedron	:	4
Quadratic hexahedron	:	8

Numerical orders of the nodes composing the elements are presented in the following figures for each type of elements.

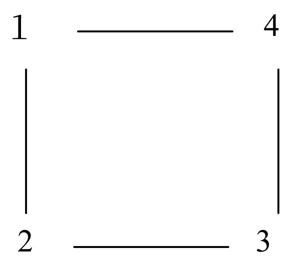
Linear Tetrahedron



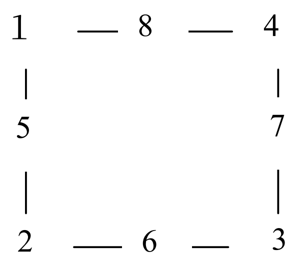
Quadratic Tetrahedron



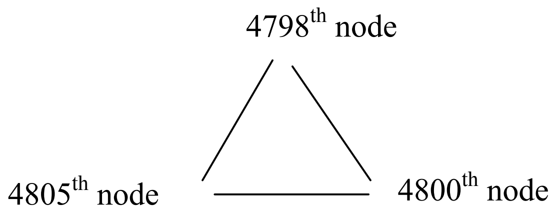
Linear Hexahedron



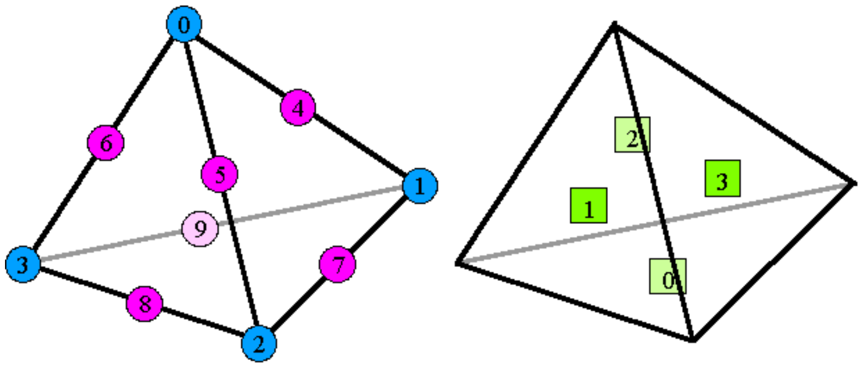
Quadratic Hexahedron



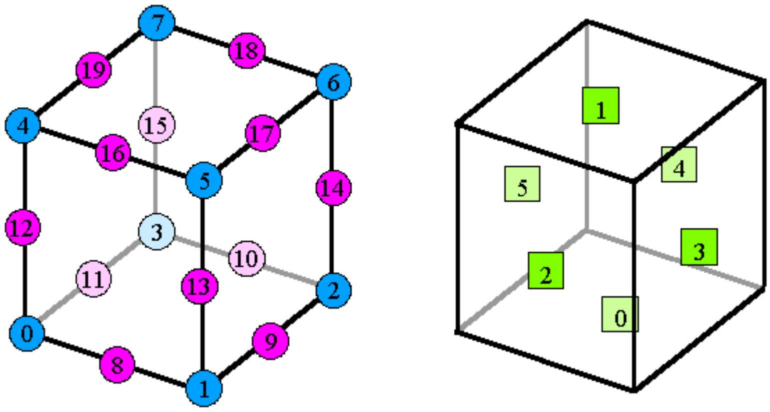
The last line of the above-mentioned file format (`23027 4798 4805 4800`) is corresponded to the triangle of the `23027`th element shown below.



Numbering of surfaces in the elements is shown in the following figures.



