

# FE-SPORTFL

(Finite Element Simulator for PORoelasticity and Two-phase FLOW )

Version 1.2

User Manual

Created by: Masaaki Aichi

November 2015

Department of Environmental Systems Science, Graduate School of Frontier  
Sciences, The University of Tokyo

5-1-5 Kashiwanoha, Kashiwa City, Chiba Prefecture, 277-8563

Tel/Fax: 04-7136-4704

Email: [aichi@ku-tokyo.ac.jp](mailto:aichi@ku-tokyo.ac.jp)

## Introduction

FE-SPORTFL (Finite Element Simulator for PORoelasticity and Two-phase FLOW) is a finite element code for analyzing two-phase flow in geological formations and the associated poroelastic deformation of the formations. This code was developed through the joint research between Kanto Natural Gas Development Co., Ltd. and the Geosphere Environmental Systems Laboratory, Department of Environmental Systems, Graduate School of Frontier Sciences, The University of Tokyo, and through the Kujukuri Project II, with the primary aim of analyzing ground deformation problems in water-dissolved natural gas fields. Therefore, this code also has the functionality to analyze vertical two-phase flow in wells drilled into the formations.

This manual was created as a user manual for FESPORTFL.

Furthermore, the creators, Kanto Natural Gas Development Co., Ltd., the Kujukuri Project II participating companies, and the University of Tokyo shall not be held responsible for any damages incurred as a result of the use of this code and manual. Furthermore, the specifications of this code and manual may be changed without notice.



## Table of contents

Introduction.....	i
1. Configuring the input file and running the analysis .....	1
2. Analysis control parameter file .....	3
2.1. Analysis control parameter file for ground model .....	3
2.2. Well model analysis control parameter file .....	5
3. Project files.....	9
4. Node File .....	11
5. Element File.....	15
6. Material File .....	19
7. Load Step File .....	25
8. Universal Parameter File .....	29
9. Fluid properties file .....	31
10. Vertical two-phase flow well file .....	33
10.1. Well List File .....	33
10.2. Well Mesh File .....	33
11. Configuring the output file.....	35
11.1. Debug File .....	35
11.2. odb file .....	35
11.3. restart file.....	36
11.4. Result file .....	36
11.5. status file.....	37
12. Post- processing.....	39
12.1. make_plt.exe .....	39
12.2. diff_plt_node.exe .....	39
12.3. diff_plt_elem.exe .....	39
12.4. time_series_node.exe .....	40
12.5. time_series_elem.exe .....	40
12.6. well_data_post.exe.....	40
13. Error handling.....	43

## 1. Configuring the input file and running the analysis

The set of input files consists of a fixed file that controls the analysis, and a series of files (project files) that define physical parameters, model structures, and analysis steps. Table 1 summarizes the input files.

Typically, you create a set of project files to suit the problem you want to analyze, and then perform the analysis. The files that control the analysis are used to correct any convergence problems that may occur.

**Table 1: Input file overview**

file	overview
Analysis control file " solution_control.txt " " solution_control_well.txt "	Specifying analytical control parameters for the soil model and well model ( 2
Project files	A file that lists the files that define the model, properties, and analysis steps described below. The file extension must be 3)
Node File	Set the node number, coordinates, boundary condition type, initial value, and field variables (temperature and water concentration). ( Chapter 4)
Element File	Set the element type, element number, element composition node, layer dip, and strike. ( Chapter 5)
Element initial value file	Initial porosity and initial effective stress are given to element integration points. ( 5 In the analysis control file, you can specify whether to use element initial values. Even if you do not use them, you need an empty file.
Geotechnical Material File	Specify the physical properties of the geomaterial ( Chapter 6) .
Load Step File	Time steps, nodal loads (forces, mass flow rates), and time-varying boundary conditions are given. If there are wells, production and injection control is also performed. ( Chapter 7)
Universal Parameter File	Define gravitational acceleration, gas constant, standard state, and molecular weight (formula weight) of fluids and solutes. ( Chapter 8)

Fluid properties file	Density, compressibility factor, viscosity, solubility, surface tension, and saturated vapor pressure are defined in function tables of temperature, pressure, and salt concentration. ( 9
Well List File (optional)	Gives a list of well definition files ( Chapter 10) .
Well definition file (optional)	Define the mesh of the well, the pipe diameter, and the connection with the formation. Also define the mesh for gas lift, injection, and pumping. ( 10

Put the above set of input files in one working folder and start FESPORTFL.exe from the command prompt. You will be asked for the project file name, so specify it without the extension. For example, if you have prepared a project file called " test.txt " , enter test and press Enter to load the file list and run the analysis.

## 2. Analysis control parameter file

The analysis control parameter file is used to control the entire analysis, including convergence judgment, and is an essential file for executing the analysis.

There are two analysis control parameter files: one for controlling the analysis of the two-phase fluid flow and poroelastic deformation in the formation, and one for controlling the analysis of the vertical two-phase fluid flow in the well. In models without wells, the latter can be omitted.

### 2.1. Analysis control parameter file for ground model

It must be created and named “ solution\_control.txt ” .

Line 01: Comment line

Line 2: Dimension and initial element integration point variable specification flag

The flag specifying the initial element integration point variables specifies whether to read from the element initial value file.

(0: no specification, 1: porosity only, 2: effective stress only, 3: porosity and effective stress)

Line 3: Comment line

Line 4: Initial condition type (HW: hydraulic head, PC: capillary pressure, PW: pore water pressure, PG: pore gas pressure, SW: water saturation)

Line 05: Comment line

Line 6: Small change value when numerically differentiating (degree of freedom 1, degree of freedom 2, degree of freedom 3, ...)

Line 7: Comment line

Line 08: Residual convergence criterion value 1

(Tolerance of residual  $\div$  equivalent nodal force average value or residual  $\div$  equivalent nodal mass flow average value)

Order of entry: (freedom 1, freedom 2, freedom 3, ...)

Line 09: Comment line

Line 10: Residual convergence criterion value 2 (absolute value of the residual to be tolerated)

Order of entry: (freedom 1, freedom 2, freedom 3, ...)

Line 11: Comment line

Line 12: Corrector convergence criterion value 1

(Modifier/deflection tolerance for the node with the largest modifier)  
 Order of entry: (freedom 1, freedom 2, freedom 3, ...)

Line 13: Comment line

Line 14: Corrector convergence criterion value 2  
 (Absolute value of allowable modifier)  
 Order of entry: (freedom 1, freedom 2, freedom 3, ...)

Line 15: Comment line

Line 16: Time step control (unused)

Line 17: Comment line

Line 18: Maximum number of iterations of Newton's method

Line 19: Comment line

Line 20: Select the matrix solver (currently, you can only choose pardiso ).

