



Dense Discrete Phase Model (DDPM)

Particle Behavior Analysis Using Eulerian
in The case of One Particle

PROCEDURE MANUAL

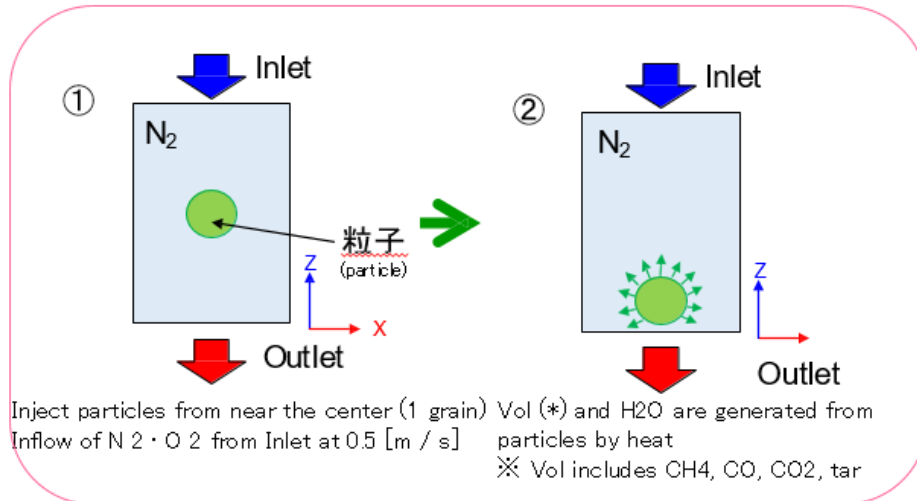
Rizqi Fitri, Mera KD | Kanazawa, July 2019

Table of Contents

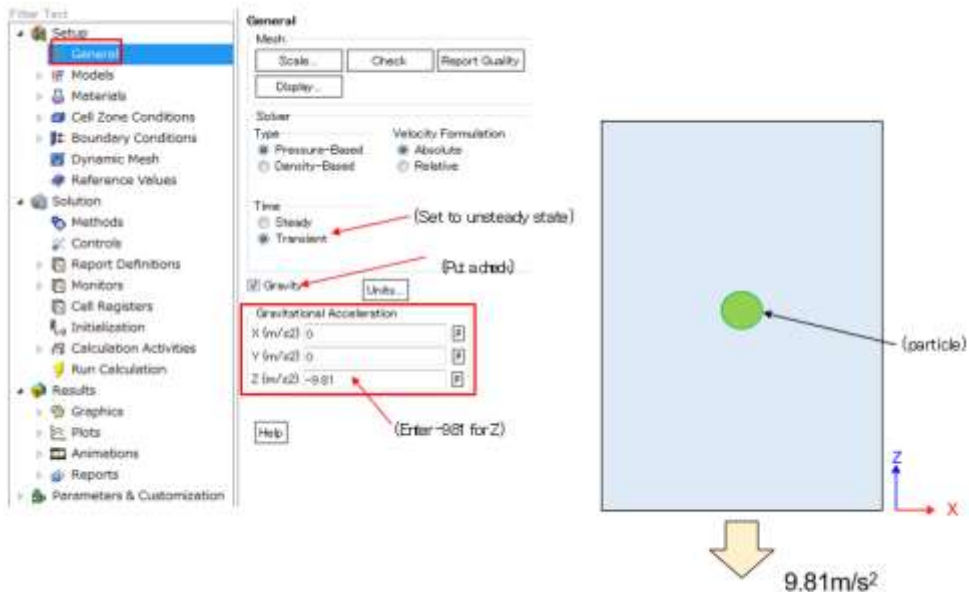
1.1	Analysis Model.....	2
2.1	General Settings.....	2
2.2	Species Settings 1	3
2.3	Models Settings	3
2.4	Materials Settings 1-a.....	4
	Materials Settings 1-b.....	4
	Materials Settings 1-c	5
	Materials Settings 1-d.....	5
	Materials Settings 1-e	6
2.5	Species Settings 2-a	7
	Species Settings 2-b.....	7
	Species Settings 2-c	8
2.6	Materials Settings 2-a	9
	Materials Settings 2-b	9
2.7	Discrete Phase Models (DPM) Settings.....	10
2.8	Injection Settings 1-a	10
	Injection Settings 1-b.....	11
2.9	Phase Settings.....	11
2.10	Operating Condition Settings.....	12
2.11	Boundary Condition Settings 1-a	12
	Boundary Condition Settings 1-b.....	13
	Boundary Condition Settings 1-c	13
	Boundary Condition Settings 1-d.....	14
3.1	Solution Methods	15
3.2	Solution Controls.....	15
3.3	Initialization	16
3.4	Calculation Activities	16
3.4 3.5	Run Calculation	17

