Introduction

In order to perform a numerical simulation, mesh and simulation conditions are essential. All numerical simulation programs have programs called PreProcessors which create mesh and set the simulation conditions. The ideal PreProcessors can provide high-quality graphics to check the shape inputted, create high precision mesh in a short amount of time and easily set simulation conditions.
anyPRE can create mesh from shapes inputted through CAD files and carry out various CAD functions. Using anyPRE, you can set simulation conditions to run anySOLVER, which is AnyCasting's simulation engine.

About anyPRE

Objectives

anyPRE performs pre-processing by reading CAD data, forming FDM meshes and setting up simulation conditions to run anySOLVER. You can check the geometry of STL data by powerful graphic skill and make millions of meshed within seconds. In addition, you can select the work processes and materials as well as boundary, heat, gate conditions, etc..

Characteristics

anyPRE based on MS-Windows has been developed in a casting process-oriented way so that users working in the casting profession may operate the program with a little knowledge.

User Interface	Window users can easily operate the program as the menus and functions are Windows-based.
Fast Meshing Algorithm	Millions of grids can be created within a few seconds using the Surface Vector Tracking method. Efficient management of the number of mesh by variable mesh
High Quality Viewing	Inputted Geometries are analyzed by the view surface, view shape, measure and control transparency functions.
Easy Setting of Simulation Condition	The method of setting the simulation conditions is casting process-oriented, thus allowing even those who are not casting simulation specialists to run the program.

General Work Flow

anyPRE reads the geometry files as STL file or files created in anyPRE as GSC file. It builds mesh and sets the simulation conditions necessary to run anySOLVER.

1. Input of Geometry Files as STL files

2. Build Mesh

① Set entity properties(See Set Entity).
② Set mould(See Set Mould).
③ Set simulation domain(See Set Domain).
④ Build uniform mesh/variable mesh(See Build Uniform Mesh, Build Variable Mesh).

3. Set Simulation Condition

Basic Procedure: Fields that must be set in order to execute the simulation

① Decide upon the process and simulation method of the simulation(See Task Design).
② Select material(See Material Setting).
③ Set initial temperature and plane boundary conditions(See Initial and Plane BC).
④ Set heat transfer conditions such as heat transfer coefficient and coating effects of the interfaces of each entity(See Heat Transfer Model).
⑤ Set gate conditions(See Gate Condition).
⑥ Set gravity force(See Gravity Force).

Optional Modules: Fields that may be optionally set depending on the simulation objective

- Set fluid flow model(viscosity, turbulence, surface tension models)(See Fluid Flow Model).
- Select solidification shrinkage model(See Shrinkage Model).
- Select microstructure model(See Microstructure Model).
- Select outlet(See Outlet).
- Set cyclic casting conditions(See Cyclic Casting).
- Set rotational force(See Rotational Force).

Set Instruments: Fields that must be set if equipment exists

- Install sensor to view temperature, velocity, pressure, etc. at a certain point(See Sensor).
- Set channel characteristics(See Channel).
- Set pouring basin conditions(See Pouring Basin).
- Set stopper conditions(See Stopper).
- Set feeder conditions(See Feeder).

4. Launching anySOLVER

Launching Conditions

① Set conditions necessary to launch anySOLVER(See Soluition Method).
② Set simulation end and save conditions(See End/Output Condition).
③ Run anySOLVER after saving to a GSC file(See Run).

Input/Output Files

In order to run anyPRE, STL files are necessary to input geometries. A GSC file is created which makes task possible again in anyPRE. In addition, a msh and prp files are saved which are needed to run anySOLVER.

Input Files

STL file	The STL format is the same file format used in CAD programs. Geometries are outputted in STL file format. anyPRE reads these shape files and builds mesh.
GSC file	anyPRE creates the GSC file where you can save task information at any time and contains STL shape information as well. This file is may be read to carry out new projects or continue previous ones.

Output Files

Select File/Save and you can save into a GSC file format. While the GSC file is being saved, MSH and PRP files, which are necessary to run anySOLVER, are created. The name of the msh and prp files are created the same as that of the gsc file.

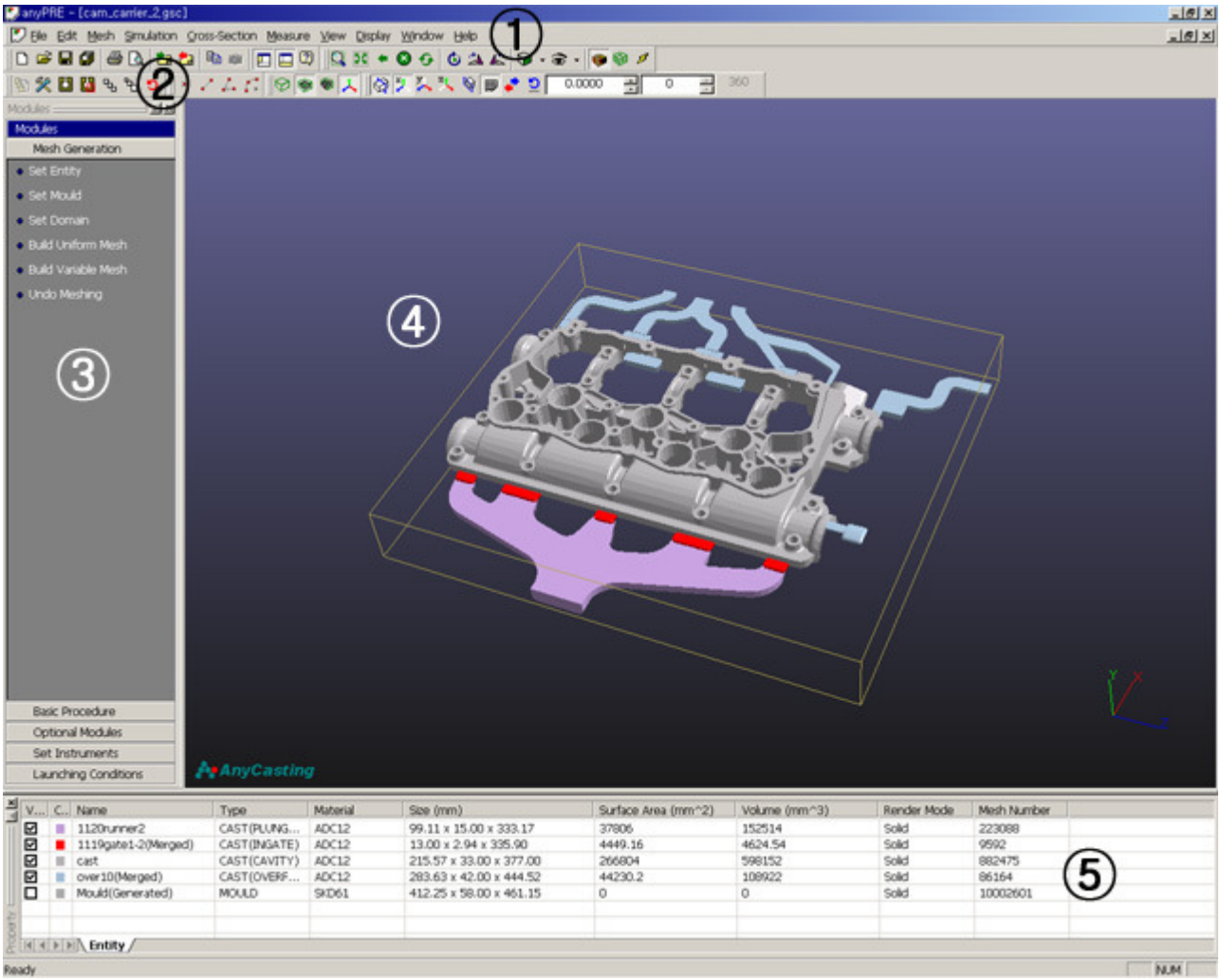
***Project*.gsc**	File that contains the content that was used in anyPRE
***Project*.msh**	File that contains mesh information
***Project*.prp**	File that contains simulation conditions

User Interface

The user interface of anyPRE can be easily used by any Windows user.
anyPRE is a user-friendly program that you can set up the conditions at a few clicks of the mouse and hit of the keypad.

Window Layout

The anyPRE window can be largely divided into 5 parts as shown below. Each window and tool bar can be arranged freely.



①	**Main Menubar**	The entire menu
②	**Various Toolbars**	Composed of the standard, view, display, mesh, measurement, and cross-section toolbar.
③	**Switch Window**	Window where main functions have been categorized into groups
④	**Main Window**	Graphic window
⑤	**Entity Window**	It is possible to view, at once, the name, color, size, volume, material, etc. of the entity.

How to Use the Mouse

Basic Methods

Rotate Click on left mouse button and drag
Move Click on middle mouse button and drag OR press Ctrl + click on left mouse button and drag
Zoom in/Zoom out Use mouse wheel OR press Shift + click on left mouse button and drag

View Fast Menu

When the mouse cursor is on top of a specific entity, click on the right mouse button to see the menu of that specific entity.
When the mouse cursor is on top of a background, click on the right mouse button to see the view menu.

Switch to Select Mode
When the function of setting gate condition, measuring and installing sensor is activated, press the space bar to turn on/off the select mode. Rotation is impossible during the select mode.

Cross section
When a plane of cross section is activated, move the mouse cursor to the four corners of the section plane and it will be changed into yellow.
Hold onto the section plane with the left mouse button and move it to the preferred location.

Mouse Cursor
Below are the cursor shapes according to mouse mode

 Basic mode, rotate

 Move

 Zoom in/Zoom out

 Rotate along screen axis

 Zoom in specific area

 Select mode

 Activate section plane

 Drag section plane

---

See Also: Utilizing the Window

Manipulation of View

Basic Functions

Rotate	Click on left mouse button and drag
Move	Click on middle mouse button and drag OR press Ctrl + left mouse button and drag
Zoom in/Zoom out	Use mouse wheel OR press Shift + left mouse button and drag
Zoom in/Zoom out Selected Area	Select **View ->Zoom Window** OR Ctrl + left mouse button and drag and drop. Especially, press hotkey, C to zoom in selected area.
Return to Previous View	Select **View->Previous View** OR press hotkey, V
Return to Initial View	Select **View->Reset View** OR press hotkey, R

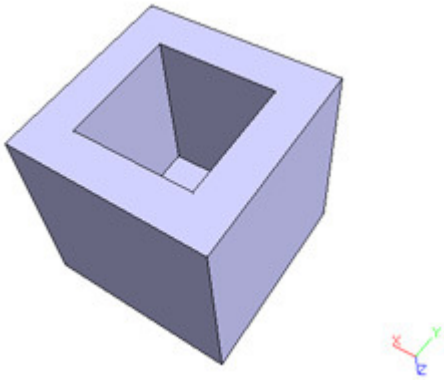
Note
Esc key enable you to return back to default mouse mode

Save Current User View Point
This can be used when creating a report or checking the design changes of two models.
In order to save the current view point,

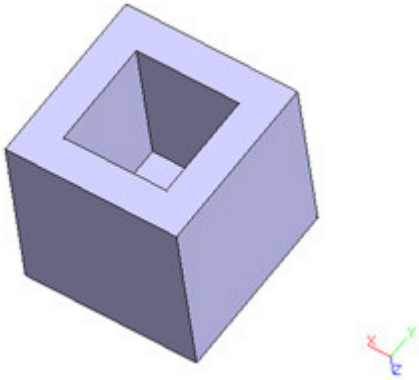
1. Select **View -> View points -> Add User View(Alt+U)** menu
2. Enter name of user in dialogue
3. A user defined view point is added as the last View Point field
4. If the added menu is selected, the saved view point is applied to the current screen

Rotate around Screen Axis
Select **View -> Axial Rotate/Rotate Left/Rotate Right** menu
It is possible to rotate at a 90 degree or selected angle based on the screen's normal vector direction.
It is possible to easily obtain the wanted screen rotating along the screen axis after switching to the basic viewpoint(±x, ±y, ±z).