Surface	Nodes		
0	1	2	3
1	0	3	2
2	0	1	3
3	0	2	1



Surface	Nodes			
0	0	3	2	1
1	4	5	6	7
2	0	1	5	4
3	1	2	6	5
4	2	3	7	6
5	0	4	7	3

6.3. *Extracted Surface Mesh Data File (extension `.pch`)*

Format of patch data file (for linear tetrahedron)

```
2163                              ← Number of nodes
-71.5766  -1.6142  0              ← X, Y, and Z coordinates of 0 node
-71.5766  -1.6142  1.38162        ← X, Y, and Z coordinates of 1st node
-71.5766  -1.6142  2.72249
-71.5766  -1.6142  4.03021
-71.5766  -1.6142  5.30006

===== Omitted =====

-21.3207  8.1324  2
-21.3207  8.1324  4
-21.3207  8.1324  6
-21.3207  8.1324  8
-21.3207  8.1324  10              ← X, Y, and Z coordinates of 2163-1th node
4322                              ← Number of patches
23  22  24                        ← Nodes, which compose 0 patch*
22  21  24                        ← Nodes, which compose 1st patch
25  23  24
44  22  23
21  5   24

===== Omitted =====

2153  2152  2159
2150  2152  2151
2158  2159  2152
2150  2157  2152
2157  2158  2152                  ← Nodes, which compose 4322-1th patch
```

* The patch is composed from the mesh surface primary nodes.

Linear tetrahedral element:	3 nodes
Quadratic tetrahedral element:	3 nodes
Linear hexahedral element:	4 nodes
Quadratic hexahedral element:	4 nodes

The node number is an index of the primary node of the mesh surface.
The connectivity of nodes is assumed in the clockwise direction if we look at the element from the outside.

6.4. *Patch Group Data File (extension* `.pcg`*)*

Format of patch group data file

`#mainVertexInfo`	
`mainVertexN 299`	← Number of main nodes*
`0`	← 0 main node
`1`	← 1st main node
===== Omitted =====	
`10`	
`27`	
===== Omitted =====	
`2161`	
`2162`	← **299-1**th main node
`#edgeGroupInfo`	
`edgeGroupN 305`	← Number of edge groups
`edgeGroup 2`	← Number of nodes, which compose 0 edge group
`0`	← 0 node of 0 edge group
`1`	← 1st node of 0 edge group
`edgeGroup 2`	← Number of nodes, which compose 1st edge group
`0`	← 0 node of 1st edge group
`10`	← 1st node of 1st edge group
===== Omitted =====	
`edgeGroup 2`	← Number of nodes, which compose **305-1**th edge group
`9`	← 0 node of **305-1**th edge group
`30`	← 1st node of **305-1**th edge group
`#faceGroupInfo`	
`faceGroupN 8`	← Number of face groups
`faceGroup 470`	← Number of patches, which compose 0 face group
`0`	← Number of 0 patch of 0 face group
`1`	← Number of 1st patch of 0 face group
===== Omitted =====	
`469`	← Number of **470-1**th patch of 0 face group
===== Omitted =====	
`faceGroup 39`	← Number of patches, which compose **8-1**th face group
`4283`	← Number of 0 patch of **8-1**th face group
`4284`	← Number of 1st patch of **8-1**th face group
===== Omitted =====	
`4321`	← Number of **39-1**th patch of **8-1**th face group

* The main node is a representative node, which characterizes a shape of model. In ADVENTURE_BCtool, a primary node of the surface group boundary is assigned for main node.

6.5. Global Index File (extension `.trn`)

Format of global index file

```
0
1
2
3
4
5
6
7
8
9

===== Omitted =====

4809
4810
```

The number of lines is the same as the number of primary nodes on the mesh surfaces.

The global index file contains tabulated data where the node number of the surface patch data file (extension `pch`) is converted to the node number of the mesh data file (extension `msh`). The line number *N* in a global index system corresponds to the line number *N-1* of a local index system.

6.6. Boundary Condition File (extension `.cnd`)

File format of boundary conditions file

```
gravity  0  0  -9.8              ← X, Y, and Z components of gravity
boundary  12                     ← Number of boundary conditions data
loadOnVertex  271  0  10.5       ← Load on node 271 in X direction is 10.5
loadOnVertex  271  1  10.5       ← Load on node 271 in Y direction is 10.5
loadOnVertex  271  2  10.5       ← Load on node 271 in Z direction is 10.5
dispOnVertex  8   0  0           ← 8th node displacement in X direction is 0
dispOnVertex  8   1  0           ← 8th node displacement in Y direction is 0
dispOnVertex  8   2  0           ← 8th node displacement in Z direction is 0
loadOnFaceGroup  1  1  5.2       ← Surface group 1 has vertical load of 5.2
loadOnFaceGroup  2  0  2  -1.1   ← Load on surface group 2 in Z direction is -1.1
dispOnFaceGroup  3  0  0   0     ← Load on surface group 3 in has X direction is 0
dispOnFaceGroup  3  0  1   0     ← Load on surface group 3 in Y direction is 0
dispOnFaceGroup  3  0  2   0     ← Load on surface group 3 in Z direction is 0
dispOnFaceGroup  4  1  1       ← Displacement in vertical direction of surface group 4 is 1
```