1.1 Analysis Model

(SETTING DESCRIBED IN THE PROCEDURE MANUAL)



2.1 General Settings



2.2 Species Settings 1

Filter Text (Click to view details)

Setup

- General
- Models**
- Materials
- Cell Zone Conditions
- Boundary Conditions
- Dynamic Mesh
- Reference Values
- Solution
- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Initialization
- Calculation Activities
- Run Calculation
- Results
- Graphics
- Plots
- Animations
- Reports
- Parameters & Customization

Models

Models

- Multiphase - Off
- Energy - On
- Viscous - Realizable k- ϵ , Enhanced Wall Fn
- Radiation - Off
- Heat Exchanger - Off
- Species - Species Transport, Reactions**
- NOx - Off
- SOx - Off
- Soot - Off
- Decoupled Detailed Chemistry - Off
- Reactor Network - Off
- Discrete Phase - On
- Solidification & Melting - Off
- Acoustics - Off
- Electric Potential - Off

Species Model

Model

- Off
- Species Transport
- Non-Premixed Combustion
- Premixed Combustion
- Particle-Premixed Combustion
- Compositional RFG Transport

Phase Properties

Phase

Phase-1

Material

coal-mv-volatiles-air

Number of Volumetric Species 5

Number Of Solid Species 0

Number Of Site Species 0

Options

Full Multicomponent Diffusion

OK Cancel Help

(▼, select coal-mv-volatiles-air)

2.3 Models Settings

Filter Text

Setup

- General
- Models**
- Materials
- Cell Zone Conditions
- Boundary Conditions
- Dynamic Mesh
- Reference Values
- Solution
- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Initialization
- Calculation Activities
- Run Calculation
- Results
- Graphics
- Plots
- Animations
- Reports
- Parameters & Customization

Models

Models

- Multiphase - Eulerian
- Energy - On
- Viscous - Laminar
- Radiation - Off
- Heat Exchanger - Off
- Species - Species Transport, Reactions
- Discrete Phase - On
- Acoustics - Off
- Eulerian Wall Film - Off

Multiphase Model

Model

- Off
- Volume of Fluid
- Mixture
- Eulerian**
- Wet-Chem

Number of Eulerian Phases

1

Enter 1

Eulerian Parameters

Dense Discrete Phase Model

Number of Discrete Phases

1

Volume Fraction Parameters

Formulation

- Explicit
- Implicit**

OK Cancel Help

(Edit 3 items)

(Select Eulerian)

((DDPM) is selected)

(Select Implicit)

Discrete Phase Model

Interaction

- Interaction with Continuous Phase
- Update DPM Sources Every Flow Iteration
- DPM Iteration Interval 200
- Contour Plots for DPM Variables
- Mean Values

Particle Treatment

- Unsteady Particle Tracking
- Track with Fluid Flow Time Step
- Inject Particles at
- Particle Time Step
- Fluid Flow Time Step
- Particle Time Step Size (s) 0.0002
- Number of Flow Phases 1

Tracking Parameters

Max. Number of Steps

500

Specify Length Scale

Step Length Factor

5

OK Injections DEM Collisions Cancel Help

(Described on page 10)

Species Model

Model

- Off
- Species Transport
- Non-Premixed Combustion
- Premixed Combustion
- Particle-Premixed Combustion
- Compositional RFG Transport

Phase Properties

Phase

Phase-1

Phase Material coal-mv-volatiles-air

Reactions

- Volumetric
- Wall Surface
- Particle Surface
- Electrochemical

Turbulence-Chemistry Interaction

- Finite-Rate/No TCI
- Finite-Rate/Edt-Disipation
- Edt-Disipation
- Edt-Disipation Concept

Options

- Inlet Diffusion
- Diffusion Energy Source
- Full Multicomponent Diffusion
- Thermal Diffusion

Coal Calculator

OK Apply Cancel Help

(Described on the next page)

2.4 Materials Settings 1-a

(Materials settings1)

(Use Excel file values)
(Molecular Weight, Standard State Enthalpy)

※1 (Since N₂, O₂, H₂O (l) have not changed, omit it)

※2 (Tar is calculated from the physical property value of air by Molecular Weight).