Line 21: Comment line

Line 22: Number of parallel calculation threads

Line 23: Comment line

Line 24: Upwind control (1: Full upwind, 0: Central difference), FEM type for water retention term (1: simpleFEM , 0: Standard Galerkin ...1 is preferred), FEM type for gas retention term (1: simpleFEM , 0: Standard Galerkin ...1 is preferred)  
 Order of entries: (Upwind method for relative permeability, Upwind method for density, simpleFEM , FEM type)

Line 25: Comment line

Line 26: Maximum number of iterations to converge the seepage boundary

Line 27: Comment line

Line 28: Maximum number of iterations for adjusting boundary conditions between well and ground models

Line 29: Comment line

Line 30: Acceleration coefficient for adjusting boundary conditions between well and ground models (0 to 1)

Line 31: Comment line

Line 32: Upper limit of the number of times to decrease the time step and retry when convergence is not achieved

(Example of input)

Dimension/(0:none,1:porosity,2:effective\_stress(6),3:porosity and effective\_stress (6))  
 2,0



Initial condition type for fluids

PWP G

Virtual change for numerical derivative

1.d-6,1.d-6,1.d-8,-1.d-8

Residual tolerance (ratio to average force/flow)

1.d-3,1.d-3,1.d-3,1.d-3

Zero remaining

1.d-10,1.d-10,1.d-15,1.d-20

Correction tolerance [ratio],[Max]

1.d-2,1.d-2,1.d-2,1.d-2,1.25d-1

Zero correction

1.d-6,1.d-6,1.d-6,1.d-6

Time control

0.5d0,1.2d0,19,20

Maximum Newton iteration

50

Solver (0:pardiso/1:gmres)

0

Number of Threads

2

Upwind parameter ( kr,density )/Zhu's integration weight ( w,g )

1.d0,1.d0, 1 .d0,0.d0

Maximum iteration for converging discharge boundary conditions

10

Maximum iteration for well productions

30

Acceleration factor for converging well/formation boundary conditions

1.0

Maximum attempt for the time step

10

## 2.2. Well model analysis control parameter file

It must be created and named “ solution\_control\_well.txt ” .

Line 01: Comment line

Line 02: Small change value when differentiating

Order of entries: (sum of gas and liquid velocities, difference between gas and liquid velocities, pressure, void fraction)

Line 3: Comment line

Line 04: Residual convergence criterion value 1

(Momentum residual  $\div$  Momentum average value or mass residual  $\div$  Mass flow average value tolerance)

Order of entries: (sum of gas and liquid velocities, difference between gas and liquid velocities, pressure, void fraction)

Line 05: Comment line

6 : Residual convergence criterion value 2 (absolute value of the residual to be tolerated)

Order of entries: (sum of gas and liquid velocities, difference between gas and liquid velocities, pressure, void fraction)

Line 7: Comment line

Line 8: Corrector convergence criterion value 1

(For the node with the largest modifier, modifier/tolerance of change)

Order of entries: (sum of gas and liquid velocities, difference between gas and liquid velocities, pressure, void fraction)

Line 09: Comment line

Line 10: Corrector convergence criterion value 2

(Absolute value of allowable modifier)

Order of entries: (sum of gas and liquid velocities, difference between gas and liquid velocities, pressure, void fraction)

Line 11: Comment line

Line 12: Time step control parameters

dt\_down , dt\_up , N1, N2

(If the number of iterations of Newton's method is N1 or less, multiply the time step by dt\_up . If the number of iterations of Newton's method is N2 or more, multiply the time step by dt\_down .)

Line 13: Comment line

Line 14: Upper limit of number of iterations for Newton's method

Line 15: Comment line

Line 16: If the time step is smaller than the initial setting value multiplied by the value specified here,

If a problem occurs, restart the analysis by going back to the beginning.

Line 17: Comment line

Line 18: Set the initial time step for restart analysis to "Initial time step before restart analysis"

Time step x value specified here

(Example of input)

Virtual change for numerical derivative

1.d-8,1.d-8,1.d-9,1.d-9

Residual tolerance (ratio to average momentum /flow)

1.d-3,1.d-3,1.d-3,1.d-3

Zero remaining

1.d-10,1.d-10,1.d-15,1.d-20

Correction tolerance

1.d-1,1.d-1,1.d-1,1.d-1

Zero correction

1.d-5,1.d-5,1.d-5,1.d-5

Time control

0.5d0,1.2d0,19,20

Maximum Newton iteration

99

Criterion for restart (ratio : time step/initial time step)

1.d-2

Time step factor for restart

1.d-1



### 3. Project files

A file that gives a list of files to be used in the analysis. The extension must be ".txt". The file name will be used as the analysis project name and for output file names, etc.

By specifying a keyword with a # symbol at the beginning, you are specifying the contents written in the file whose file name comes on the next line. For example, after the keyword #NODE, write the file name of the file that defines the node information on the next line. A list of keywords and their meanings is summarized in Table 2. All keywords and file names that are not optional must be specified, and the corresponding files must also be created. Since it is a keyword specification, they can be specified in any order as long as the necessary items are included without any excess or deficiency. If the same keyword appears more than once, the last specification takes precedence.

**Table 2: Project file keywords**

<b>keyword</b>	<b>Specified file</b>
#NODE	Node File
#ELEMENT	Element File Element initial value file
#MATERIAL	Geotechnical material property file
#LOAD	Analysis step file
#UNIVERSAL	Universal Parameter File
#WELL	Well list file (optional)
#FLUID	Fluid properties file

(Example of input)

#NODE

node.txt

#ELEMENT

elem.txt

elem\_ini.txt

#MATERIAL

material.txt

#LOAD

step.txt

#UNIVERSAL

universal\_parameter.txt

#WELL

well\_list.txt

#FLUID

fluid\_properties.txt

#### 4. Node File

A file that defines node information. The file name, including the extension, can be any name. 3

Here, we define the node number, node coordinates, node boundary condition type, initial value of the node variable, and field variable values (temperature, irrigation concentration). Write the information for one node together on one line, and write as many lines as necessary for the number of nodes. Since the program counts the total number of nodes by the number of lines, there should be no unnecessary blank lines, no duplication of information for the same node, and no inclusion of information for unnecessary nodes. The content of the node information to be written on one line is summarized in Table 3 each item from left to right in the order shown in Table 3, and separate items with tabs or commas.