Changing Projection Modes
The projection mode of the current screen may be changed by selecting the **View -> Perspective/Isometric** menu.
Perspective projection reflects perspective and isometric projection displays all in the same size regardless of the object's location.



Perspective Projection



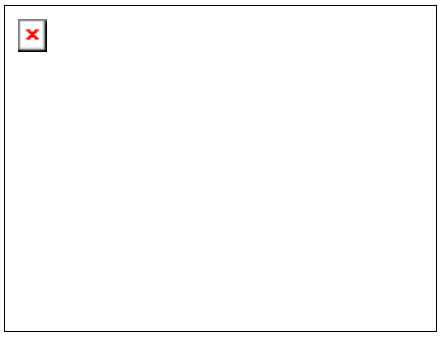
Orthographic Projection

View Cross-section

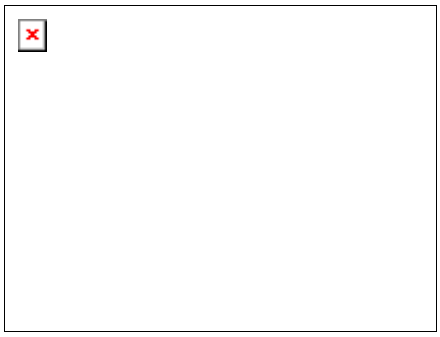
It is possible to move the cross-section by entering the coordinates/indices in the cross section, using the spin button or the mouse. Using the mouse, you,

1. Select the **Cross-Section->Active** menu and turn on the cross-section.
2. When a cross-section plane is activated, move the mouse cursor to the four corners of the plane and it will be changed into yellow.
3. Hold onto the plane with the left mouse button and move it to the preferred location.

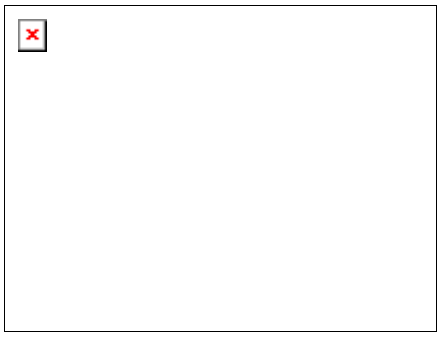
It is possible to make a maximum of 3-direction(x, y, z) cross-sections.



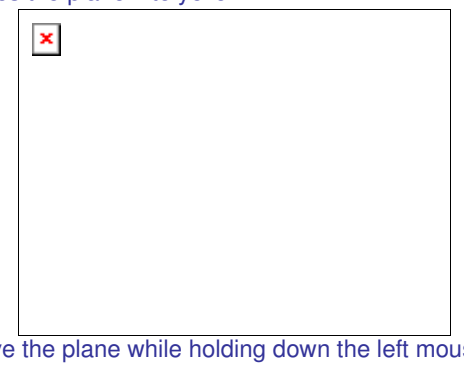
1. Cross-section Plane is ON



2. Move the mouse cursor on the plane edge and it changes the plane into yellow.



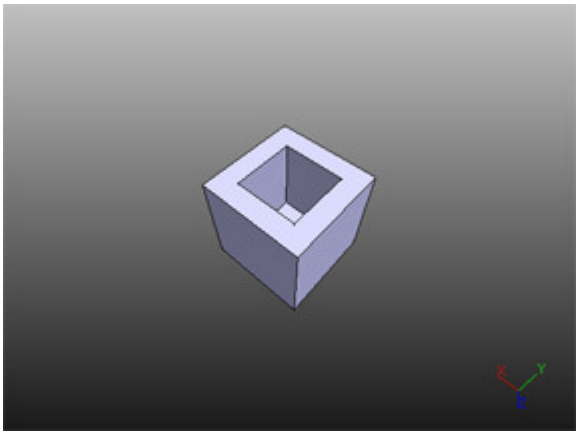
3. Grab the plane by clicking the left mouse button.



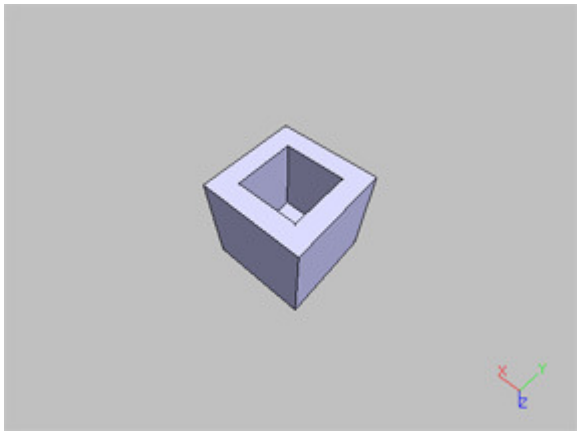
4. Move the plane while holding down the left mouse button.

Set Background

The background color and gradient can be selected in the **File->Preference** menu.



Gradient Option ON



Single Color Background

See Also: How to Use the Mouse, Setting Preferences

Menu

File Menu

New	Creates a blank document
Open	Opens *.gsc files
Close	Closes current document
Save	Saves current document
Save As	Saves current document in another name
Import	Imports CAD files(*.stl)
Export-Image	Saves current screen into BMP or JPEG file format
Print	Prints current screen
Print Preview	Provides preview of current screen to be printed
Preference	Sets environment including background color, database unit
Description	Inputs and edits description of current document
Exit	Closes all open documents and terminates program

Edit Menu

Copy to Clipboard	Copies current screen to clipboard
Merge Entity	Merges selected entities
Reset Transformation	Resets entity transformations including move, rotate, etc.

Mesh Menu

Set Entity	Sets properties of each entity
Set Mould	Sets size and type of mould
Set Domain	Sets domain size and symmetrical planes
Build Uniform Mesh	Builds uniform mesh
Build Variable Mesh	Builds variable mesh
Undo Meshing	Deletes built mesh, any simulation conditions set previously are not maintained

Simulation Menu

Basic Procedure : Basic simulation conditions

Task Design	Sets casting process and analysis type
Material Setting	Sets and changes properties of each entity from database
Initial and Plane BC	Sets initial conditions of each entity and boundary conditions of six planes
Heat Transfer Model	Sets heat transfer coefficient and coating conditions between entities
Gate Condition	Selects gate and sets thermal and flow conditions
Gravity Force	Sets gravity force

Optional Modules :

Fluid Flow Model	Sets viscosity/turbulence/surface tension models
Shrinkage Model	Sets solidification shrinkage defect prediction model
Microstructure Model	Sets microstructure prediction model
Outlet	Sets conditions when the melt flows out the domain
Cyclic Casting	Sets cyclic casting conditions
Rotational Force	Sets rotational force

Set Instruments : Sets various instruments

Sensor	Install and set sensor
Channel	Sets cooling or heating channel conditions (Activated when a CHANNEL ENTITY is defined)
Pouring Basin	Sets pouring basin conditions (Activated when a POURING BASIN ENTITY is defined)
Stopper	Sets stopper (Activated when a STOPPER ENTITY is defined)
Feeder	Sets feeder (Activated when a FEEDER ENTITY is defined)

Launching Conditions : Sets anySOLVER launching conditions

Solution Method	Sets solution method of heat and flow solver
End/Output Condition	Sets end condition and output conditions of result file
Run	Launches anySOLVER

Cross-Section Menu

Active	Activates cross section plane
Cross-Section YZ	Sets current plane to cross-section YZ(x-axis direction)
Cross-Section XZ	sets current plane to cross-section XZ(y-axis direction)
Cross-Section XY	sets current plane to cross-section XY(z-axis direction)
Flip	Flips currently visible area around plane
Section Plane	Shows/hides section plane
Section Grid	Shows/hides section grid
Reset	Resets plane to initial mode

Measure Menu

Vertex Coordinate	Indicates coordinates of one vertex
Distance between 2 Vertices	Indicates distance between 2 vertices
3 Vertices Angle	Indicates angle between 2 straight lines composed of 3 vertices
3 Vertices Radius	Indicates radius of circle composed of 3 vertices
Surface Area	Calculates total surface area of all entities except automatic build mold
Volume	Calculates total volume of all entities except automatic build mold

View Menu

Perspective	Switches current screen to perspective projection (perspective)
Isometric	Switches current screen to isometric projection (no perspective)
View Point	Sets view point of viewer (switches to perspective projection)
Add User View	Adds view point defined by user
Rotate Left	Rotates current screen 90 degrees to the left based on screen's normal vector
Rotate Right	Rotates current screen 90 degrees to the right based on screen's normal vector
Axial Rotate	Rotates current screen at a specific angle based on screen's normal vector
View Mode	
Geometry	Shows entities as geometries inputted
Mesh	Shows entities as mesh
Mesh Line	Shows mesh line
Zoom Window	Zooms in selected area
Fit to Window	Zooms in or Zooms out fit current screen size
Previous View	Returns to previous view
Reset View	Returns to initial view
Spinning	Rotate all entities at a selected speed and direction
Fast Moving	Turns on/off function to swiftly display current screen

Display Menu

Axis	Shows/hides axis on lower right area of screen
Auxiliary Axis	Shows/hides auxiliary axis in center of screen
World Axis	Shows/hides world axis which is located at 0,0,0
Vertex	Shows/hides vertex existing in space
Entity Extent	Indicates entity extent
Mold Extent	Indicates mold extent
Domain Extent	Indicates domain extent
Logo	Shows/hides AnyCasting logo

Window Menu

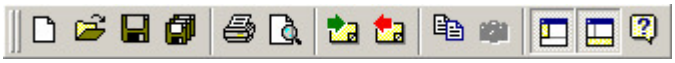
New Window	Creates new window
Cascade	Cascades all windows currently open
Tile Horizontal	Arrange windows as non-overlapping tiles horizontally
Tile Vertical	Arrange windows as non-overlapping tiles vertically
Arrange Icons	Arranges all icons

Help Menu

Contents	Shows online help contents
About	Shows version information and various contact numbers
License	Certification/disuse of license

Toolbar

Standard Toolbar



New	Creates new document
Open	Opens document
Save	Saves current document
Save All	Saves all documents
Print	Prints document
Print Preview	Previews print document
Import	Imports CAD file
Export	Exports current screen as image file
Copy to Clipboard	Copies current screen to clipboard
Shows/hides module window	
Shows/hides entity window	
About	About anyPRE

Measure Toolbar

Vertex Coordinate	Finds the coordinates of a vertex
Distance between 2 Vertices	Finds the distance between 2 vertices
Center of Segment	Finds the angle made by 2 lines composed of 3 vertices
3 Vertices Angle	Finds the angle made by 2 straight lines composed of 3 vertices
3 Vertices Radius	Finds radius of circle composed of 3 vertices

View Toolbar

Zoom Window	Zooms selected area
Fit to Window	Zooms in or Zoom out to current screen size
Previous View	Returns to previous view
Reset View	Returns to initial view
Spinning	Rotates all entities at a specific velocity and direction on screen
Axial Rotate	Rotates current screen at a specific angle based on screen's normal vector
Rotate Right	Rotates current screen 90 degrees to the left based on screen's normal vector
Rotate Left	Rotates current screen 90 degrees to the right based on screen's normal vector
View Point	Sets view point of viewer
Projection Mode	Switches to projection mode
Geometry Mode	Switches entity to view inputted geometries
Mesh Mode	Switches entity to view mesh
Fast Moving	Turns on/off function to swiftly display current screen

Display Toolbar

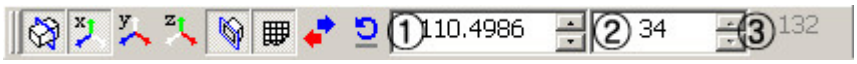
Entity Extent	Shows entity extent
Mold Extent	Shows mold extent
Domain Extent	Shows domain extent
Vertex	Shows/hides vertex existing in space
Axis	Shows/hides axis in lower right area of screen

Auxiliary Axis	Shows/hides auxiliary axis in center of screen

Mesh Toolbar

Merge Entity	Merges selected entities
Set Entity	Sets properties of each entity
Set Mold	Sets size and type of mold
Set Domain	Sets size and symmetrical plane of domain
Build Uniform Mesh	Builds uniform mesh
Build Variable Mesh	Builds variable mesh
Undo Meshing	Deletes built mesh

Cross Section Toolbar



Active	Activates view plane function
Cross-Section YZ	Sets current plane to cross-section YZ(x-axis direction)
Cross-Section XZ	Sets current plane to cross-section YZ(y-axis direction)
Cross-Section XY	Sets current plane to cross-section XY(z-axis direction)
Section Plane	Shows/hides section plane
Section Grid	Shows/hides section grid
Flip	Flips currently visible area around plane
Reset	Resets plane to initial state
①	Coordinates of current plane
②	Number of meshes of current plane
③	Number of meshes in currently selected direction

Entities

An entity is the smallest unit composing the casting system in anyPRE. It is created by reading CAD files. In case of box or cell mold, it is generated automatically.