Materials Settings 1-b

(Material Type
Click, Select fluid)

(Select CO, CH₄, H₂, H₂O (l))

After copy the substance, click COPY

When you copy the substance

Confirm that Substance Increasing

Materials Settings 1-c

(When using User-Defined Database (.scm file) ~ .scm file creation method ~)

If the selected substance is Confirm, that it is copied.

Select Substance From the Material Displayed

Click SAVE

(※ If the file name is "a" the extension will be none, When attaching .scm extension "A.scm" needs to be filename)

(Enter file name)

Materials Settings 1-d

(When using User-Defined Database (.scm file) ~ .scm file creation method ~)

※ When .scm file is saved with fluent, there is nothing attached to the extension

※ When using a file with an extension of .scm, editing with Notepad can cause errors, so edit with WordPad (If the extension is not available, Notepad or WordPad is OK)

(When opened in Notepad)

(When opened with WordPad)

※ 2 Data of Materials necessarily make fluid substance exist
If there is no fluid, an error occurs

Materials Settings 1-e

User-Defined Database(When using scm file) ~ How to insert scm file

(Select material to copy)

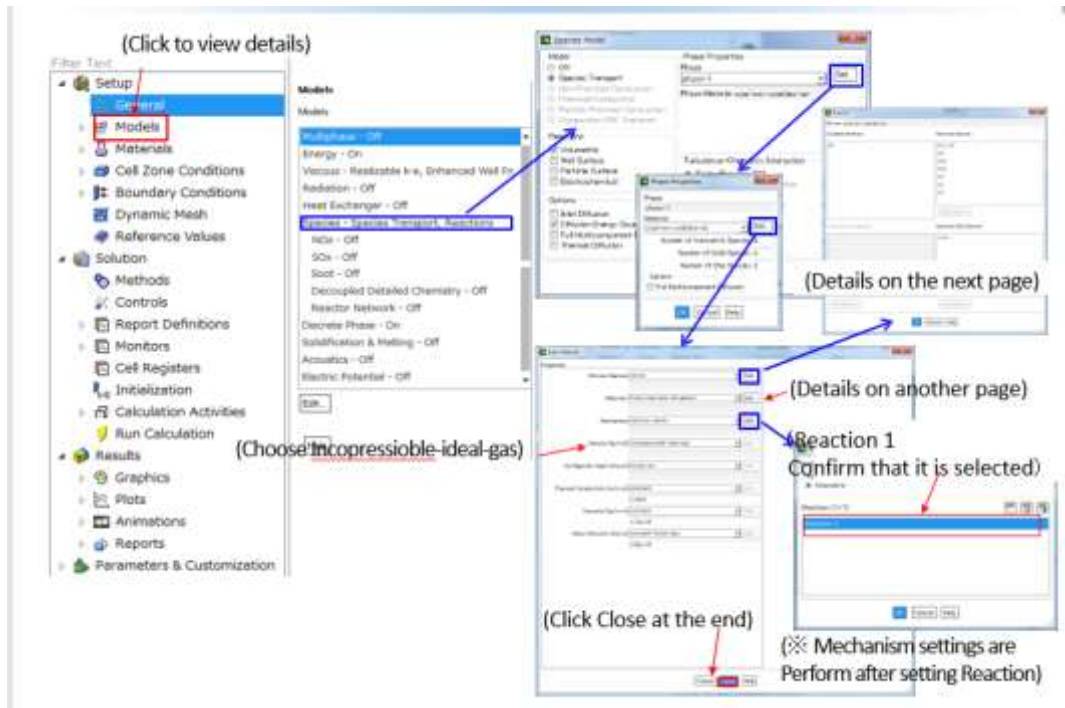
(Select the file to use)