`loadOnVertex` is the load on node.
`dispOnVertex` is the displacement of node.
`loadOnFaceGroup` is the load on surface group.
`dispOnFaceGroup` is the displacement of surface group.

The meanings of three numbers following `loadOnVertex` and `dispOnVertex` are:

1 - Node number
2 - 0 : X component, 1 : Y component, 2 : Z component
3 - Value of load or displacement

The meanings of numbers following `loadOnFaceGroup` and `dispOnFaceGroup` are:

1 - Surface group number
2 - 0 : X, Y, and Z directions are specified, 1: Direction, which is vertical to surface;
3 - If the X, Y, and Z directions are selected, the components in X, Y, and Z directions are specified (0: X component, 1: Y component, 2: Z component).
If the direction vertical to surface is selected, the values of load or displacement are specified.
4 - Load or displacement are specified in X, Y, and Z directions

6.7. Material Properties Data File (extension `.dat`)

Property	*Name of Label*	*Remarks*
Young's Modulus	`YoungModulus`	
Poisson's Ratio	`PoissonRatio`	
Hardening Parameter	`HardeningParameter`	Elastic-plastic FEA
Yield Stress	`YieldStress`	Elastic-plastic FEA
Density	`Density`	Body force FEA
Thermal Expansion Coefficient	`ThermalExpansionCoefficient`	Thermal stress FEA
Reference Temperature	`ReferenceTemperature`	Thermal stress FEA

Format of data file with material properties (single-material model)

```
YoungModulus          21000.0          ← Young's Modulus
PoissonRatio          0.4              ← Poisson Ratio
HardeningParameter    1000.0           ← Hardening Parameter
YieldStress           500.0            ← Yield Stress
Density               760.0            ← Material Density
```

Format of data file with material properties (multi-material model)

```
#materialInfo
materialN    2                ← Number of materials
propertyN    2                ← Number of defined properties
YoungModulus     21000.0    } 2 properties are defined  }
PoissonRatio     0.4                                    } 2 materials are defined
YoungModulus     205940.0                               }
PoissonRatio     0.3                                    }
#volumeInfo
volumeN     3                 ← Number of volumes
1                             ← Material number of Volume No. 0
0                             ← Material number of Volume No. 1
1                             ← Material number of Volume No. 2
```

- The user should create these data files in a text format.
- The program will determine automatically whether the file contains the data for a single material or for a multi-material model.
- It is allowed that the material properties data contain materials with IDs, which are not used in the current analysis.
- If the material properties data, which have a format for a single-material model is applied to multi-material model (multi-volume model), the model will be treated as a single-material model. All volumes will have the same material properties.
- If the material properties data, which have a format for a multi-material model is applied to a single-material model (single-volume model), the model will be treated as a single-material model and the material properties No. 0 will be applied.
- Prior to assigning of material properties data to volumes, check their correspondences as it is described in *Section 5.3*.

6.8. Nodal Temperature Data File (extension `.tmp`)

Format of file with nodal temperature data

```
3.00e+02
3.00e+02
3.00e+02
3.00e+02
3.00e+02
3.00e+02
3.00e+02      } Number of rows is equal to the number of nodes
3.00e+02
3.00e+02
3.00e+02
====Ommitted====
3.00e+02
3.00e+02
```

- These data should be provided for a thermal stress analysis.

6.9. Entire-type FEA Model File (extension `.adv`)

As a final output, ADVENTURE_BCtool creates an entire-type analysis model file in the ADVENTURE binary format. This file cannot be viewed without reprocessing by a viewer program supplied with ADVENTURE system. The entire-type analysis model file contains the data on element, node, displacement, load, gravity, Young's modulus, Poisson's ratio, hardening parameter, yield stress, density of material, thermal expansion coefficient, reference temperature, nodal temperature, and material number (for multi-material case). Examples of contents stored in the entire-type FEA model file will be given below.