**Table 3: Definition of node information**

item	Contents
Node Number	Node numbers are assigned with positive integers. They do not need to be consecutive numbers. No duplicates are allowed.
coordinate	In the two-dimensional case, it is represented by a set of two real numbers, and in the three-dimensional case, it is represented by a set of three real numbers.
Boundary Condition Types	<p>The second integer from the left in the case of two dimensions, and the third integer from the left in the case of three dimensions, specify the type of boundary condition related to displacement or mechanics, followed by integers specifying the types of boundary conditions for pore water pressure, pore gas pressure, temperature, and brine concentration.</p> <p>Mechanical boundary condition type: 0: Neuman condition or internal node, 1: Displacement 0, 2: Time-varying displacement (linked with load step file, 7</p> <p>- Boundary condition types for pore water pressure and pore gas pressure are: 0: Neuman condition or internal node, 1: No pressure change, 2: Pressure that changes with time (linked to the load step file, see Chapter 7), 4: Pore gas pressure is equal to pore water pressure, 6: Seepage surface boundary condition, Negative numbers: Enter the well number in the pore water</p>

	<p>pressure field (for example, enter -1 for well number 1) and any negative number in the pore gas pressure field.</p> <p>- The boundary condition type for temperature and water concentration 7</p>
Initial values of nodal variables	<p>In the case of two dimensions, the second from the left, and in the case of three dimensions, the third from the left, are the initial values of the displacement. Next, specify the values of the pore water pressure, pore gas pressure, temperature, and brine concentration in that order.</p>

(Example of input)

○Two-dimensional

101,0,-600,1,1,-1,-1,1,1,0,0,0,0.96,298.95,5.00E-04  
102,0.2,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
103,0.424223782,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
104,0.675605303,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
105,0.95743388,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
106,1.273397227,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
107,1.627629709,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
108,2.024766443,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
109,2.470003945,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
110,2.969168127,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
111,3.52879053,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
112,4.156193788,-600,0,1,0,0,1,1,0,0,0,0.96,298.95,5.00E-04  
.....

○Three Dimensions

10101,0,0,-600,1,1,1,-1,-1,1,1,0,0,0,0.96,277.35,0.0005  
10102,10,0,-600,0,1,1,0,0,1,1,0,0,0,0.96,277.35,0.0005  
10103,20,0,-600,0,1,1,0,0,1,1,0,0,0,0.96,277.35,0.0005  
10104,30,0,-600,0,1,1,0,0,1,1,0,0,0,0.96,277.35,0.0005  
10105,40,0,-600,0,1,1,0,0,1,1,0,0,0,0.96,277.35,0.0005  
10106,50,0,-600,0,1,1,0,0,1,1,0,0,0,0.96,277.35,0.0005  
10107,60,0,-600,0,1,1,0,0,1,1,0,0,0,0.96,277.35,0.0005  
10108,70,0,-600,0,1,1,0,0,1,1,0,0,0,0.96,277.35,0.0005



10109,80,0,-600,0,1,1,0,0,1,1,0,0,0,0,0.96,277.35,0.0005  
.....



## 5. Element File

An element file consists of two files: one that defines element information and one that specifies the initial values to be set at the integration points in the elements. The latter will not work unless it is enabled in " solution\_control.txt " ( see Section 2.1), but even if it is disabled, an empty file must be prepared. The file names of both files, including their extensions, can be freely set. 3

In the file that defines element information, the element number, element type, node numbers that compose the element, material number ( see Chapter 6), and dip and strike of the layer are defined. The dip and strike of the layer are only meaningful when anisotropic material ( see Chapter 6) is defined.

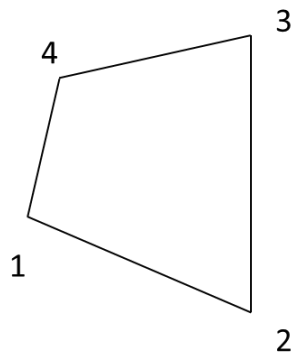
Write the information for one element together on one line, and write as many elements as necessary. Since the program counts the total number of elements by the number of lines, you should not include unnecessary blank lines, duplicate information for the same element, or include unnecessary elements. The content of the element information to be written on one line Table 4summarized in Table 4Table 3, from left to right, separated by tabs or commas.

**Table 4: Definition of element information**

item	Contents
Element Type	Define the element type. Currently available element types are CPEBL1: plane strain quadrilateral linear element, C3DBL1: hexahedral linear element, and ZAXBL1: axisymmetric quadrilateral linear element.
Element Number	Specify a positive integer. It is not necessary to use consecutive numbers. Duplicate numbers must not be used.
Node numbers that compose the element	Specify the numbers of the nodes that compose the element. The numbering order is the same as that often used in commercially available finite element codes (e.g. ABAQUS). For quadrilateral elements, specify the numbering in a counterclockwise direction ( Figure 1). For hexahedral elements, describe the elements counterclockwise from the bottom face first. However, number the elements so that the top face is in the direction of a right-hand screw ( Figure 1).
Material Number	Define the material properties of the element. 6

Formation dip angle	Specify the dip of the stratum (gradient from the horizontal plane) in degrees (°) from 0 to 90.
Strike angle of formation	Specify the strike of the strata (north-south is set to 0, and the angle is taken counterclockwise from there) in degrees (°) in the range of -180 to +180.

Linear quadrilateral element



Linear hexahedron element

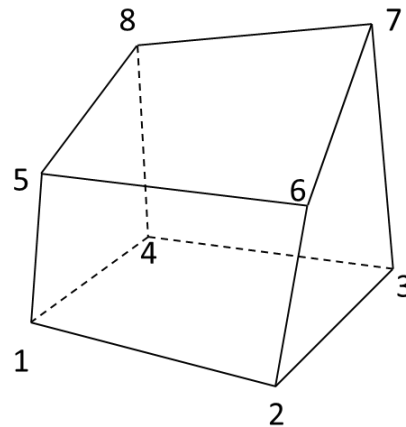


Figure 1: Node number order

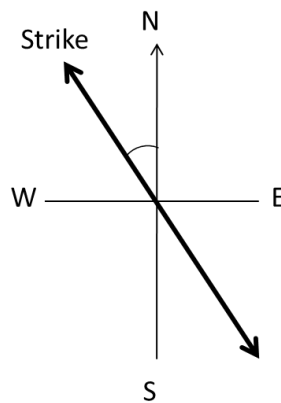
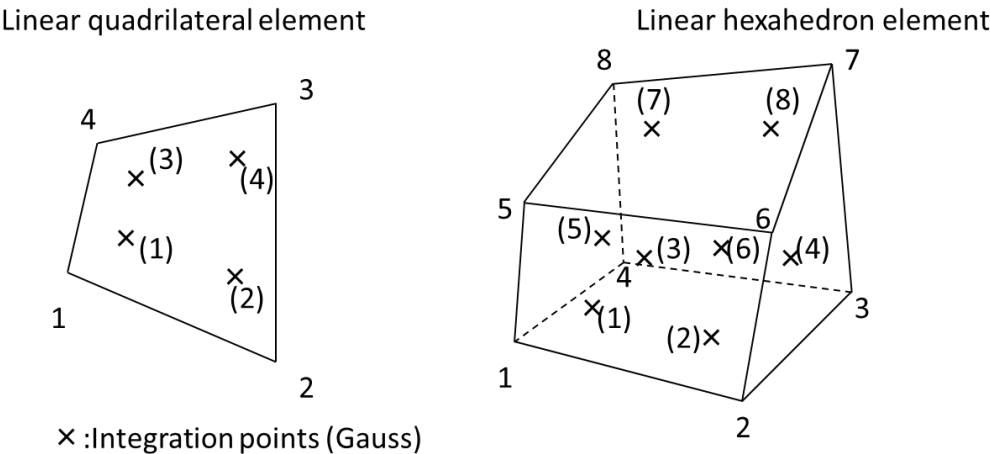