Entity Introduction

Entity Properties

In order to build mesh or input simulation conditions, it is necessary to give properties to all entities.

CAST The domain which melt fills up

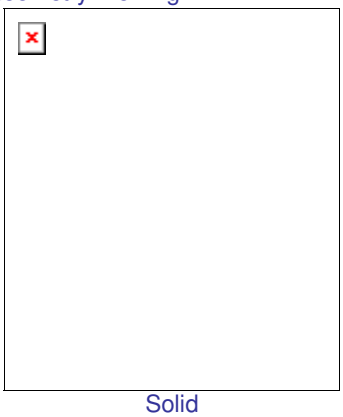
CAVITY	Cavity
INGATE	Ingate
OVERFLOW	Overflow
POURING_BASIN	Pouring Basin
STOPPER	Stopper
FEEDER	Part which feeds the melt in low pressure casting
GATE	Gate located inside of a mold (Inner-Gate)

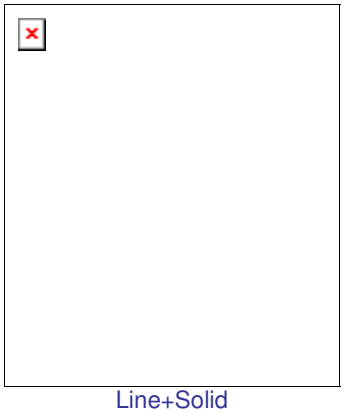
MOLD mold
INSERTED Material inserted into the cast such as core
ATTACHED Materials attached to the cast or mold such as chill
CHANNEL cooling or heating channels

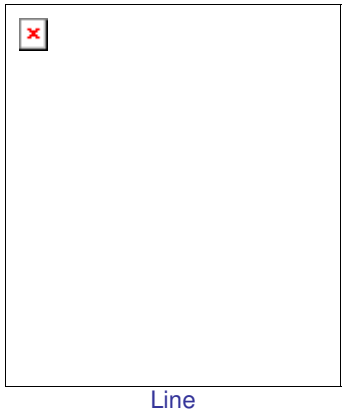
Viewing Entities

The rendering mode of each entity is as follows. You can change the render mode by the following two way.
1. Move the moue on top of entity and click on the right mouse button. Select the **Render Mode**.
2. Click on the **Set Entity** field. In Set Entity, change the **Mode** in **Display** field.

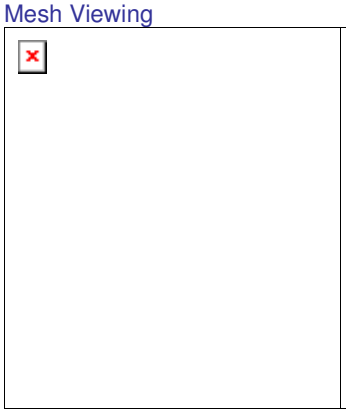
Geometry Viewing

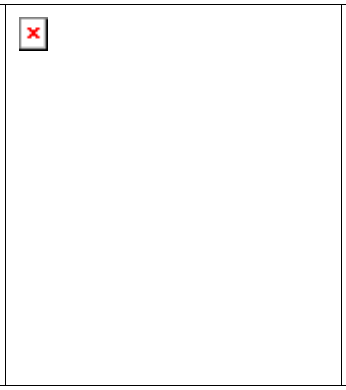

Solid

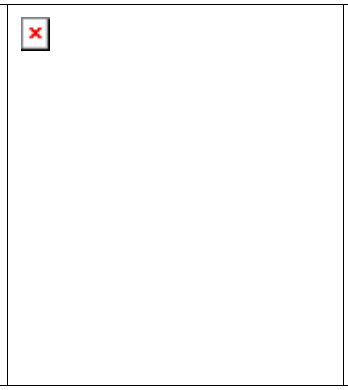

Line+Solid

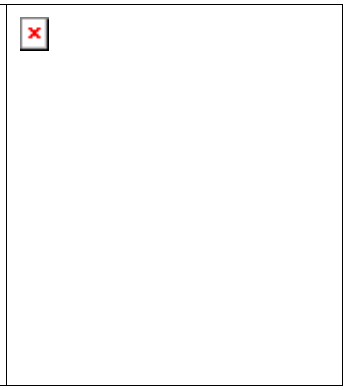

Line

Mesh Viewing


Solid


Line+Solid

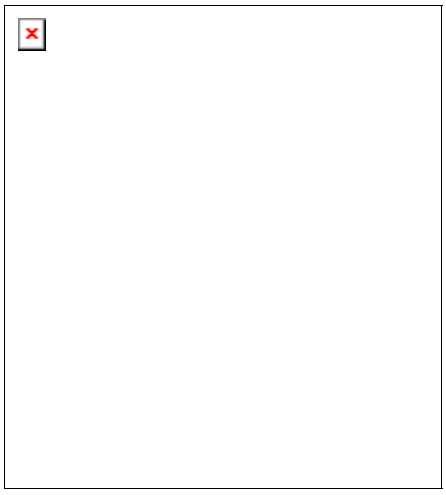

Line


Mesh Line

Using the Right Mouse Button Menu

When the mouse cursor is on top of an entity, the menu of that specific entity is displayed. The functions of each menu is as follows.

① Name and properties of entity



② Select render mode among solid, line + solid, or line

③ Change color

④ Control transparency

⑤ Hide entity (show: Have 'visible' field checked in the entity window)

⑥ Volume, surface area, modulus, number of triangles, number of cells

⑦ Move/rotate current entity

⑧ Replace entity's geometry keeping its attribute

⑨ Smash merged entity

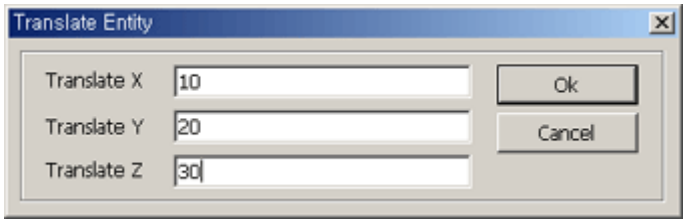
⑩ Delete entity

Moving and Rotating Entities

Place mouse cursor on top of entity that needs to be moved/rotate and click on right mouse button. Then, select *Transform* in entity menu.

Move

Relative Move the currently selected entity in the x, y, z directions by the relative displacement



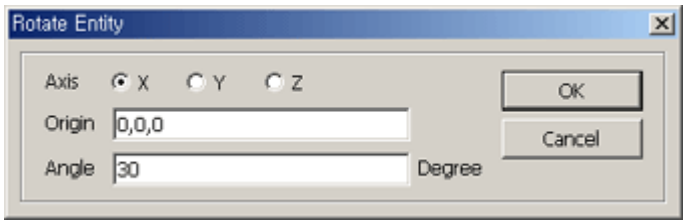
Vector Move the current entity by the distance between the two vertices selected with the mouse

Note

Esc key enable you to cancel and return back to default(rotate) mouse mode

Rotate

Rotate the currently selected entity with the origin of the rotational axis.



Axis Select rotational axis

Origin Input pivot point

Angle Input rotational angle

Undo Changes

to undo transformation :

1. Place mouse cursor on top of entity of which the changes need to be undone.

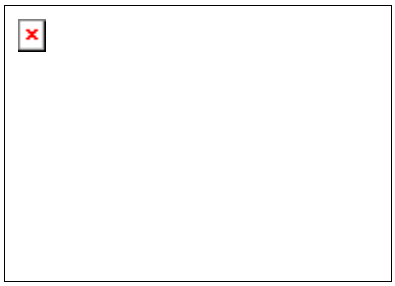
2. Click on the right mouse button to load the fast menu, then select *Transform -> Reset.*

Merging/Smashing Entities

Merge

Merge two or more entities into an entity with one property.

Select the entities to be merged by checking the box and click the 'Merge' button.



Select all Selects all entities

Smash (Undo Merge)

if you want to "undo merge", use "smash" of the merged entity's right-mouse button menu.

See Also : Entity Introduction

Measurement

Switch the mouse to the select mode by hitting the space key and select the white vertex. A measuring value will be displayed in the measuring output window.

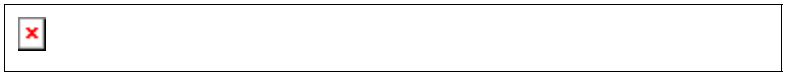
Value measured by Vertex

1. Select one of the below fields in the Toolbar of Measure Menu.

Vertex Coordinate	Show coordinates of one selected vertex
Distance between 2 Vertices	Show distance between two selected vertices
Center of Segment	Show center point of a segment
3 Vertices Angle	Show angle formed by 3 selected vertices
3 Vertices Radius	Show radius of circle composed of three selected vertices

2. Press the space bar and switch from mouse mode to select mode.

3. Click on the indicated vertex and the measurement will be displayed as follows:



Value measured by Entity

1. Move the mouse on top of entity that needs to be measured.
2. Click on the right mouse button and select *Volume, Surface Area, Modulus* in property fields of the fast menu.

Value of total Entity

1. Select the **Measure** Menu

Volume Total volume of all entities except castings that have been created automatically

Surface Area Total surface area of all entities except castings that have been created automatically

2. Values of each entity are displayed in the volume and surface area fields in the lower entity window

---

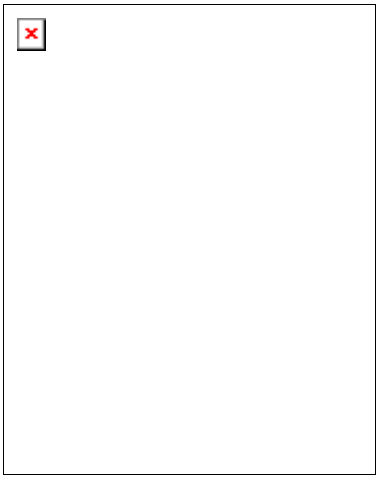
See Also How to Use the Mouse

Mesh Generation

A hexahedron of a certain size, a mesh is the smallest unit that composes the simulation domain.
As it has the largest effect on the reliability of the simulation results, you must fully understand.
After a mesh has been built, it must always be checked by cross section and other observation function.
As the quality of the mesh depends on the ability of the user, you must read, learn and practice repeatedly.