Element

```
[Properties]
content_type=Element
num_items=23441                     ← Number of elements
format=i4i4i4i4           ← "i4" is repeated for the number of nodes in each element
num_nodes_per_element=4             ← Number of nodes in each element
element_type=3DLinearTetrahedron ←     Linear tetrahedral element or
dimension=3                            Quadratic tetrahedral element, or
index_byte=4                           Linear hexahedral element, or
                                       Quadratic hexahedral element
[Data]
2770  2610  2600  3480
3480  2770  2610  3490
23  22  310  24
22  21  310  24
21  5   310  24                  Information on the nodes, which compose the
                                 element (the same as in the Mesh data file).
===== Omitted =====

4727  4703  4708  4830
4708  4703  4707  4830
4732  4726  4727  4830
```

Node

```
[Properties]
content_type=Node
num_items=4831                    ← Number of nodes
format=f8f8f8
dimension=3

[Data]
-7.157656e+01 -1.614198e+00 0.000000e+00
-7.157656e+01 -1.614198e+00 1.381617e+00
-7.157656e+01 -1.614198e+00 2.722493e+00
-7.157656e+01 -1.614198e+00 4.030211e+00
-7.157656e+01 -1.614198e+00 5.300060e+00

===== Omitted =====

-5.550617e+01 3.848006e+00 9.440700e+00
-5.305014e+01 8.798020e-01 4.377307e+00
-5.457649e+01 4.474329e+00 8.997737e+00
-5.389650e+01 7.741545e+00 9.582692e+00
-3.052194e+01 6.716313e+00 7.342463e+00
```

(Data: It is the same as coordinates of nodes in the mesh data file.)

Displacement Boundary Conditions

```
[Properties]
content_type=FEGenericAttribute
num_items=816                    ← Number of displacements data
fega_type=NodeVariable
label=ForcedDisplacement
format=i4f8
index_byte=4

[Data]
0  0  0.000000e+00
0  1  0.000000e+00
0  2  0.000000e+00

===== Omitted =====

271  2  0.000000e+00
```

(Data: 1st value : Node number; 2nd value : X, Y, Z are shown by 0, 1, 2; 3rd value : Displacement)

Load Boundary Conditions

```
[Properties]
content_type=FEGenericAttribute
num_items=305                          ← Number of load datasets
fega_type=NodeVariable
label=Load
format=i4f8
index_byte=4

[Data]
2725 0 -1.372480e+00
2726 0 -1.069377e+00
2727 0 -2.261483e+00

===== Omitted =====

3245 0 -2.684982e+01
3246 0 -2.329293e+01
3247 0 -1.423615e+01
```

1st value : Node number
2nd value : X, Y, Z are shown by 0, 1, 2.
3rd value : Load

Gravity

```
[Properties]
content_type=FEGenericAttribute
num_items=1
fega_type=AllElementConstant
label=GravityAcceleration
format=f8f8f8
index_byte=4

[Data]
0.000000e+00 0.000000e+00 -9.800000e+00   ← X, Y, and Z components of gravity
```

Young's Modulus

```
[Properties]
content_type=FEGenericAttribute
num_items=1
fega_type=AllElementConstant
label=YoungModulus
format=f8
material_id=0                 ← Material ID
index_byte=4

[Data]
2.100000e+04                  ← Young's modulus
```

Poisson's Ratio

```
[Properties]
content_type=FEGenericAttribute
num_items=1
fega_type=AllElementConstant
label=PoissonRatio
format=f8
material_id=0                ← Material ID
index_byte=4

[Data]
4.000000e-01                 ← Poisson's ratio
```

Hardening Parameter

```
[Properties]
content_type=FEGenericAttribute
num_items=1
fega_type=AllElementConstant
label=HardeningParameter
format=f8
material_id=0                ← Material ID
index_byte=4

[Data]
1.000000e+03                 ← Hardening Parameter
```