Figure 2: Strike of formation

In the file that defines the element initial values, the order is element number, element integration point number, and element integration point initial value, separated by tabs or commas. The order of element integration point numbers is the same as that often used in commercially available finite element codes (e.g. ABAQUS, etc.), as shown in Figure 3. The element integration point initial value can specify

porosity and effective stress. When the flag for specifying the initial element integration point variables on the second line of “ solution\_control.txt ” ( Section 2.1) is 1, only porosity is specified, when it is 2, only effective stress is specified, when it is 3, both porosity and effective stress are specified (porosity is specified first (left)). For effective stress, all six independent components ( $\sigma'_{11}$ ,  $\sigma'_{22}$ ,  $\sigma'_{33}$ ,  $\sigma'_{12}$ ,  $\sigma'_{13}$ ,  $\sigma'_{23}$ ) are given in this order. The effective stress is given with tension being positive.

However, within the scope of poroelasticity, the results are determined by the amount of change in effective stress, not the absolute value, so the initial value does not affect the analysis itself. The effective stress specification function is provided as an input function only, with the intention of expanding it to materials with physical properties including stress history characteristics in the future.



**Figure 3: Integration point numbering order**

(Example of element information file input)

```
CPEBL1      1      56      57      670      746      1      0      0
...
```

(Example of input in the element initial value file)

```
1      1      0.23  -1e-4  -1e-4  -1e-4  0      0      0
1      2      0.23  -1e-4  -1e-4  -1e-4  0      0      0
1      3      0.24  -1e-4  -1e-4  -1e-4  0      0      0
1      4      0.24  -1e-4  -1e-4  -1e-4  0      0      0
...
```



## 6. Material File

This file defines the geomaterial properties. The file name, including the extension, can be freely set. 3

In this file, various parameters for defining the physical properties of the geomaterial are set by specifying keywords beginning with #. 5shows the keywords and their settings. Note that the #Material keyword must be placed at the beginning of each material definition.

**Table 5: Definitions of material properties**

keyword	explanation	Example
#Material	Specify at the beginning of each material definition. Define the material number and material anisotropy. Specify the material number as a positive integer on the line following the #MATERIAL keyword. It does not have to be consecutive numbers, but they must not overlap. Define the anisotropy on the line following the material number. Currently, there are two types supported: Isotropy: isotropy, and Transverse: axisymmetric anisotropy.	#Material 1 Transverse
#Density	Specify the density of the solid phase	#Density 2.7e-3
#Elastic	The first line specifies Young's modulus, Poisson's ratio, and modulus of rigidity (in the case of anisotropy), and the second line specifies the Biot -Willis coefficient. In the case of isotropy, the first line defines Young's modulus and Poisson's ratio. In the case of axisymmetric anisotropy, the Young's modulus in the plane of symmetry ( $E_h$ ), the Young's modulus in the axial direction of symmetry ( $E_z$ ), the Poisson's ratio in the plane of symmetry ( $\nu_{hh}$ ), the Poisson's ratio ( $\nu_{hz}$ ), which represents the elongation in the axial	(Isotropic) #Elastic 500,0.25 0.99 (Axisymmetric anisotropy) #Elastic 5000,500,0.25,0.3,200 0.95,0.99

	direction of symmetry when stress is applied in the axial direction , and the modulus of rigidity outside the plane of symmetry ( $G_{zh}$ ). In the case of isotropy, the second line specifies one Biot -Willis coefficient. In the case of axisymmetric anisotropy, two coefficients are specified: the Biot -Willis coefficient in the plane of symmetry and the Biot -Willis coefficient in the axial direction .	
#Porosity	Specify the porosity. If you specify it at the element integration point initial value, it takes precedence.	#Porosity 0.37
#Permeability	<p>The first line gives the absolute permeability, and the second line gives the porosity dependence of the absolute permeability . In the case of isotropic materials, one absolute permeability value is given on the first line, and in the case of axially symmetric anisotropy, two absolute permeability values are given, in the order of the value within the plane of symmetry and the value in the axial direction of symmetry. The parameters expressing the porosity dependence on the second line are the values of a and b that appear in the following equation. The porosity dependence of absolute permeability is expressed as K, porosity n, initial absolute permeability <math>K_0</math> and initial porosity <math>n_0</math> .</p> $\frac{K}{K_0} = a \frac{n^3 (1 - n_0)^2}{n_0^3 (1 - n)^2} + (1 - a) \left( \frac{n}{n_0} \right)^b$ <p>When a is 1, the equation is the Kozeny -Carman equation, which is suitable for gravelly layers. When a is 0, the equation is the power law equation, which is suitable for muddy layers.</p>	#Permeability 5e-14,5e-15 0.,10



# K <sub>rw</sub>	<p>Define the table of relative permeability of the aqueous phase. In the first line, enter a symbol indicating what function the relative permeability is to be expressed as. SW: Saturation of the aqueous phase, SG: Saturation of the gas phase, SU: Suction. In the second line and onwards, set as many pairs of variables (saturation, etc.) and relative permeability values as necessary. Hysteresis cannot be defined at present. The variables (saturation, etc.) must be set in ascending order of value. There should be no blank lines or duplications. The values of the variables (saturation, etc.) do not need to be equally spaced, but equally spaced values often result in faster calculations. In the case of isotropy, one relative permeability is given. In the case of axisymmetric anisotropy, two columns are given: the relative permeability within the plane of symmetry and the relative permeability along the axis of symmetry.</p>	<p># K<sub>rw</sub></p> <p>SW</p> <p>0,0,0</p> <p>0.67,0.018228,0.014582</p> <p>4</p> <p>0.74,0.04924,0.039392</p> <p>0.8,0.10737,0.085896</p> <p>0.83,0.15516,0.124128</p> <p>0.86,0.2213,0.17704</p> <p>0.89,0.31182,0.249456</p> <p>0.92,0.43439,0.347512</p> <p>0.95,0.59874,0.478992</p> <p>0.98,0.81707,0.653656</p> <p>1,1,1</p>
# K <sub>rg</sub>	<p>Define the gas phase relative permeability table. In the first line, enter a symbol indicating what function the relative permeability is to be expressed as. SW: Water phase saturation, SG: Gas phase saturation, SU: Suction. From the second line onwards, set as many pairs of variables (saturation, etc.) and relative permeability values as necessary. Hysteresis cannot currently be defined. Variables (saturation, etc.) must be set in ascending order. There should be no blank lines or duplications. The values of variables (saturation, etc.) do not need to be equally spaced, but equally</p>	<p># K<sub>rg</sub></p> <p>SW</p> <p>0,1,1</p> <p>0.67,1,0.95</p> <p>0.74,1,0.9</p> <p>0.8,1,0.8</p> <p>0.83,0.61413,0.491304</p> <p>0.86,0.343,0.2744</p> <p>0.89,0.16638,0.133104</p> <p>0.92,0.064,0.0512</p> <p>0.95,0.015625,0.0125</p> <p>0.98,0.001,0.0008</p> <p>1,0,0</p>