(After Selecting, click COPY)

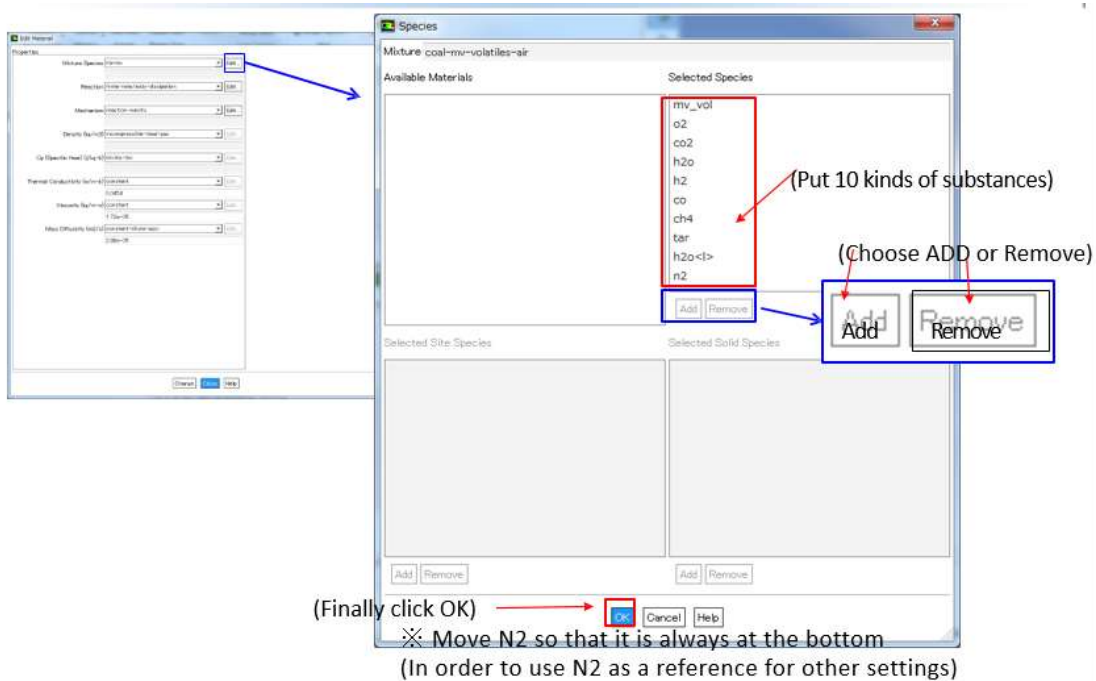
The screenshots show the following steps:

- Materials List:** A list of materials including 'nitrogen' is shown. A red box highlights 'nitrogen'.
- Create/Edit Material:** The 'Create/Edit Material' dialog is open. The 'User-Defined Database' option is selected under 'Order Materials by'. A red box highlights this option.
- Open Database:** The 'Open Database' dialog is open. The 'Database Name' is 'es/dp0/FLU-21/Fluent/test2.scm'. A red box highlights the 'Browse' button.
- Select File:** The 'Select File' dialog is open. A list of files is shown, including 'test2.scm'. A red box highlights 'test2.scm'. A red arrow points to the 'test2.scm' file.

2.5 Species Settings 2-a



Species Settings 2-b



Species Settings 2-c

(One chemical reaction formula
(Enter the items shown below))

Setting items Select 6

(Left side of the reaction equation)

Activation energy

Temperature index

Stoichiometric coefficient

Speed index

After entering all, click "OK"

Right side of the reaction equation

Enter chemical reaction formula with reference to Excel material

Species	Stoich. Coefficient	Rate Exponent
mv_val	1	0.2

Species	Stoich. Coefficient	Rate Exponent
ch4	-0.176	0
co2	-0.096	0
co	-0.077	0
h2	-0.385	0
h2o	-0.172	0
tar	-0.154	0