Setting Entity Properties

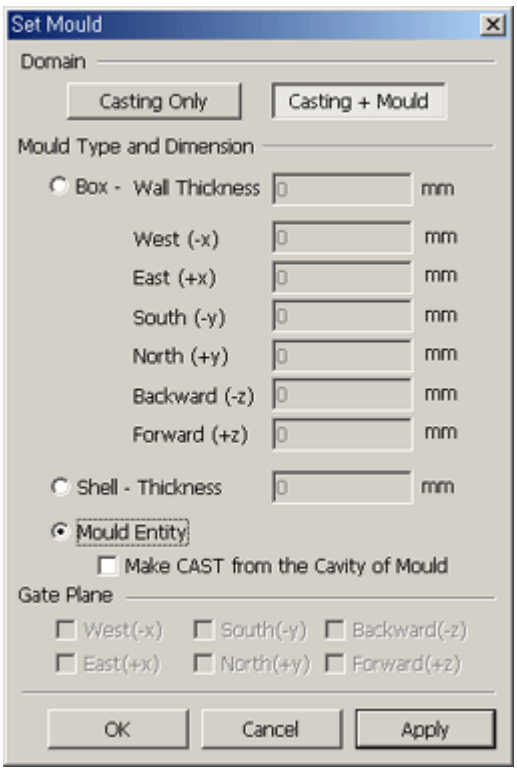
Double-click the entity or select **Mesh -> Set Entity** from the menu.
The properties, color, names of the entities in the system will be shown.
Select the properties of each entity in combo box of **Type** field.



Name	Entity name
Type	Select entity property. Auxiliary properties can be set in case of a cast.
Display	Change the display properties of the current entity.
Visible	Show/hide
Mode	Select rendering properties
Color	Select color
Opacity	Set opacity

See also : Entity Introduction

Setting the Mould



Domain

Casting Only	choose when there is only a casting
Casting+Mould	choose when there is a casting and a mold

Mould Type and Dimension

Box	Making a rectangular mold You can enter the wall thickness of each direction
Shell	Making a shell mold You can enter the shell thickness
Mould Entity	Select mold made by reading the geometry from a CAD file Possible to select only when the entity properties have been set as mould

Make CAST from the Cavity of Mould
Check it if you want to make CAST from the mould cavity (inverse meshing), and imported mould geometry must be a box shape.

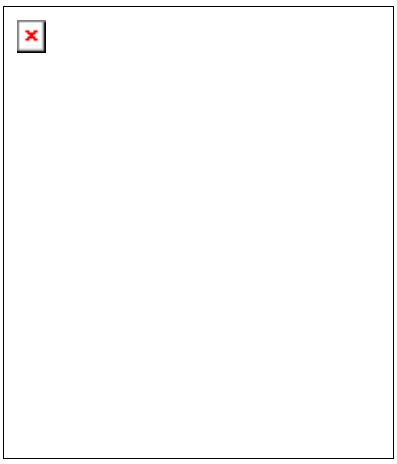
Note
In case of cyclic casting, there has to exist two or more mold entities. "Mould Entity" must be choosen.

Gate Plane
Select the plane where the gate exists and the wall thickness automatically becomes '0'.

See Also: Setting Cyclic Casting

Setting the Domain and Symmetrical Planes

Of the entire domain, create the new domain by selecting a interested area by inputting the coordinates or the ratio for each wall.
If the geometry is symmetrical, half of domain can be solved.



Information

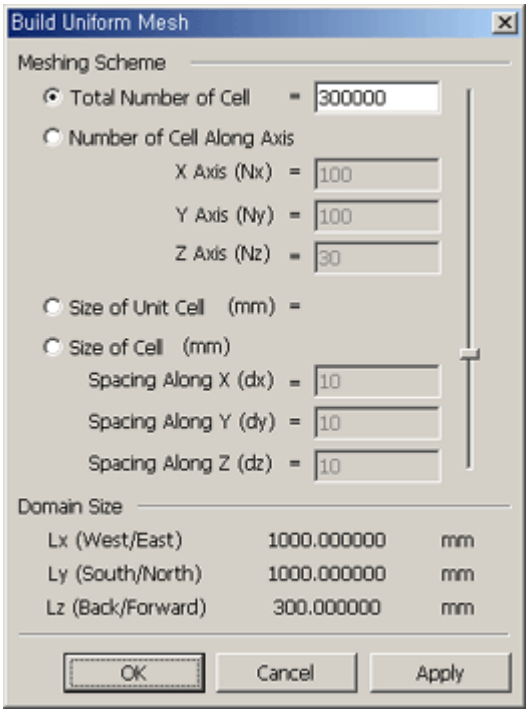
Displays the size of the entity and mold.

Domain and Symmetry

Coordinate	Coordinates of each boundary plane
Ratio	The ratio of the size of the entire area to the location of the current boundary plane
Symm.	Of the 6 planes, select the symmetrical planes by checking the boxes

Building a Uniform Mesh

Divide the selected domain into meshes of a certain size. Enter the values in each field and click on 'Apply' or hit the 'Enter' key and the optimized mesh size will be displayed.



Note
When the cast entity becomes divided two or more isolated parts during building a mesh, a warning message pops up and each area is displayed in different colors.

Meshing Scheme

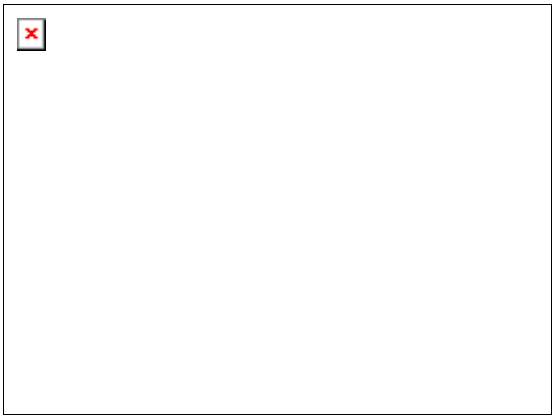
Total Number of Cell	Sets the total number of cells in the domain Move the slide bar on the right up and down or enter the value directly into the field
Number of Cell Along Axis	Sets number of cells along each axis
Size of Unit Cell	Sets size of unit cell for cube
Size of Cell	Sets size of cell along x,y,z directions

Domain Size

Show domain size

Building a Variable Mesh

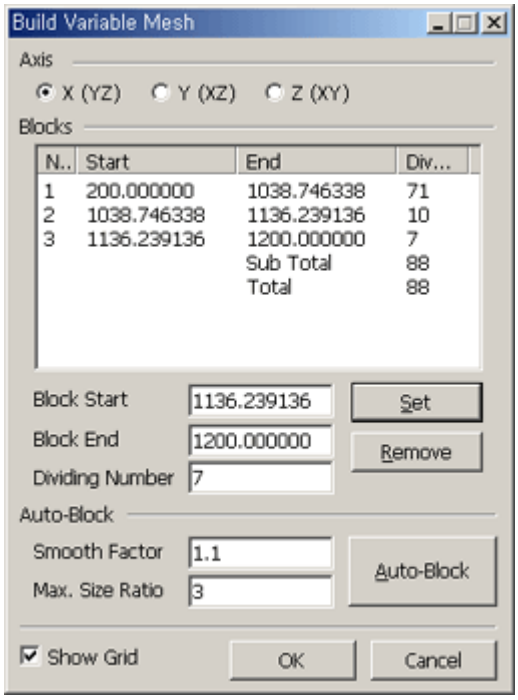
It is possible to focus meshes in a certain area of the domain and decrease the number of meshes by applying an auto-block. It raises the accuracy and reduces the calculation time of the simulation.



Types of Blocks

User Block	Block that has been built as a result of user-defined block setting the Start and End points. Displays in blue.
Auto-Block	Block that has been automatically built by clicking on the **Auto** button. Displays in gray

Dialog



Axis	Select the current axis of direction
Block Setting	
Start	Start point of the block
End	End point of the block
DivNumber	Number of divisions of the block
Set	Set the block according to the start point, end point and division number
Remove	Current Block : remove currently selected block All Blocks : remove all blocks Auto-Block : remove only automatically created blocks
Show Grid	Show/hide the grids of the block
Smooth Factor	Increase ratio of neighboring cell sizes
Max. Size Ratio	Ratio of Maximum mesh size to the smallest mesh in auto block
OK	Build mesh when blocks settings are complete
Cancel	Maintain current block settings and exit without building mesh

Build Block

User Block

1. Select the axis.

2. Press the space bar and switch from mouse mode to select mode.

3. Of the white vertices, select two vertices. The smaller is the starting point and the bigger is the end point.

4. After setting the division number, press the 'Set' button or 'Enter' key. (the newly build block will be displayed on screen)

Auto-Block

After completing setting the user block, click on the 'Auto' button and a block will be automatically created for the empty area.

Build Mesh

1. Create a user block.

2. Create an auto-block.

3. Repeat steps 1 and 2 for each axis and then click on the 'OK' button to build mesh.

Note

Should a block with red marks be created, it means that mesh couldn't be built automatically. In that case, you must modify the following values.

① User sets the division number

② Change Factor and Ratio

③ Change division number of blocks at each end

After modifying these values, you build the mesh again.

Note

When the cast entity becomes divided two or more isolated parts after building a mesh, a warning message pops up and each area is displayed in different colors.

Change Block

Change the values for the **Start, End, and DivNum** fields, then click on the **Set** button.

Remove Block

Remove current block	Click on 'Remove' button then select *Current-Block*
Remove Auto-block	Click on the 'Remove' button then select *Auto-Block*
Remove all blocks	Click on the 'Remove' button then select *All Blocks*

Setting Simulation Conditions

There are 4 basic categories to set for simulation conditions.

Basic Procedure

Task Design
Select Material
Initial Conditions and Plane Boundary Conditions
Heat Transfer Model
Setting the Gate
Setting Gravity Force

Optional Module

Setting Flow Models (Viscosity, turbulence, surface tension)
Solidification Shrinkage Prediction Model
Setting Cyclic Casting
Setting Outlets
Setting Rotational Force

Set Instruments

Setting Sensors
Setting Cooling Channels
Setting Pouring Basins
Setting Stoppers
Setting Feeders

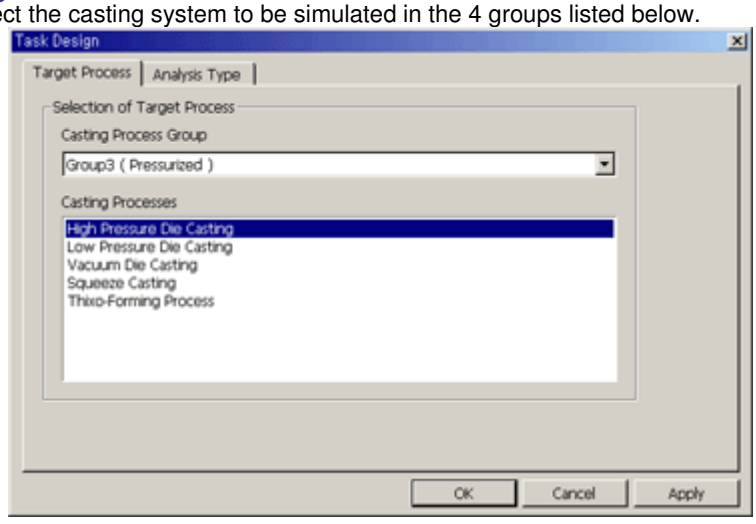
Launching Condition

Setting Simulation Method
Simulation End and Result Output Conditions
Launching anySOLVER

Task Design

Target Process

Select the casting system to be simulated in the 4 groups listed below.