	spaced values often result in faster calculations. In the case of isotropy, one relative permeability is given. In the case of axisymmetric anisotropy, two columns are given: the relative permeability within the plane of symmetry and the relative permeability along the axis of symmetry.	
#Suction	<p>Defines a table of suction as a function of the aqueous phase saturation. Hysteresis cannot currently be defined. Arrange as many pairs of aqueous phase saturation and suction values as necessary in ascending order of aqueous phase saturation. There should be no blank lines or duplicates. The aqueous phase saturation values do not need to be equally spaced, but calculations often proceed more quickly if they are equally spaced. Suction, as defined here, is calculated as depending on the changes in porosity, absolute permeability, and surface tension according to the following formula:</p> $\frac{s}{s_0} = \frac{\tau}{\tau_0} \sqrt{\frac{K_0}{K} \frac{n}{n_0}}$ <p>Here, s is suction, s<sub>0</sub> is initial suction, τ is surface tension, τ<sub>0</sub> is initial surface tension, K is absolute permeability, K<sub>0</sub> is initial absolute permeability, n is porosity, and n<sub>0</sub> is initial porosity.</p>	#Suction 0.13,0.95 0.14,0.7 0.15,0.5 0.16,0.35 0.17,0.25 0.18,0.2 0.21,0.15 0.25,0.1 0.4,0.05 1,0
#Bishop	Define Bishop's effective stress coefficient in a table. On the first line, enter a symbol indicating what function the relative permeability is to be expressed as. SW: Water phase saturation, SG: Gas phase saturation, SU: Suction. From the second line onwards, set as many pairs of variables	#Bishop SW 0.,0. 1,1

	(saturation, etc.) and effective stress coefficient values as necessary. Hysteresis cannot be defined at this time. Variables (saturation, etc.) must be set in ascending order. There should be no blank lines or duplications. Variables (saturation, etc.) do not need to be spaced at equal intervals, but calculations often proceed faster when they are spaced at equal intervals.	
--	---	--

(Example of input)

#Material

1

Transverse

#Density

2.7e-3

#Elastic

500,500,0.25,0.25,200

0.99,0.99

#Porosity

0.43

#Permeability

0,7e-18

0.,10.

# K<sub>rw</sub>

SW

0,0.0000E+00

0.67,8.1827E-03

0.74,2.6964E-02

0.8,6.8719E-02

0.83,1.0689E-01

0.86,1.6367E-01

0.89,2.4699E-01

0.92,3.6767E-01

0.95,5.4036E-01

0.98,7.8472E-01  
1,1.0000E+00  
# Krg  
SW  
0,1  
0.67,1  
0.74,0.40993586  
0.8,0.186588921  
0.83,0.114588921  
0.86,0.064  
0.89,0.031043732  
0.92,0.011941691  
0.95,0.002915452  
0.96,0  
#Suction  
0.78,0.95  
0.8,0.7  
0.83,0.5  
0.85,0.35  
0.87,0.25  
0.9,0.2  
0.95,0.15  
0.98,0.1  
0.99,0.05  
1,0  
#Bishop  
SW  
0.,0.  
1,1  
#End

## 7. Load Step File

This file specifies production conditions such as time step, equivalent nodal load, equivalent nodal flow rate, time-varying boundary conditions, well head pressure, gas lift, etc. The file name, including the extension, can be freely chosen, and is specified 3 In FESPORTFL, there are two types of steps: analysis steps (steps to which the same loading conditions are applied) and time steps, which are the time increments within the analysis steps.

The first line specifies the number of analysis steps as a positive integer. Load steps will only be read up to the number of steps specified here.

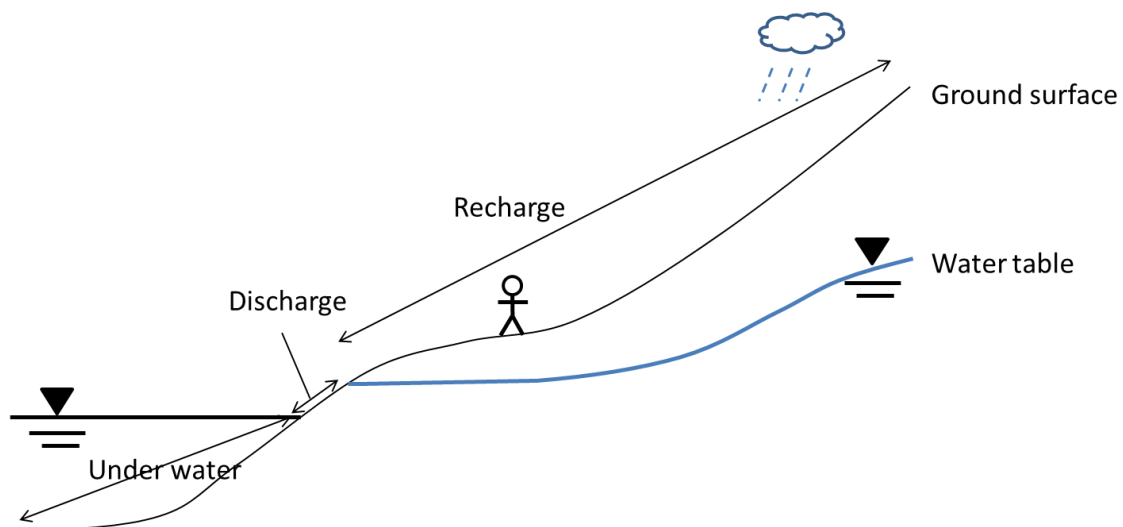
A load step defines the step type, time step, end time, and the load conditions to be applied to the nodes.

There are three types of load steps available: transient: unsteady analysis, geostatic: geostatic stability analysis, and steady: steady-state analysis. Normally, transient is used. Geostatic performs an analysis in which the pore water pressure and pore gas pressure are fixed and the displacement and stress are calculated so that the forces are balanced. Steady performs an analysis in which the displacement is set to 0 and the pore water pressure and pore gas pressure are in a steady state.

Following the load step type, the time step and end time are specified. The time step must be less than or equal to the end time.