Yield Stress

```
[Properties]
content_type=FEGenericAttribute
num_items=1
fega_type=AllElementConstant
label=YieldStress
format=f8
material_id=0                ← Material ID
index_byte=4

[Data]
5.000000e+02                 ← Yield Stress
```

Density

```
[Properties]
content_type=FEGenericAttribute
num_items=1
fega_type=AllElementConstant
label=Density
format=f8
material_id=0                 ← Material ID
index_byte=4

[Data]
7.600000e+02                  ← Density of material
```

Thermal Expansion Coefficient

```
[Properties]
content_type=FEGenericAttribute
num_items=1
fega_type=AllElementConstant
label=ThermalExpansionCoefficient
format=f8
material_id=0                 ← Material ID
index_byte=4

[Data]
1.000000e-06                  ← Thermal expansion coefficient
```

Reference Temperature

```
[Properties]
content_type=FEGenericAttribute
num_items=1
fega_type=AllElementConstant
label=ReferenceTemperature
format=f8
material_id=0                 ← Material ID
index_byte=4

[Data]
3.000000e+02                  ← Reference temperature
```

Nodal Temperature

```
[Properties]
content_type=FEGenericAttribute
num_items=4831                          ← Number of nodes in Mesh data file
fega_type=AllNodeVariable
label=Temperature
format=f8
index_byte=4

[Data]
3.000000e+02                            ← Temperature of node No. 1
3.000000e+02                            ← Temperature of node No. 2

===== Omitted =====

3.000000e+02                            ← Temperature of node No. 4830
```

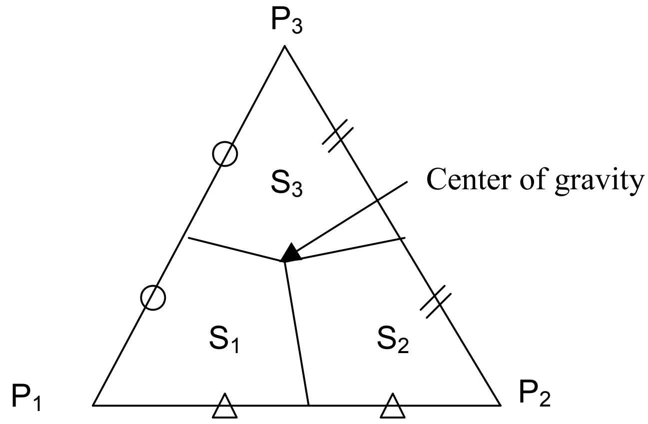
Material ID

```
[Properties]
content_type=FEGenericAttribute
num_items=23441                         ← Number of nodes in Mesh data file
fega_type=AllElementVariable
label=MaterialID
format=i4
index_byte=4

[Data]
1                                       ← Material ID of Element No. 0
1                                       ← Material ID of Element No. 1

===== Omitted =====

0                                       ← Material ID of Element No. 23439
0                                       ← Material ID of Element No. 23440
```

7. Conversion of Surface Load to Nodal Load

ADVENTURE_BCtool converts the load on surface groups (specified by a value of load per unit area) into the nodal load. The method of conversion is described in the following sections for different types of elements.

7.1. Linear Tetrahedral Element



If the load distributions f_1~f_3 are given for the surface group areas S_1~S_3 respectively,

the load L_1 concentrated on the node P_1 will be

$L_1 = S_1 \times f_1$

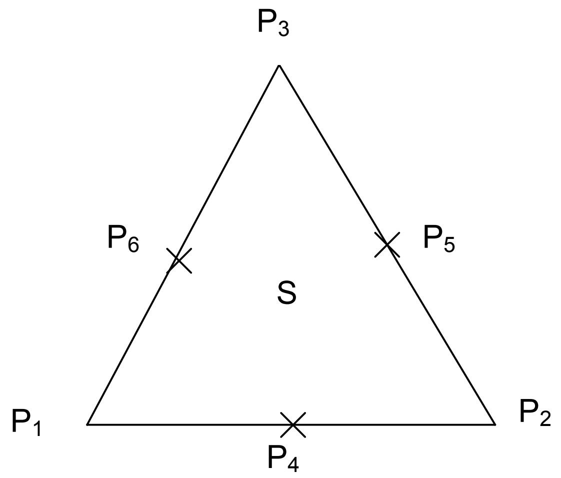
the load L_2 concentrated on the node P_2 will be