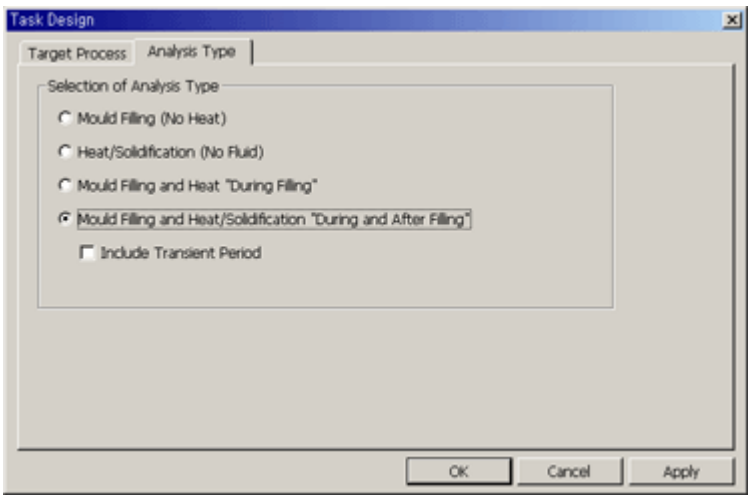


- **Group 1 (Non-Metal Mould)**
  - Sand Casting
  - Investment Casting
- **Group 2 (Metal-Mould)**
  - Metal-Mould Casting
  - Gravity Tilt Pour Casting
- **Group 3 (Pressurized)**
  - High Pressure Die Casting
  - Low Pressure Die Casting
  - Vacuum Die Casting
  - Squeeze Casting

- Thixo-Forming Process

- **Group 4 (Special)**
  - Centrifugal Casting
  - Electro Slug Melting Process

Analysis Type

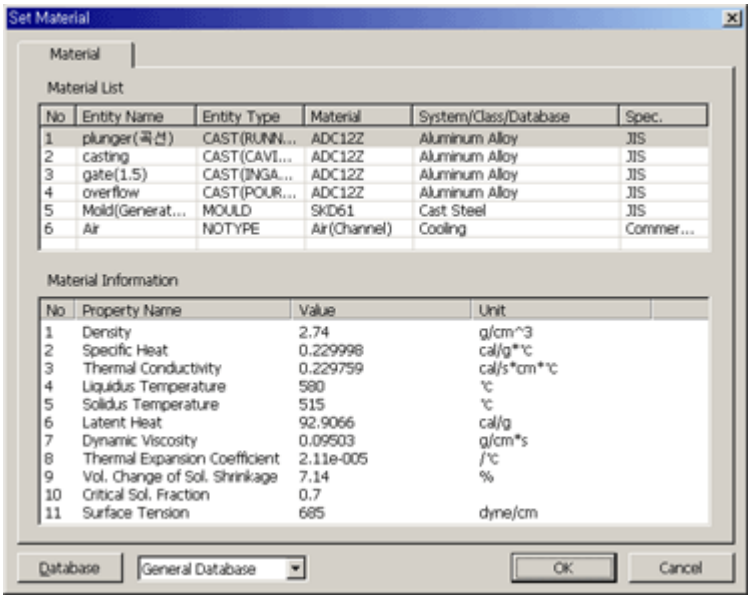


- **Mould Filling (No Heat)** Select when only carrying out a filling simulation
- **Heat/Solidification** Select when only carrying out a heat/solidification simulation
- **Mould Filling and Heat "During Filling"** Select when only carrying out a filling and heat simulation during filling
- **Mould Filling and Heat / Solidification "During and After Filling"** In this type, a filling and heat/solidification simulation is carried out during filling. After filling, only heat and solidification simulation is performed
- **Include Transient Period** In this type, a filling and heat/solidification simulation is carried out during and after filling.

Note The **Include Transient Period** option is selected when the flow patterns of the mushy zone plays an important effect on solidification after filling. Thus, the result of this option may be better than those of above other options. But it takes a lot of simulation time. We recommend using the **Mould Filling and Heat / Solidification "During and After Filling"** option.

Selecting Materials

Set the material of a selected entity in the database.



Selecting and Changing an Entity Material

1. Select the entity in the list. Multiple selections are possible using ctrl or shift + left mouse button.
2. Click on the **Database** button or double click the selected entities.
3. Select the material in the Material Selection box.
4. Double-click on the material or click on the 'OK' button with selecting material. The selected entity will be set.
   - The properties of air are set automatically.

Changing the Properties of the Material

1. 11 properties appear in the Material Information list.
2. Double click on each property and a View / Edit window will appears as below.
3. Click on the 'OK' button and complete the change. (the 'OK' button activates only when the materials of each entity have all been set.)

Using User Defined Material Information

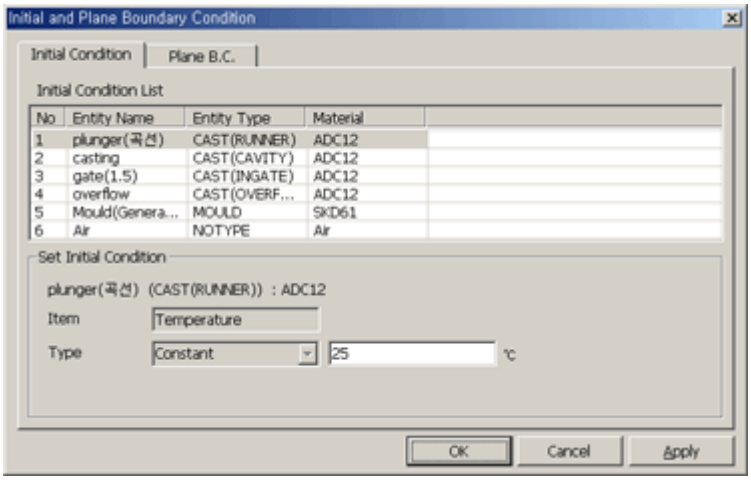
1. Load the preferences window in the File-Preference menu.
2. Click on ... in the User Database menu and select the *.mdb file created in anyDBASE.
3. Select User Database in the list General Database.

Note Material information is not saved in case the Undo Meshing function is executed

Initial Conditions and Plane Boundary Conditions

Initial Conditions

Set the initial thermal conditions of each entity.
If it is an entity with cast properties, assume that it is filled with air as it is empty before filling.
On the contrary, if only carrying out a heat/solidification simulation(select **Heat/Solidification** field in Analysis Type), enter the pouring temperature of melt as it has already been filled.
If preheating the mould, enter the preheating temperature.

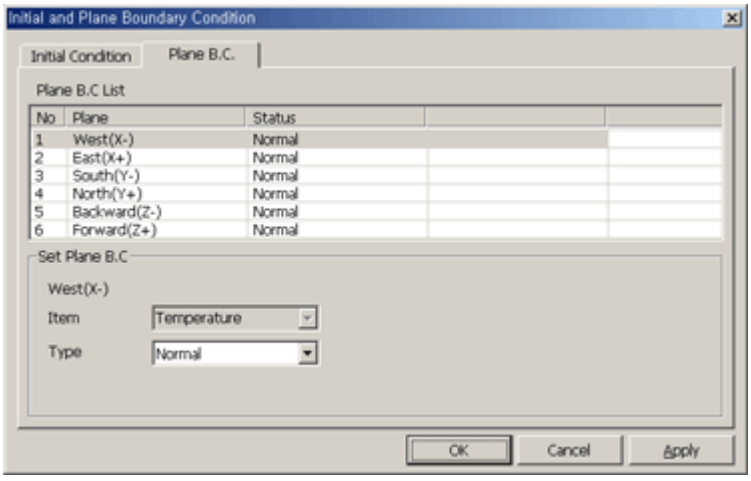


Item (Thermal Condition)

Temperature	Enter initial temperature. Default value is 25℃.

Plane Boundary Conditions

Set the thermal conditions of each plane(±x, ±y, ±z).



Item (Thermal conditions)

Normal	Heat transfer including convection and conduction
Adiabatic	Adiabatic plane

Constant Temp.	Constant temperature
Variable Temp.	Time dependent temperature
Constant Flux	Constant heat flux
Variable Flux	Time dependent heat flux

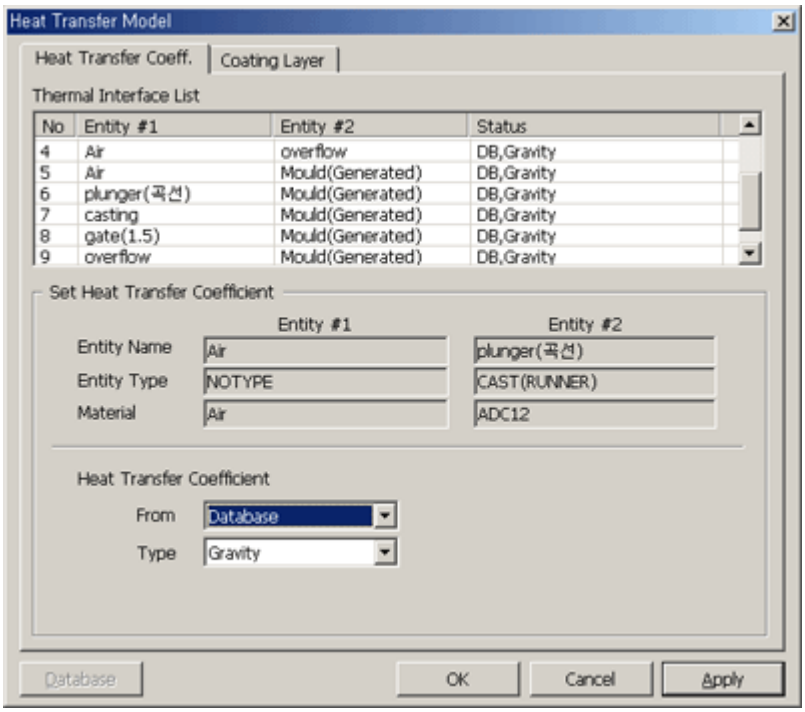
See Also Inputting Variable Data, Task Design

Setting Heat Transfer Model

This function automatically finds and displays the interfaces between each entity. Multiple selection of the interfaces are possible.

Setting the Heat Transfer Coefficient

The coefficient can be imported from the heat transfer coefficient database of each interface or you can set it up.

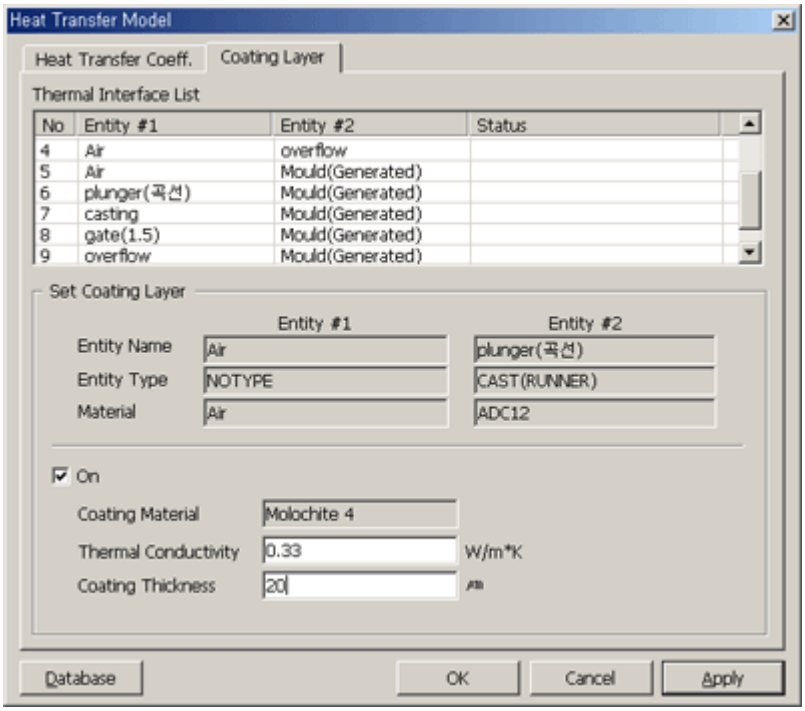


Heat Transfer Coefficient

Database	Automatically sets the HTC according to the two adjacent materials
	Select from gravity and low/high pressure depending on the process
User defined	Defined by user

Setting Coating Layer

Define the heat transfer and coating thickness of the materials to be coated on the various interfaces or one interface.
Double-click on the interfaces in the list or click on the **Database** button with selecting the interface. And then select the materials from the 'Coating' group.