2.6 Materials Settings 2-a

Enter 1300
Enter 1000
Enter 28
Enter 0.0005
Enter 64

Change/Create Delete Close Help

After Setting, click Change/Create and then Click Close

Materials Settings 2-b

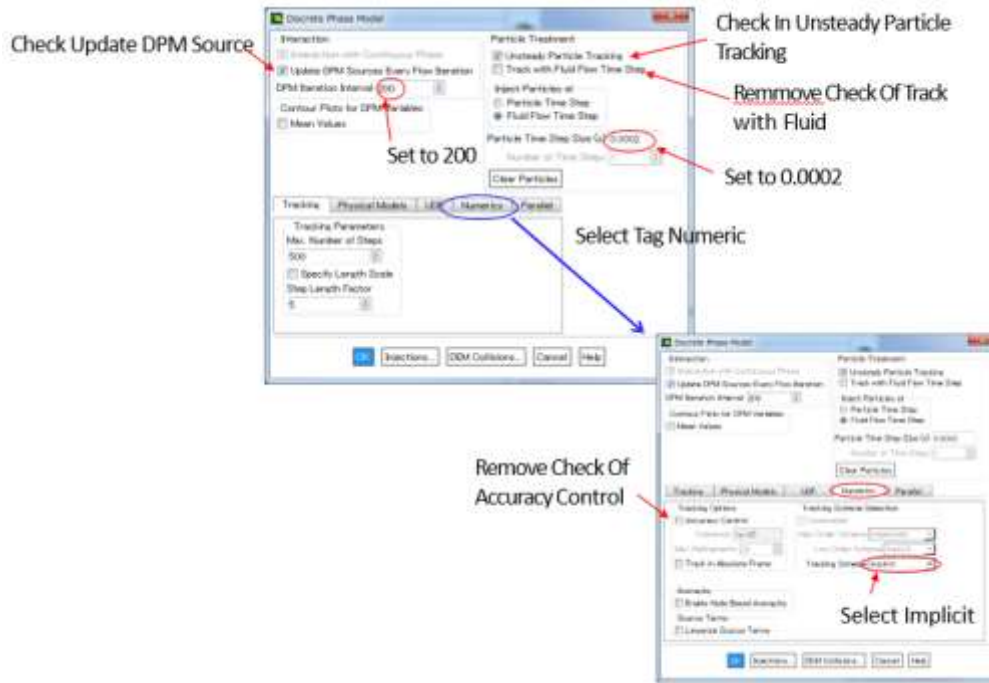
SETTINGS FOR VOLATILIZATION OF PARTICLES

In the Volatile Component Fraction -> constant
In the Combustible Fraction -> constant
Devolatilization Model -> single-rate

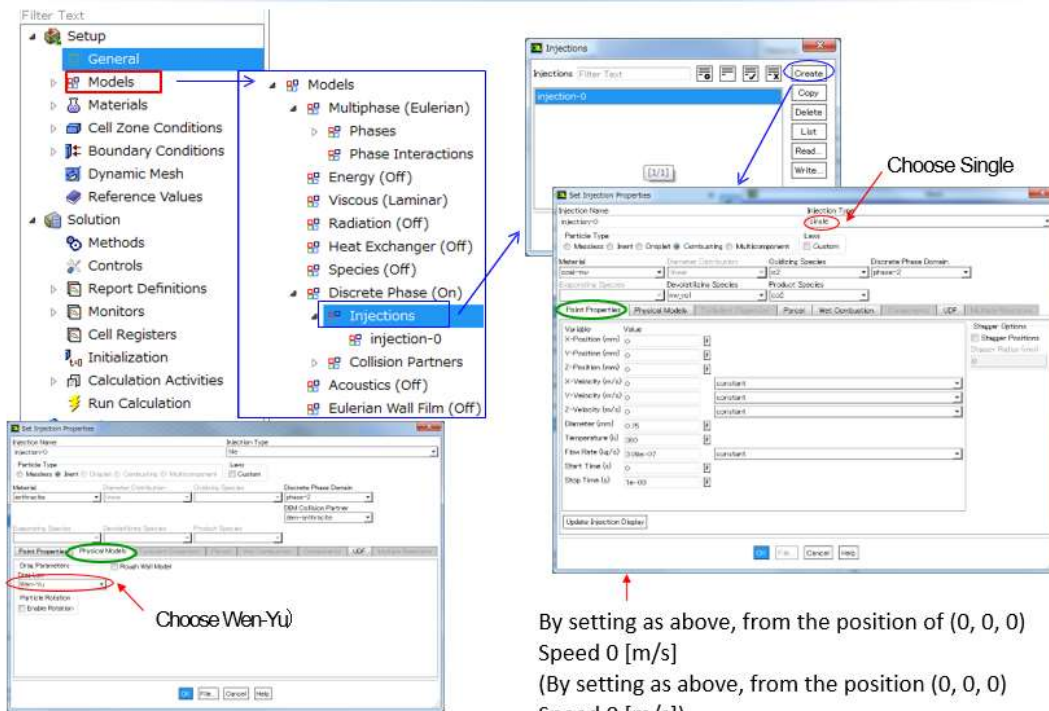
※ Ash is not set by a numerical value (it is judged by the following formula)