$L_2 = S_2 \times f_2$

the load L_3 concentrated on the node P_3 will be

$L_3 = S_3 \times f_3$

7.2. Quadratic Tetrahedral Element



If the load distributions f_1~f_3 are given for the area of the triangular surface group S, the load L_1 concentrated on the node P_1 will be

$L_1 = - f_1 \times S \times (1/9)$

the load L_2 concentrated on the node P_2 will be

$L_2 = - f_3 \times S \times (1/9)$

the load L_3 concentrated on the node P_3 will be

$L_3 = - f_3 \times S \times (1/9)$

the load L_4 concentrated on the node P_4 will be

$L_4 = (f_1 + f_2) \times (1/2) \times S \times (4/9)$

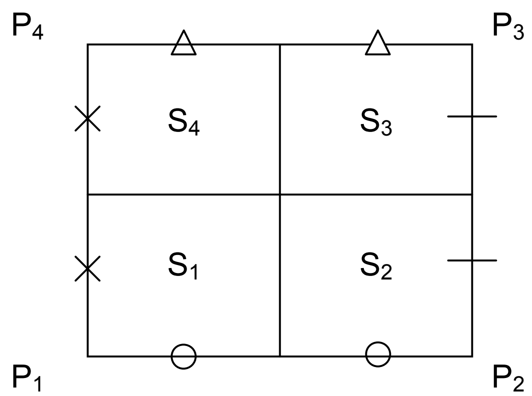
the load L_5 concentrated on the node P_5 will be

$L_5 = (f_2 + f_3) \times (1/2) \times S \times (4/9)$

the load L_6 concentrated on the node P_6 will be

$L_6 = (f_3 + f_1) \times (1/2) \times S \times (4/9)$

7.3. Linear Hexahedral Element



If the load distributions f_1~f_4 are given for the face groups areas S_1~S_4 respectively, the load L_1 concentrated on the node P_1 will be

$L_1 = S_1 \times f_1$

the load L_2 concentrated on the node P_2 will be

$L_2 = S_2 \times f_2$

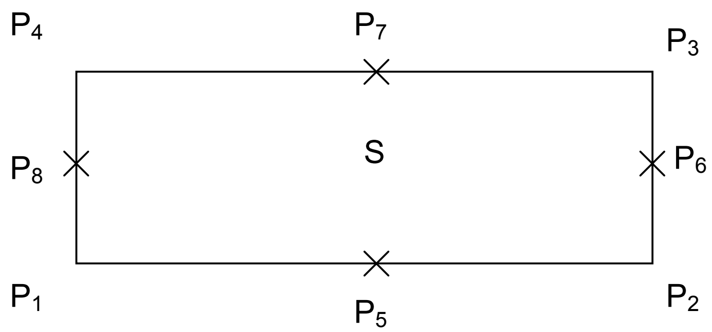
the load L_3 concentrated on the node P_3 will be

$L_3 = S_3 \times f_3$

the load L_4 concentrated on the node P_4 will be

$L_4 = S_4 \times f_4$

7.4. Quadratic Hexahedral Element



If the load distributions f_1~f_4 are given for the quadrangle face group area S, the load L_1 concentrated on the node P_1 will be

L_1 = - f_1 x S x (1/12)

the load L_2 concentrated on the node P_2 will be

L_2 = - f_2 x S x (1/12)

the load L_3 concentrated on the node P_3 will be

L_3 = - f_2 x S x (1/12)

the load L_4 concentrated on the node P_4 will be

L_4 = - f_2 x S x (1/12)

the load L_5 concentrated on the node P_5 will be

L_5 = (f_1 + f_2) x (1/2) x S x (1/3)

the load L_6 concentrated on the node P_6 will be

L_6 = (f_2 + f_3) x (1/2) x S x (1/3)

the load L_7 concentrated on the node P_7 will be

L_7 = (f_3 + f_4) x (1/2) x S x (1/3)

the load L_8 concentrated on the node P_8 will be

L_8 = (f_4 + f_1) x (1/2) x S x (1/3)