Coating Material	Name of coating material
Thermal Conductivity	Thermal conductivity of coating material
Coating Thickness	Coating thickness of coating material

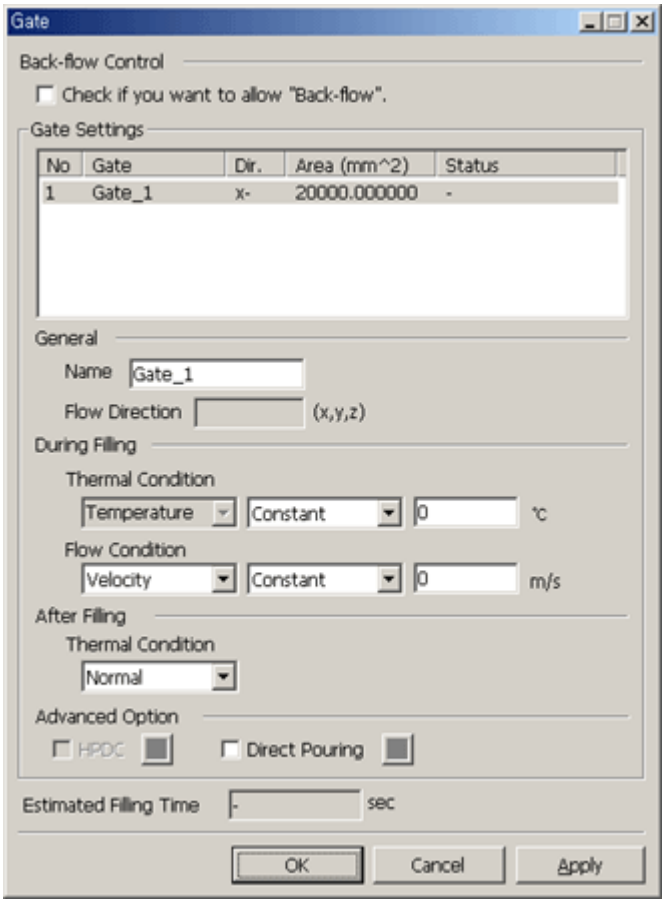
Setting the Gate

Gate conditions can be largely categorized into two levels: during filling and after filling.
It is possible to use the mouse to add/delete on the screen. The program supports multiple gates and enables setting of direct-pouring and modules for the optimized simulation of die casting processes.

Adding/Deleting Gates

1. Select *Set Gate* from the menu and gate area is displayed in red on the window.
2. Press the space bar and switch from mouse mode to select mode.
3. Select the area in red using the mouse and add it to the list.
4. If you delete the current gate in yellow, select it again.

Setting Gate Heat/Flow Conditions



Name
Set the gate name. The default value is "Gate_number".

During Filling (before cavity is filled up completely)

Thermal Condition	Set the thermal conditions of the melt
Flow Condition	Set the flow conditions among velocity, pressure and height

After Filling (after melt filling is completed)

Thermal Condition	Set thermal conditions of the gate plane exposed to the outside
Normal	Heat transfer conditions including convention, conduction, and radiation
Adiabatic	Adiabatic condition
Constant Temp.	Constant temperature
Var Temp.	Time dependent temperature
Constant Flux	Constant heat flux
Var. Flux	Time dependent heat flux

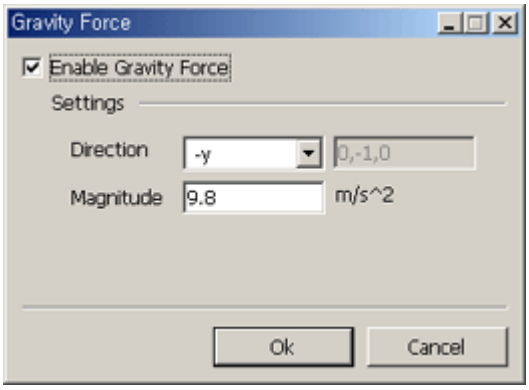
Option

HPDC	Velocity condition optimized in high pressure die casting(HPDC)
Direct Pouring	Gravity Casting using the Lip Pouring Ladle
Filling Time	Indicates estimated filling time when the velocity is constant.

See Also Inputting Variable Data

Setting Gravity Force

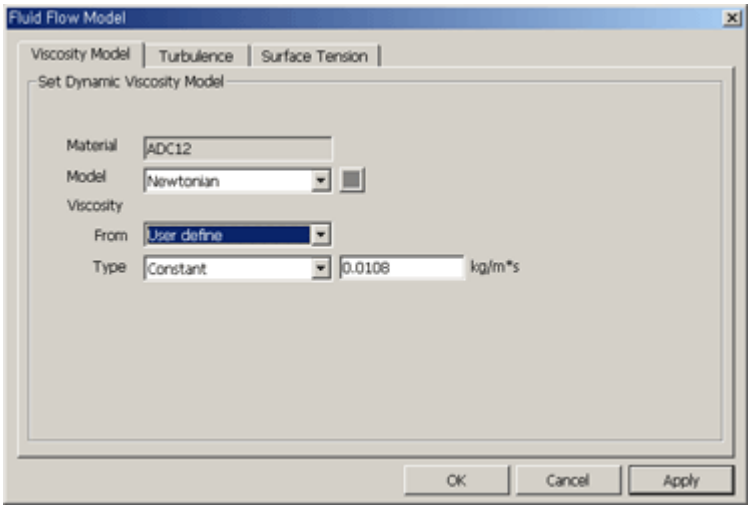
The gravity force is set.



Enable Gravity Force Turns on/off gravity force
Direction Select direction or input vector
Magnitude Input magnitude of gravity force

Viscosity Model

This is where the viscosity for a non-Newtonian flow is set.



Material Indicates material of entity set as cast
Model Select viscosity model

Newtonian	Select in most metal castings, default value
Effective Viscosity	
Bingham	
PowerLaw	
Carreau	
Ellis	

Viscosity

Database	Use viscosity value imported from the database
User define	Input user defined viscosity value

Note
The 5 viscosity models excluding the Newtonian flow are as follows:

Effective Viscosity Curve Model
The change in viscosity for a solidification fraction have been modeled as a function

$$\mu = \mu_o \cdot \left[1 + C_1 \cdot \left(1 - \left(0.5 - \frac{\tan^{-1}(C_2 \cdot (fs - C_3))}{\pi}\right)\right)\right]$$

μ: viscosity value

C_1, C_2, C_3 : model consonant

Bingham Viscosity Model

, $\gamma_{ij} = \left(\frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_i} \right)$ (Shear rate)

: viscosity value

: yield stress of material

Carreau Viscosity Model

viscosity value

viscosity value when shear rate reaches infinity

model constant

Ellis Viscosity Model

,

viscosity value

for

model constant

Powerlaw Viscosity Model

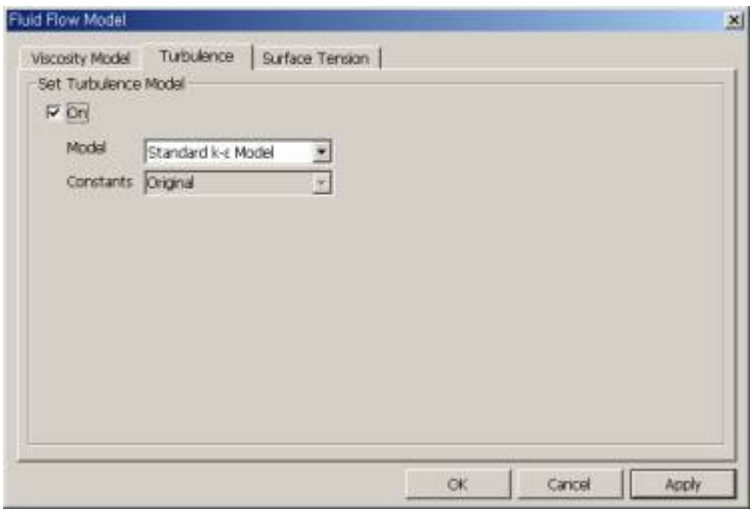
,

: viscosity value

: model consonant

Turbulence Model

This is where the turbulence model is set.



On Turn on/off turbulence model

Model Select turbulence model

Standard k-ε model

RNG k-ε model

Wilcox k-ω model

Constants Constants used in a turbulence model

Note

There are various types of formulas used in turbulence models. For the Standard k-ε model, the most standard model of turbulence models, there are two differential equations and one equation on turbulent viscosity.

$$\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_s}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{s1}P_k\frac{\varepsilon}{k} - C_{s2}\frac{\varepsilon^2}{k}$$

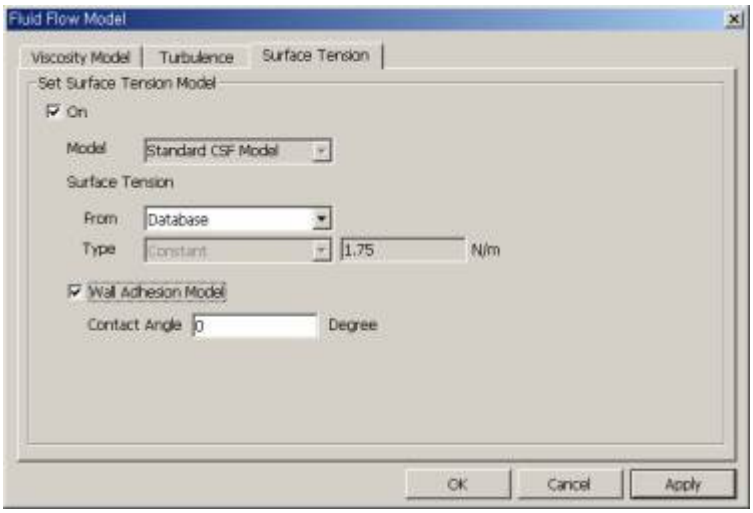
$$\frac{\partial k}{\partial t} + u_j\frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \varepsilon$$

The consonant values are as follows:

	Standard k-ε	RNG k-ε	Wilcox k-ω
C_{s1}	0.09	0.085	0.09
C_{s2}	1.44	1.42	0.56
C_μ	1.92	1.68	0.075
σ_k	1.00	0.7179	2.00
σ_s	1.30	0.7179	2.00

Surface Tension Model

This is where the coefficient of surface tension and the wall adhesion angle are set.