(Combustible within Vol + particle in particle + water content in particle) = Ash in particle

2.7 Discrete Phase Models (DPM) Settings



2.8 Injection Settings 1-a



Injection Settings 1-b

Setting related to particle evaporation

Confirm that it is H2O

Check in Wet Combustion Model

Set Liquid Material

Enter a number in Liquid Fraction

$$1 - (\text{Vol in partide} + \text{Combustible in particle} + \text{water content in particle}) = \text{Ash in particle}$$

2.9 Phase Settings

Setting of Phase

Remove Check Volume Fraction

2.10 Operating Condition Settings

Setting of Operating Conditions

The image shows the 'Operating Conditions' dialog box in ANSYS Fluent. The 'Gravity' section is checked, and 'Specified Operating Density' is set to 1.225. A diagram shows a rectangular domain with 'Inlet' at the top and 'Outlet' at the bottom, with a particle labeled '粒子' inside.

Gravity set in General is reflected

Select Tag Of Operating Conditions

Specified Operating Density Put a check in and enter 1.225

2.11 Boundary Condition Settings 1-a

Setting of Boundary Conditions

The image shows the 'Boundary Conditions' dialog box in ANSYS Fluent. The 'Interior' boundary type is selected. A diagram shows a rectangular domain with 'Inlet' at the top and 'Outlet' at the bottom, with a particle labeled '粒子' inside.

Inlet

Outlet

Interior is the Default setting

粒子

Boundary Condition Settings 1-b

Setting of Boundary Conditions

Setting of Inlet

Setting of Momentum
Setting to Default

In the Case of Phase : Mixture

Choose Trap

In the Case of Phase : Mixture

Enter 0.5

Enter 0.23

Enter 1500

Boundary Condition Settings 1-c

Setting of Boundary Conditions

Outletの設定

Momentumの設定は
デフォルトのままのため省略

Phase:mixtureの場合

trapを選択

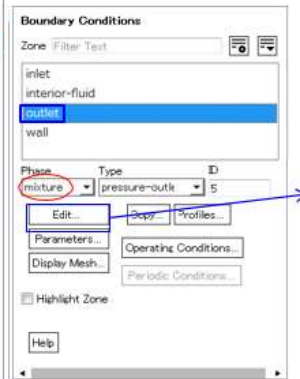
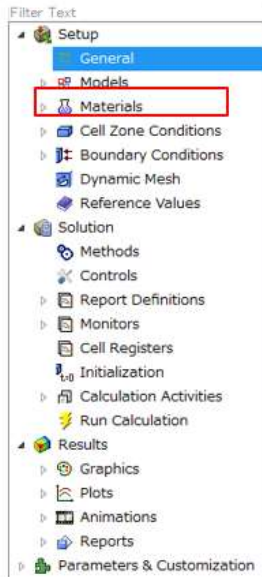
Phase:phase-1の場合

2,000を選択

0.23を選択

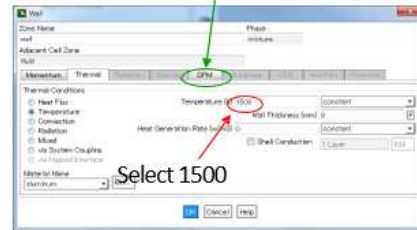
Boundary Condition Settings 1-d

Setting of Walls



Phase: mixture

The setting of DPM is Omitted because it remains the default



3.1 Solution Methods

Setting of Solution Methods

The screenshot shows the 'Solution Methods' panel in ANSYS Fluent. The 'Methods' sub-panel is selected in the left-hand tree. The main panel displays various numerical methods for different physical quantities. Red arrows point to specific settings:

- Spatial Discretization:**
 - Green-Cells Node Based (Set to Green Node Based)
 - QUICK (Set to QUICK)
- Energy:** Second Order Upwind (Set to Second Order Upwind)
- Phase-1 nu, vol:** Second Order Upwind (Set to Second Order Upwind)
- Phase-1 co2:** Second Order Upwind (Set to Second Order Upwind)
- Phase-1 h2o:** Second Order Upwind (Set to Second Order Upwind)
- Phase-1 h2:** Second Order Upwind (Set to Second Order Upwind)
- Phase-1 co:** Second Order Upwind (Set to Second Order Upwind)
- Phase-1 ch4:** Second Order Upwind (Set to Second Order Upwind)
- Phase-1 tar:** Second Order Upwind (Set to Second Order Upwind)

Other options include: Pressure-Velocity Coupling (Scheme: Phase Coupled SIMPLE), Solve N-Phase Volume Fraction Equations, Transient Formulation (First Order Implicit), Non-Iterative Time Advancement, Phase-Flux Formulation, Waxed-Face Gradient Correction, High Order Term Relaxation (Options...), and Set All Species Discretizations Together. A 'Default' button is also present.

3.2 Solution Controls

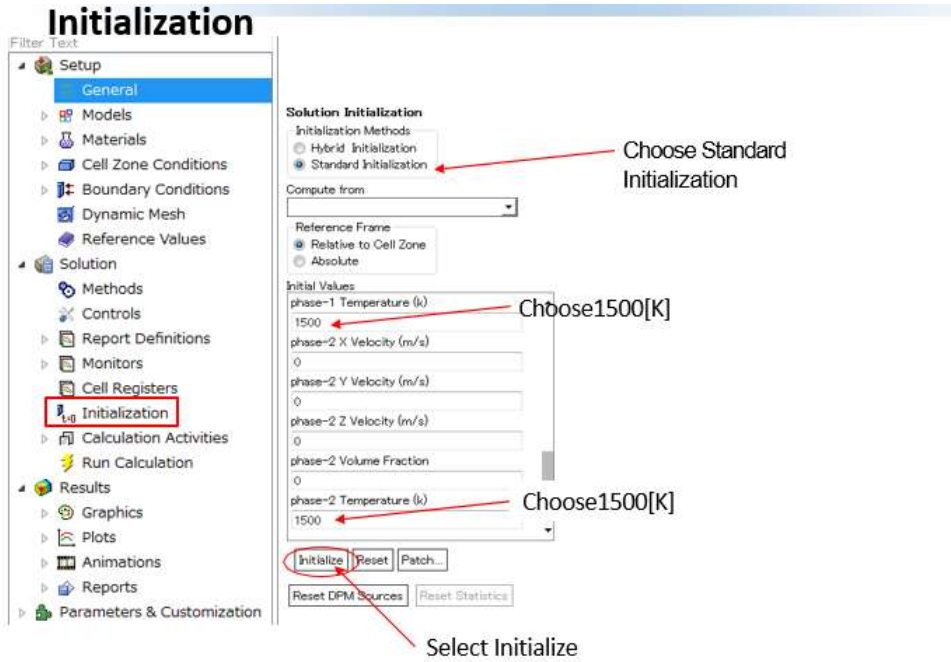
Setting of Solution Controls

The screenshot shows the 'Solution Controls' panel in ANSYS Fluent. The 'Controls' sub-panel is selected in the left-hand tree. The main panel displays under-relaxation factors for various physical quantities. Red arrows point to specific settings:

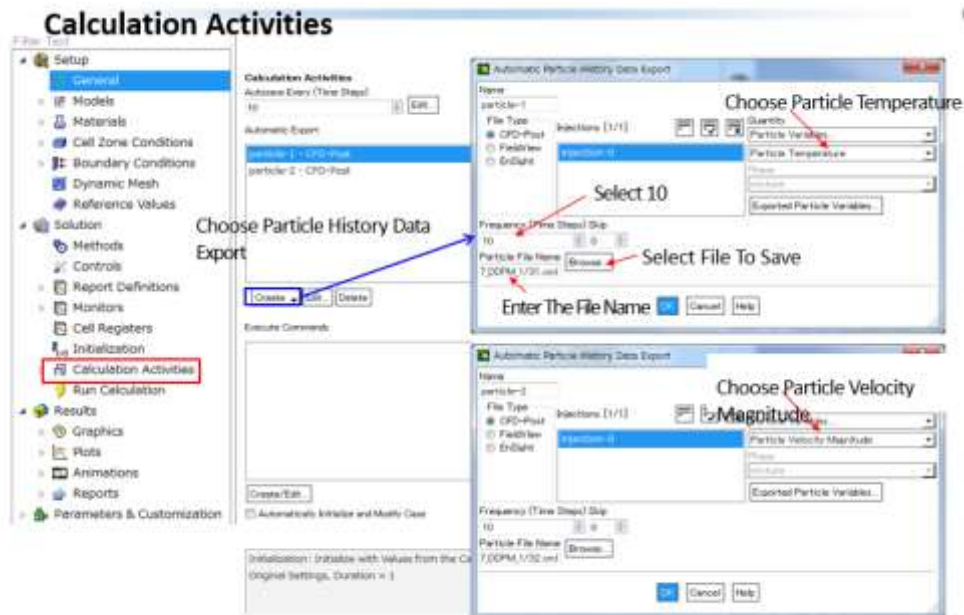
- Pressure: 0.9 (Set 0.9)
- Momentum: 0.2 (Set 0.2)
- Energy: 1 (Set 1)
- Volume Fraction: 1 (Set 1)

Other options include: Discrete Phase Sources (with fields for phase-1 nu, vol, phase-1 co2, phase-1 coo2, phase-1 h2o, phase-1 h2, phase-1 co, phase-1 ch4, phase-1 tar), a 'Default' button, and buttons for 'Equations...', 'Limits...', and 'Advanced...'. A checkbox 'Set All Species URFs Together' is also present.

3.3 Initialization



3.4 Calculation Activities



3.5 Run Calculation

The screenshot shows the 'Run Calculation' dialog box in a software application. The left sidebar contains a tree view with categories: Setup (General, Models, Materials, Cell Zone Conditions, Boundary Conditions, Dynamic Mesh, Reference Values), Solution (Methods, Controls, Report Definitions, Monitors, Cell Registers, Initialization, Calculation Activities), Results (Graphics, Plots, Animations, Reports), and Parameters & Customization. The 'Run Calculation' option under 'Calculation Activities' is highlighted with a red box. The main panel is titled 'Run Calculation' and includes several sections: 'Time Stepping Method' set to 'Fixed', 'Time Step Size (s)' set to '0.001', and 'Number of Time Steps' set to '1000'. There are also 'Options' for 'Data Sampling for Time Statistics' and 'Solid Time Step' (set to 'Automatic'). At the bottom, there is a 'Calculate' button highlighted with a red box and a 'Help' button. Red arrows point from text labels to the 'Time Step Size (s)' field (labeled 'Set 0.001'), the 'Number of Time Steps' field (labeled 'Enter The Number Calculation'), and the 'Calculate' button (labeled 'Start Calculation').

Run Calculation

Filter Text

Setup

- General
- Models
- Materials
- Cell Zone Conditions
- Boundary Conditions
- Dynamic Mesh
- Reference Values

Solution

- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Initialization
- Calculation Activities
 - Run Calculation**

Results

- Graphics
- Plots
- Animations
- Reports

Parameters & Customization

Run Calculation

Check Case... Preview Mesh Motion...

Time Stepping Method: Fixed

Time Step Size (s): 0.001

Number of Time Steps: 1000

Options

- Extrapolate Variables
- Data Sampling for Time Statistics
 - Sampling Interval: 1
 - Sampling Options...
 - Time Sampled (s): 0
- Solid Time Step
 - User Specified
 - Automatic**

Max Iterations/Time Step Reporting Interval: 20

Profile Update Interval: 1

Data File Quantities... Acoustic Signals... Acoustic Sources FFT...

Calculate Help

Set 0.001

Enter The Number Calculation

Start Calculation

---000---