The load condition is specified in the lines following the time step and end time. The load condition consists of three sets of node number, degree of freedom, and specified value. The meaning of the specified value changes depending on the node boundary condition type ( see Chapter 4). First, we will explain the case when the node boundary condition type ( see Chapter 4) is 0. When the degree of freedom is in the range related to displacement (less than the number of dimensions), it specifies the equivalent nodal force. The positive direction of each coordinate is positive. When the degree of freedom is pore water pressure (dimension + 1) and pore gas pressure (dimension + 2), it specifies the equivalent nodal mass flow rate of water and gas, respectively. A positive value of the equivalent nodal mass flow rate is the direction in which it is removed from the model (e.g. pumping), and a negative value is the direction in which it is added to the model (e.g. injection). Next, we will explain the case when the node boundary condition type ( see Chapter 4) is 2. In this case, the specified value indicates the actual value of each displacement or pressure because it is a time-varying Dirichlet type boundary condition. When the nodal boundary condition type ( see Chapter 4) is 2, if you forget to define it in the load step, it will become a Dirichlet boundary condition with a

value of zero, so special care is required. When the nodal boundary condition type ( see Chapter 4) is 6, it is a water-surface-compatible seepage boundary condition ( Figure 4), and both the pore water pressure degree of freedom (dimension + 1) and the pore gas pressure degree of freedom (dimension + 2) must be specified. For the pore gas pressure degree of freedom (dimension + 2), specify the gas phase pressure. For the pore water pressure degree of freedom (dimension + 1), specify the pore water pressure or the equivalent nodal flow rate of rainfall. When below the water surface, specify the pore water pressure, and when above the water surface, specify the rainfall. Which is specified is identified by whether the value specified for the pore water pressure degree of freedom (dimension + 1) is greater than the pore gas pressure degree of freedom (dimension + 2). Below the water surface, pore water pressure is generally higher than atmospheric pressure, and the value of rainfall is specified as a negative value on the recharge side, so it is always smaller than atmospheric pressure, making it identifiable. The range of the seepage surface for the water surface -corresponding seepage boundary condition is not known in advance, so it is found by iterative calculation during analysis.



**Figure 4: Water surface-corresponding seepage surface boundary condition (boundary condition type 6)**

To control the production conditions of a well, specify the well number and the well mesh number, and provide the specified value. When specifying the well number, write it as a negative integer. For example, if you want to control the production conditions of well number 1, specify -1. If you enter 0 as the well mesh number, you can specify the wellhead pressure (positive value) or wellhead return water volume (negative value). If you enter -1 as the well mesh number, you can specify the wellhead temperature. If any

other well mesh number is specified, gas for gas lift is sent to that mesh, and it is specified by the gas mass flow rate. Even if a well boundary condition is specified for a node, the node is treated as if no well exists until the first specification is made for the well number.

The end of the file should be marked as "transient end".

(Example of input)

- Example of control of ground model only

```
1
transient
1.377000E+03 1.377000E+03
6 4 1.013250E-01
454 4 1.013250E-01
455 4 1.013250E-01
456 4 1.013250E-01
457 4 1.013250E-01
458 4 1.013250E-01
459 4 1.013250E-01
460 4 1.013250E-01
461 4 1.013250E-01
463 1 -3.266667E-05
462 4 1.013250E-01
463 1 -4.900000E-05
464 1 -3.266667E-05
463 4 1.013250E-01
464 1 -4.900000E-05
465 1 -3.266667E-05
464 3 1.013250E-01
464 4 1.013250E-01
...
Transient End
```

- Well control example

```
2
transient
86400,86400
```

transient  
86400,86400  
-1,0,0.101325  
-1,-1,288.16  
-1,-348,-1d-8  
-1,-597,0.  
Transient end



## 8. Universal Parameter File

A file that specifies universal physical parameters that do not change during analysis, such as gravitational acceleration, gas constant, etc. The file name, including the extension, can be any name, and 3

Line 01: Comment line

Line 02: Gravitational acceleration

Line 3: Comment line

Line 4: Gas constant

Line 05: Comment line

Line 6: Standard atmospheric pressure

Line 7: Comment line

Line 8: Standard temperature

Line 09: Comment line

Line 10: Standard water concentration

Line 11: Comment line

Line 12: Molecular weight of the wetting fluid component

Line 13: Comment line

Line 14: Molecular weight of gas component

Line 15: Comment line

Line 16: Formula weight of salt

(Example of input)

1: Gravity acceleration (m/s<sup>2</sup>)

9.8d0

2: gas constant (J/K/ mol )

8.31

3: Standard pressure (MPa)

0.101325d0

4: Standard Temperature (K)

298.15d0

5: Standard Concentration ( Mmol /m<sup>3</sup>-solution)

0.d0

6: M water (kg/ mol )

18d-3

7:M gas (kg/ mol )

28.966d-3

8:M solute (kg/ mol )

58.5d-3

## 9. Fluid properties file

A file that specifies the physical properties of a fluid, such as density and viscosity, in table format, as they may depend on pressure, temperature, and brine concentration. The file name, including the extension, can be freely chosen, and 3

In this file, various parameters for defining the physical properties of the geomaterial are set by specifying keywords beginning with #. A list of keywords and the physical properties they specify are summarized in Table 6. For each keyword, the increments for salt concentration, temperature, and pressure are specified with #Concentration, #Temperature, and #Pressure, respectively, and the value is specified with #VALUE. It ends with a # to close. If there is only one increment for salt concentration, temperature, or pressure, it is assumed that there is no dependency on that variable, regardless of the value. Also, if the salt concentration, temperature, or pressure goes outside the range during the analysis, a constant value is extrapolated.

**Table 6: Definitions of fluid properties**

keyword	explanation	example												
# Mole_fraction_wn	<p>Specifies the solubility ( mol / mol ) of the non-wetting fluid component in the wetting fluid phase . In the example on the right, there is no dependence on salt concentration, and it is determined only by temperature and pressure.</p> <p>The definition on the right is the input format for data like the table below.</p> <table border="1"> <tr> <td>Pressure / Temperature</td><td>293</td><td>313</td></tr> <tr> <td>0</td><td>0.00E+00</td><td>0.00E+00</td></tr> <tr> <td>0.98</td><td>2.04E-04</td><td>1.43E-04</td></tr> <tr> <td>1.96</td><td>3.47E-04</td><td>2.86E-04</td></tr> </table> <p>Pressure is the innermost loop, followed by temperature, and finally</p>	Pressure / Temperature	293	313	0	0.00E+00	0.00E+00	0.98	2.04E-04	1.43E-04	1.96	3.47E-04	2.86E-04	<p># Mole_fraction_wn</p> <p>#Concentration</p> <p>0</p> <p>#Temperature</p> <p>293</p> <p>313</p> <p>#Pressure</p> <p>0</p> <p>0.98</p> <p>1.96</p> <p>#VALUE</p> <p>0.00E+00</p> <p>2.04E-04</p> <p>3.47E-04</p> <p>0.00E+00</p> <p>1.43E-04</p> <p>2.86E-04</p> <p>#</p>
Pressure / Temperature	293	313												
0	0.00E+00	0.00E+00												
0.98	2.04E-04	1.43E-04												
1.96	3.47E-04	2.86E-04												

	salt concentration (the same goes for all the following keywords). It is not necessary to create tables with equal intervals for salt concentration, temperature, and pressure, but calculations will often be faster if they are spaced evenly.	
# Viscosity_n	Specifies the viscosity of the non-wetting phase.	
# Viscosity_w	Specifies the viscosity of the wetting phase.	
#SVP	Specifies the vapor pressure of the wet phase.	
# Density_w	Specifies the density of the liquid phase.	
# Density_n	Specifies the density of the gas phase.	
# Surface_Tension	Specifies the surface tension of the wetting fluid phase.	