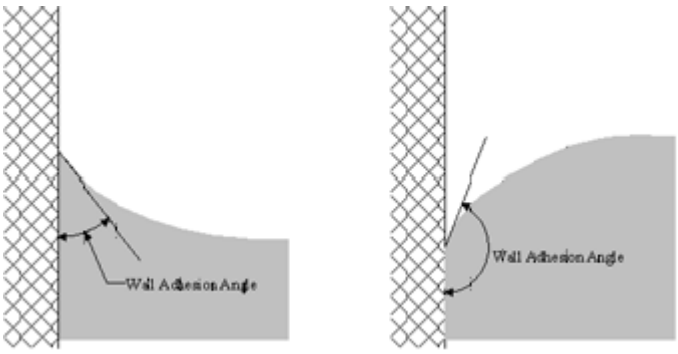
On	Turn on/off surface tension model
Model	Standard CSF Model
Surface Tension	Select method to enter surface tension
Database	Use value imported from database
User defined	Input user defined value
Wall Adhesion Model	Set whether or not wall adhesion force will be simulated Enter angle in activated input field

---

Note
Surface tension can be expressed as follows:

$\sigma = C_\sigma \cdot K$ C_σ : Coefficient of Surface Tension, : **K** : Curvature

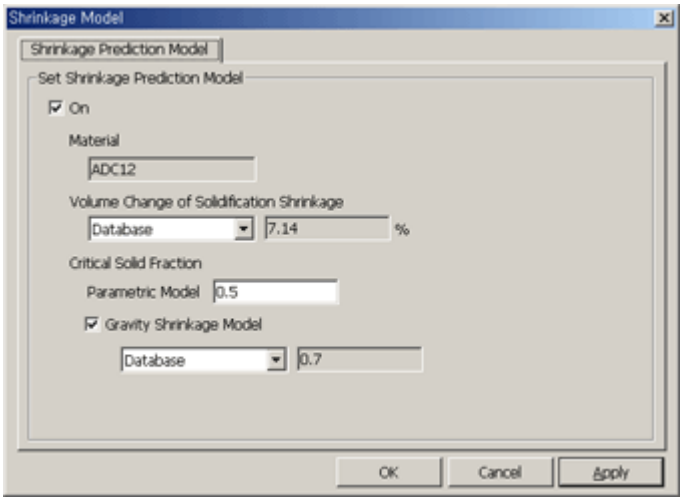
The surface tension induces the wall adhesion. By setting the wall adhesion angle at the point where the melt contacts with the wall, it is possible to reflect the wall adhesion effect in the simulation. The wall adhesion angle is the angle indicated below.



Definition of Wall Adhesion Angle

Solidification Shrinkage Model

This model is used to predict solidification shrinkage.



On	Turn on/off shrinkage model
Material	Material of casting

Volume Change of Solidification Shrinkage

DataBase	Use value inputted from database
User defined	Input user defined value

Critical Solidification Fraction

Critical solidification fraction is categorized as follows:

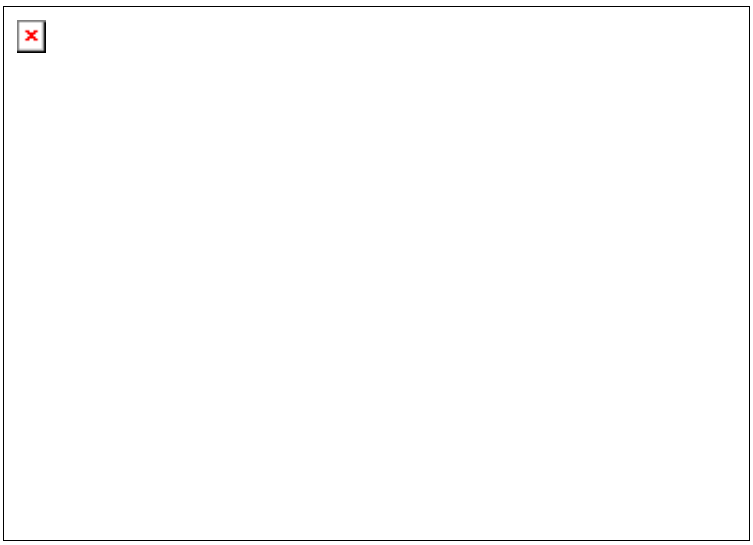
Parametric Model	Solid fraction that records the basic parameters necessary in predicting shrinkage
Gravity Shrinkage	Percentage that gravity shrinkage takes up of the entire shrinkage
DataBase	Use value inputted from database
User defined	Input user defined value

Microstructure Prediction Model

The deterministic model and parametric model are models used to predict microstructure.

Deterministic Model

The size of a grain or density of spheroidal graphite nodules can be found using this model.

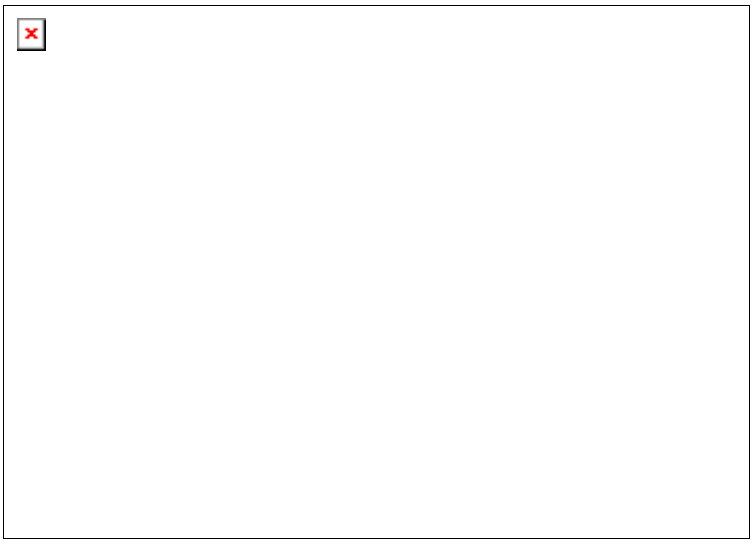


On	Turn on/off deterministic model
Model	Select deterministic model
Alloy System	Select alloy system
Al-Cu	Al - 4.5wt%Cu alloy standard
Al-Si	Al - 7wt%Si alloy standard
Ductile Cast Iron	Spheroidal graphite iron
Magnesium Alloy	AZ901D standard
User defined	Use new values other than those provided above
NucleationKinetics	Enter data on nucleation kinetics
Growth Kinetics	Enter data on growth kinetics

Note
The above model is based on the nucleation and growth kinematics.
Thus, if the data on both kinetics are valid, it is possible to obtain reliable results. You must carefully enter and select the data.

Parametric Model

It is possible obtain information on various types of microstructure using this model. Solute diffusion, heat diffusion, and length of the capillary tube are obtained and used as the basic parameters to predict the microstructure.



On	Turns on/off parametric model
Model	Select parametric model
Alloy System	Select alloy system
Al-Cu	4.5wt%Cu basis
Al-Si	7wt%Si basis
Ductile Cast Iron	Ductile Cast Iron basis
Magnesium Alloy	AZ901D basis
User Define	Use new value other than those provided above

Parameters

Gibbs-Thomson Coefficient	Super-cooling coefficient resulting from capillary tube effects
Initial Solute Concentration	Initial solute concentration
Solute Diffusivity in Liquid	Solute diffusivity in liquid
Liquid Slope	Liquid slope
Partition Coefficient	Partition coefficient
Interatomic Distance	Interatomic distance

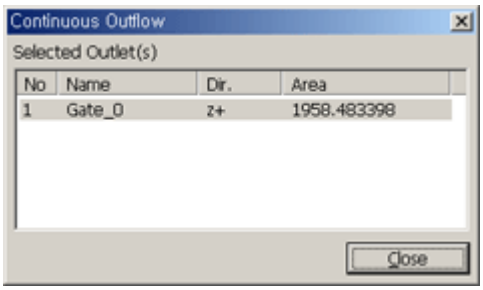
Setting Outlet

Like a vent or a exit, an outlet is the area where the melt flows to the outside.

Select/Delete Outlet

1. Open the *Outlet* window.
2. The areas that may be selected are indicated in brown.
3. Press the space bar and switch from mouse mode to select mode.
4. Click on the brown area. (That will be selected and added to the list.)
5. If you delete the current outflow in yellow, select it again.

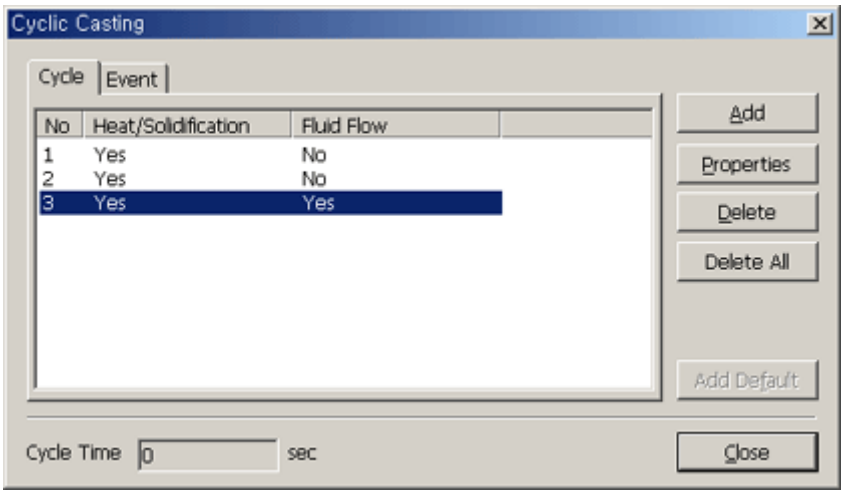
Setting the Outlet



No	Number of outlets
Name	Name of each outlets(default value - outlet #)
Dir.	Plane directions
Area	Cross-sectional area of each outlets

Setting Cyclic Casting

This is where the general conditions for cyclic casting are set.



Setting Cycle

Add a Cycle Click Add Button, default analysis type is "Heat/Solidification only".
Delete a Cycle Click Delete or Delete All button.
Modify Analysis Type of Each Cycle Double click an item, or click Properties button then it shows dialog.

Note
It is very general that set "Heat/Solidification + Fluid Flow" for the last cycle only.

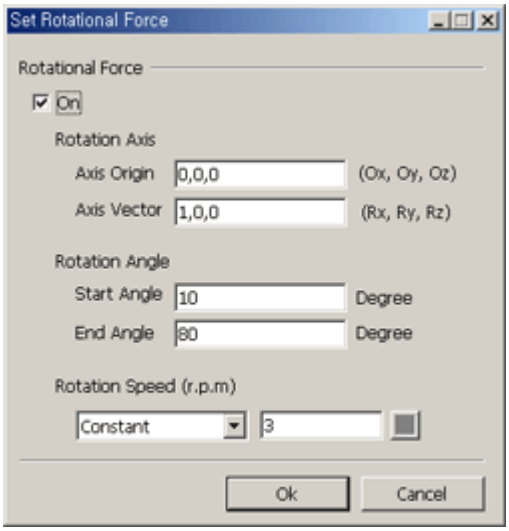
Setting Events

Every Event must have a happen time and/or duration. You can Add/Delete an event using Add/Delete button.
Click Add Default button to add default events (Die-Open, Die-Open, Spraying, Die-Close, Lead Time).
Cycle Time is automatically calculated and shown at the bottom part of above dialog.

for more detail, click followings
Event in Cyclic Casting
Event Diagram

Setting Rotational Force

The rotational force is set in this section.