## 10. Vertical two-phase flow well file

### 10.1. Well List File

A list of files that define well numbers and meshes in the wells. The file names, including extensions, can be any name, and 3

" .txt " extension) that define the mesh, all on one line, and should be specified as many times as necessary. Well numbers do not need to be consecutive. However, there should be no duplications or blank lines. The file name of the file that defines the well mesh will be the well name as is. The well name should not exceed 64 alphanumeric characters.

(Example of input)

1.Well01

2 , Well0 2

### 10.2. Well Mesh File

In the well mesh file, specify the mesh number, elevation, radius, initial pressure, initial void fraction, initial liquid phase flow velocity, initial gas phase flow velocity, temperature, salinity, and lift type. Parts that are connected to the ground model share the same mesh number and ground model node number. For parts that are not connected, assign mesh numbers so that they do not overlap with the ground model. It is usually safe to assign negative integers so that there is absolutely no overlap. However, note that it is not possible to assign a gas lift to a mesh of -1. Lift types are 0: no lift, 1: reduced water injection position, and 2: gas lift injection position.

(Input example: -600 to -598 m is connected to node numbers 101 to 501 of the ground model, and gas lift injection is performed at an elevation of -250 m (mesh number -348))

```
101,-600,0.05,6.087165E+00,0.001,0,0,298.95,5.00E-10,0
201,-599.5,0.05,6.082177E+00,0.001,0,0,298.941,5.00E-10,0
301,-599,0.05,6.077189E+00,0.001,0,0,298.932,5.00E-10,0
401,-598.5,0.05,6.072200E+00,0.001,0,0,298.923,5.00E-10,0
501,-598,0.05,6.067212E+00,0.001,0,0,298.914,5.00E-10,0
-1,-597,0.05,6.057236E+00,0.001,0,0,298.896,5.00E-10,0
-2,-596,0.05,6.047259E+00,0.001,0,0,298.878,5.00E-10,0
-3,-595,0.05,6.037283E+00,0.001,0,0,298.86,5.00E-10,0
-4,-594,0.05,6.027307E+00,0.001,0,0,298.842,5.00E-10,0
-5,-593,0.05,6.017330E+00,0.001,0,0,298.824,5.00E-10,0
...
-345,-253,0.05,2.625354E+00,0.001,0,0,292.704,5.00E-10,0
-346,-252,0.05,2.615378E+00,0.001,0,0,292.686,5.00E-10,0
-347,-251,0.05,2.605401E+00,0.001,0,0,292.668,5.00E-10,0
-348,-250,0.05,2.595425E+00,0.001,0,0,292.65,5.00E-10,2
-349,-249,0.05,2.585449E+00,0.001,0,0,292.632,5.00E-10,0
-350,-248,0.05,2.575472E+00,0.001,0,0,292.614,5.00E-10,0
-351,-247,0.05,2.565496E+00,0.001,0,0,292.596,5.00E-10,0
-352,-246,0.05,2.555519E+00,0.001,0,0,292.578,5.00E-10,0
...
-593,-5,0.05,1.512070E-01,0.001,0,0,288.24,5.00E-10,0
-594,-4,0.05,1.412306E-01,0.001,0,0,288.222,5.00E-10,0
-595,-3,0.05,1.312542E-01,0.001,0,0,288.204,5.00E-10,0
-596,-2,0.05,1.212778E-01,0.001,0,0,288.186,5.00E-10,0
-597,-1,0.05,1.113014E-01,0.001,0,0,288.168,5.00E-10,0
```

## 11. Content of the output file

debug file, odb file, restart file, result file, status file, and wellresult file. All files except the status file are created separately for each analysis step. The output contents of each file are described below.

### 11.1. Debug File

Text format.

This shows an outline of the progress of the Newton method. It describes the process of searching for the optimal step size in the Newton method, the maximum residual value and its node for each equation, and the maximum corrector value and its node.

Also shown is the intermediate process of how the mass flow rate and pressure are corrected in the iterative calculation to converge the mass flow rate and pressure in the portion where the well and the ground are connected.

### 11.2. odb file

Binary format. Stores model information and calculation results.

project name\_model.odb file stores model information such as dimension (integer), total number of nodes (integer), total number of elements (integer), maximum number of nodes in an element (integer), maximum number of element integration points (integer), followed by a list of (node number (integer), node coordinates (real(8)  $\times$  number of dimensions), node boundary condition type (integer  $\times$  (degrees of freedom))  $\times$  total number of nodes, and a list of (element number (integer), number of nodes in an element (integer), number of integration points in an element (integer), element constituent nodes (integer), material number (integer), slope (real(8)), strike (real(8)))  $\times$  total number of elements.

The analysis results are saved in the project name\_step\_analysis step number.odb file . (analysis step (integer), time step (integer), time (real(8)), (main variable (real(8)  $\times$  degrees of freedom), residual (real(8)  $\times$  degrees of freedom))  $\times$  total number of nodes, (pore water pressure (real(8)) , water saturation ( real(8)) , pore gas pressure ( real(8)), porosity ( real(8)) , relative permeability of water phase (real(8)) , relative permeability of gas phase (real ( 8)), suction ( real (8)) , Bishop's effective stress coefficient (real(8)) , partial porosity of water phase (real(8)) , partial porosity change due to deformation of pores occupied by water (real(8)) , partial porosity change due to deformation of pores

occupied by gas (real(8)) , mass fraction of water in the water phase (real(8)) , mass fraction of gas components in the water phase (real(8)) , mass fraction of water vapor in the gas phase (real(8)) , mass fraction of gas components in the gas phase (real(8)) , displacement (real(8)  $\times$  dimension) , coordinates (real(8)  $\times$  dimension) , volume flux of the water phase (real(8)  $\times$  dimension) , volume flux of the gas phase (real(8)  $\times$  dimension) , mass flux of the water phase (real(8)  $\times$  dimension), mass flux of the gas phase (real(8)  $\times$  dimension)  $\times$  number of element integration points  $\times$  total number of elements)  $\times$  total number of time steps are recorded.

Project name\_well\_model.odb contains well mesh model data. The following data is recorded: total number of wells ( integer), maximum number of meshes per well (integer), (well number (integer), well name ( character(64) ), number of meshes (integer), (mesh number (integer) , elevation (real(8)) , radius (real(8)) , gas lift type (integer))  $\times$  number of meshes)  $\times$  total number of wells.

well project name\_step\_number of steps. In the odb , the calculation results of each analysis step are saved. (Analysis step (integer), time step (integer), time (real(8)), (wellhead pressure (real(8)) , gas production volume (standard state volume) (real(8)) , water production volume (standard state volume) (real(8)) , gas water ratio (standard state volume ratio) (real(8)), (liquid phase flow velocity (real(8)) , gas phase flow velocity (real(8)) , void fraction (real(8)) , pressure (real(8)) , temperature (real(8)) , salt concentration (real(8)), water inflow rate (real(8)) , gas inflow rate (real(8)))  $\times$  number of meshes)  $\times$  total number of wells)  $\times$  total number of time steps data is recorded.

Post-processing programs for visualizing these 12

### 11.3. restart file

Text format. Contains information for restart analysis.

The information for the node file and element initial value file is written, so you can use it by copying and pasting.

### 11.4. Result file

Text format. Calculation results at nodes are output.

The coordinates, main variables, residuals, mean values of nodal forces and mass flow rates are recorded.



### 11.5. status file

Text format. Contains an outline of the analysis information.

The analysis step, time step, time, and number of Newton methods are recorded.



## 12. Post- processing

- processing programs available are `make_plt` ( creates a file for tecplot ), `time_series_node` (extracts time series changes in node values), `time_series_elem` ( extracts time series changes in element values), `diff_plt_node` ( takes the difference between files for tecplot to display node values ), `diff_plt_elem` ( takes the difference between files for tecplot to display element medians ), and `well_data_post` (extracts well production data).

### 12.1. `make_plt.exe`

Run it from the command line and follow the instructions to enter 1. the project name, 2. a name to identify the output tecplot file, 3. the analysis step, and 4. the time step within the analysis step, in that order. The calculation results will be output in tecplot format.

The output files are an element integration point value file (`○○_ elem.plt` ), an element median value file (`○○_ elem_centroid.plt` ), and a nodal value file (`○○_ node.plt` ). Usually, the nodal value file and the element median value file are used for visualization. The element integration point values are displayed using tin interpolation because tecplot does not support finite element output. This provides more detailed data than the element median values, but often takes longer to display.

### 12.2. `diff_plt_node.exe`

Run it from the command line and follow the instructions to input 1. the name of the reference file, 2. the name of the file to be compared for difference, in that order. It will output the value of (the calculation result of the file specified in 2) - (the calculation result of the file specified in 1) in tecplot format.

The output file will have the name of the file specified in 2 with " diff\_ " prefixed.

### 12.3. `diff_plt_elem.exe`

Run it from the command line and follow the instructions to input 1. the name of the reference file, 2. the name of the file to be compared for difference, in that order. It will output the value of (the calculation result of the file specified in 2) - (the calculation result of the file specified in 1) in tecplot format.

The output file will have the name of the file specified in 2 with " diff\_ " prefixed.

#### 12.4. time\_series\_node.exe

Run it on the command line and follow the instructions to specify 1. the project name, 2. the name of the output file, and 3. the node number of the node to be output, in that order. The time series changes in the calculation results at the specified node will be output in text format. The output results are a table of "time, coordinate, displacement, pore water pressure, pore gas pressure, and point water head." Coordinates and displacements have two components in the case of two-dimensional analysis and three components in the case of three-dimensional analysis.

#### 12.5. time\_series\_elem.exe

Execute it from the command line and follow the instructions to specify, in order, 1. the project name, 2. the name of the output file, 3. the element number of the element you want to output, and 4. the integration point number. The integration point numbers are as shown in Figure 3. The output is in text format, and the time series changes in the calculation results at the specified element integration points are output. The output results are "time, coordinate, displacement, pore water pressure, pore gas pressure, capillary pressure, water saturation, porosity, volumetric water content, volumetric gas content, porosity change of pores occupied by water, porosity change of pores occupied by gas, relative permeability of water, relative permeability of gas, Bishop's chi, mass fraction of water in water phase, mass fraction of gas in water phase, mass fraction of water in gas layer, mass fraction of gas in gas layer, volume flux vector of water phase, volume flux vector of gas phase, strain tensor ( $\epsilon_{11}, \epsilon_{22}, \epsilon_{33}, \epsilon_{12}, \epsilon_{13}, \epsilon_{23}$ ), total stress tensor ( $\sigma_{11}, \sigma_{22}, \sigma_{33}, \sigma_{12}, \sigma_{13}, \sigma_{23}$ ), effective stress tensor ( $\sigma'_{11}, \sigma'_{22}, \sigma'_{33}, \sigma'_{12}, \sigma'_{13}, \sigma'_{23}$ ).

#### 12.6. well\_data\_post.exe

Run it on the command line and follow the instructions to specify 1. the project name, 2. the name of the output file, in that order. The output is in text format, and for each well, a table is output showing "time, gas volumetric flow rate (surface state), water volumetric flow rate (surface state), gas-water ratio (surface state)." Note that the gas volumetric flow rate is the net production volume at that time, with the gas lift injection

volume subtracted.



## 13. Error handling

This section describes how to deal with errors that are likely to occur during simulation. Error messages are

table 7



table 7: Error message

message	phenomenon	Common causes and solutions
Program encountered I/O error during the access to (file name)	Error while accessing file	There is a formatting error in the file. The file itself is empty. → Check whether the file is created properly.
Program encountered end of file during the access to (file name)	The end of the file is reached while accessing the file	There is a formatting error in the file. The file itself is empty. → Check whether the file is created properly.
Check the unit consistency : C, Mc and $\rho_w$	An error occurred while calculating the mass fraction of the aqueous phase.	There is a high possibility that the units of the brine concentration, salinity formula, and water density are not consistent . Check the universal parameter file, fluid property file, and nodal information file to see if they are in a consistent unit system.
unsupported element type	An element type name is used that is not in the element library	The element type name is incorrect → Check the element file
element type is not supported.	An element type name is used that is not in the element library	The element type name is incorrect → Check the element file
Unknown initial condition type for wetting phase:	Wrong initial condition type for water phase	Check solution_control.txt
Unknown initial condition type for non-wetting phase:	Wrong initial condition type for gas phase	Check solution_control.txt
Unknown keyword was found.	The keyword beginning with # in the project file is incorrect.	Loading → Check project file
Well files were not detected.	Well file not found	No well file Well list file is incorrect