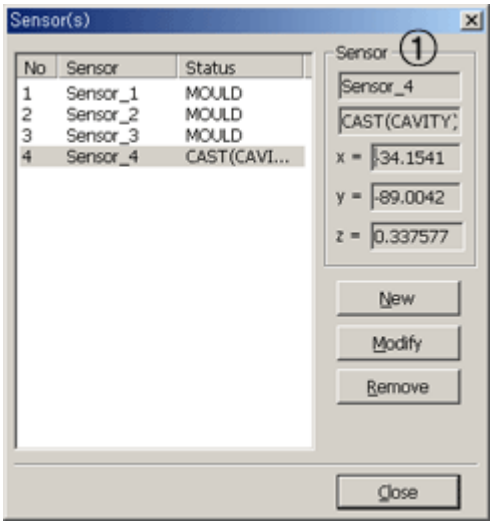


On	Turn on/off rotational force
Rotation Axis	Select rotation axis
Axis Origin	Axis origin
Axis Vector	Axis vector ex) Rotation around ±x axis ±1,0,0 Rotation around ±y axis 0,±1,0 Rotation around ±z axis 0,0,±1
Rotation Angle	Select rotation angle
Start Angle	Angle where rotation starts
End Angle	Angle where rotation ends
Rotation Speed	Set rotation speed(R.P.M)
Constant	Constant rotation speed
Variable	Click on button ⌐

See Also Inputting Variable DAta

Setting Sensors

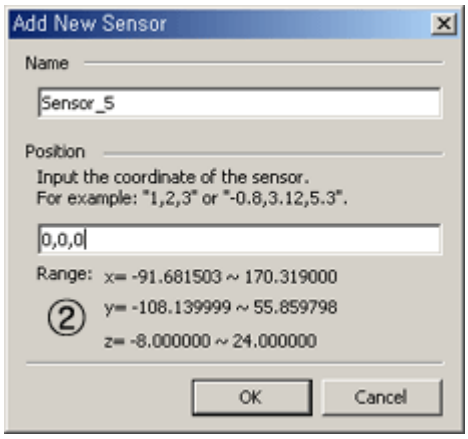
Using sensors, you can check velocity, pressure and temperature at specific positions. In this section, learn how to embedded in the interested positions.
Click on the **Simulation -> Set Instruments -> Sensor** menu. The cross-section plane will be activated and the sensor input window will load.



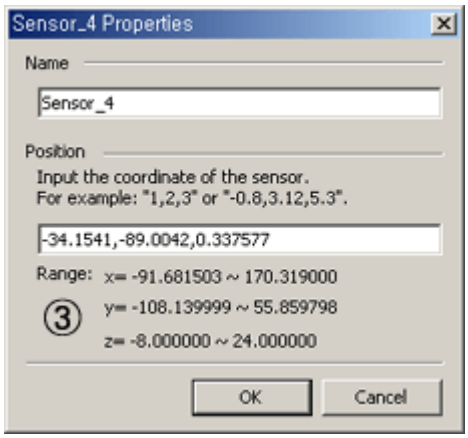
Adding a Sensor

Adding a Sensor Using the Mouse
1. Move the activated section to the area of interest.
2. Press the space bar to switch from mouse mode to select mode.
3. Click on the point of interest and it will be added to the list
4. In field ①, it is possible to check the sensor name, coordinates and entity under which the sensor is located.

Adding a Sensor Using Coordinates
Click on the 'New' button. In the following dialogue, enter the name(default value: Sensor_#) and coordinates of sensor to be added. The available range of the coordinates will appear in ②.



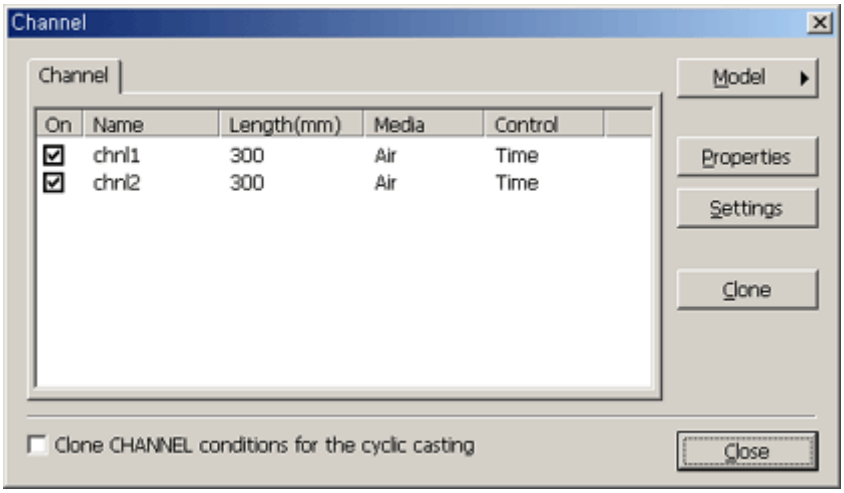
Change Position of a Sensor
Click on the 'Modify' button and modify the name and position of the current sensor. The available range of the coordinates will appear in ③.



Removing a Sensor
Click on the 'Remove' button and the current sensor will be removed.

Setting Cooling/Heating Channels

The operation conditions of cooling/heating channels are set in this section.
CHANNEL type entities are shown in the list.



1. Turn on/off by clicking Check button on the 1st column of each entity.
2. Select Channel Model. (Constant HTC Model, Constant Flux Model)
3. Set other conditions
4. Check "Clone CHANNEL conditions for the cyclic casting" button if you want to adjust time-dependent channel conditions automatically.

For more detail, click below.
Setting Conditions
Clone Channel Conditions

Note
If you turned off the channel, it could not be activated even you input all conditions.
it treated as a cavity filled with air.

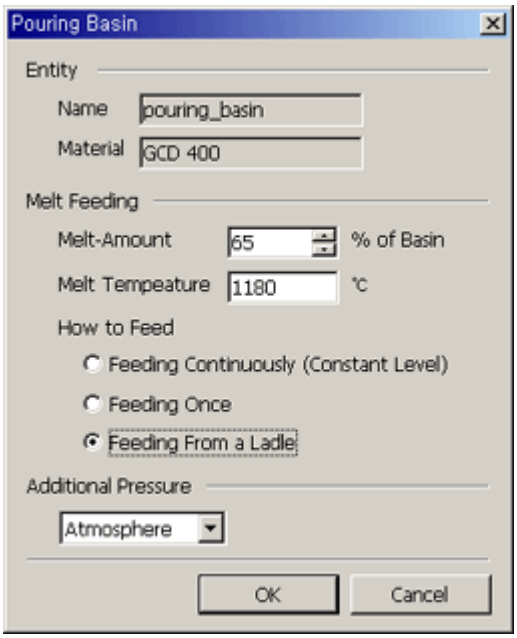
The heat transfer model of a channel is categorized into the Constant HTC and Constant Flux models.

Constant HTC Model
In this model, heat transfer coefficient, h is found by correlation equation with inlet/outlet temperature, inflow flux, cross-section shape of channel. With the found HTC value between adjacent entities, heat transfer is calculated.
The strong point of this model is that the flux towards the channel changes according to the temperature of the entity that lies adjacent to the channel. On the other hand, its weak point is that it fails to hold a generality because it uses the experimental correlation equation.

Constant Flux Model
In this model, total heat flux from the channels to neighbor entities is found from the channel's inlet/outlet temperature.
Unlike the HTC Based Model, this model does not use the experimental correlation equation. This means that it may be applied generally regardless of the shape of channel. However, there exists the weak point that the heat flux between channel and neighbor entities is calculated constantly.

Setting Pouring Basins

This is where the working conditions of a pouring basin are set.



Entity

Name	Indicate name of entity set as pouring basin
Material	Material of pouring basin

Melt Feeding

Melt-Amount	This has two meaning 1. Volume of melt fed into the pouring basin initially for **feeding continuously** or **feeding once** option. 2. Volume of melt at that the feeding is stopped for **Feeding From Ladle** option
Melt Temperature	Temperature of the molten alloy
How to Feed	
Feeding Continuously (Constant Level)	The volume of the melt in the pouring basin is maintained at a constant level
Feeding Once	The pouring basin is fed once at the beginning with a certain amount of melt, then no longer feeding. The amount of melt in the pouring basin changes.
Feeding From Ladle	Feeding the pouring basin from the ladle The gate condition must be set on the gate plane.

Note
Melt-Temperature is set as initial temperature of the POURING_BASIN type entity if you 'Feeding Continuously' or 'Feeding Once'. if you choose 'Feeding from a Ladle', the temperature is correspond to the pouring temperature of the gate.

Pressure

To force the melt to enter into the cavity, the empty volume of pouring basin is filled by pressurized air. In this field, set the pressure value.

See Also Types of Input Values, Inputting Variable Data

Setting Stoppers

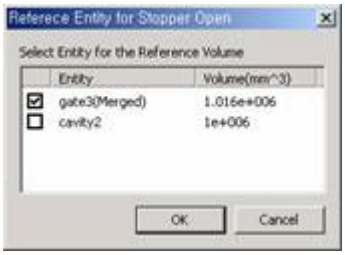
This is where the working conditions of the entity set as a stopper are set.

Settings

Set the conditions when the stopper opens.

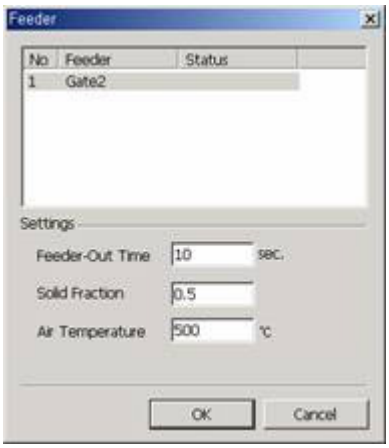
Holding Time	The stopper opens when the time set by the user expires.
Volume Fraction	The stopper opens when a specific entity is filled with more than a certain amount of melt

When the volume fraction function is set, click on the 'Select Entity' button and the cast-type entity will appear in the window below. Choose one or more entities to be used as objects in volume fraction



Setting Feeders

This is where the working conditions of the entity set as the feeder are set.



Feeder-Out Time	Time when melt inside feeder is removed
Solid Fraction	Standard solidification when melt inside feeder is removed
Air Temperature	Temperature of air to be filled inside feeder after melt has been removed

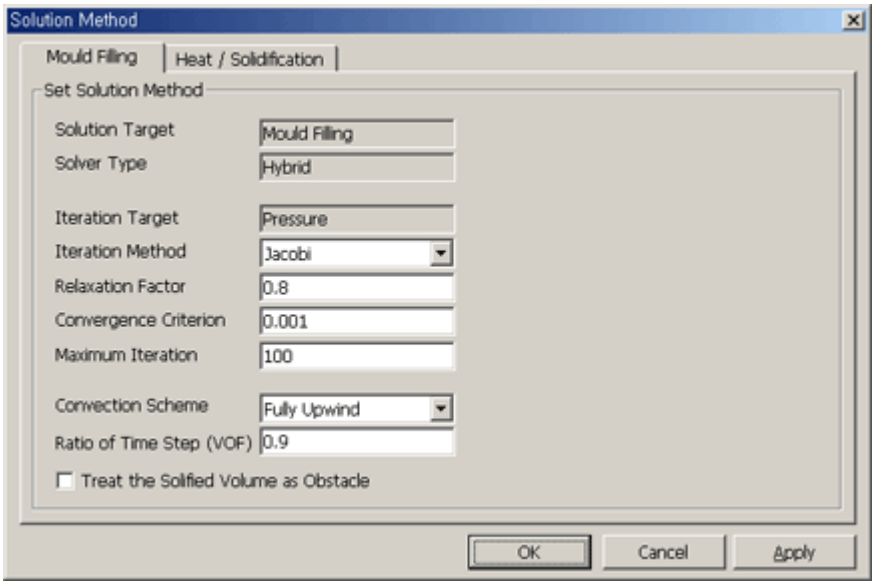
Note
When the solid fraction of one mesh is below the input value, it is considered as liquid.
The melt is removed at feeder-out time and filled with air of a constant temperature.

Setting Simulation Method

anySOLVER is composed of a solver that simulates the mould filling of the melt and a solver that simulates the heat and solidification of the melt. The organic combination of both solvers enables simulation of the entire process from melt filling to solidification.

Mould Filling

This is where the condition factors of the solver simulating the melt flow are set.



Solution Target	Indicates the target to be simulated by the solver
Solver Type	Indicates solver type (Hybrid Only)
Iteration Target	Variable that is solved by iteration method
Iteration Method	Select iteration method among Jacobi (default value), SOR
Relaxation Factor	Relaxation factor to be used during iteration simulation Jacobi (0 < Relaxation Factor < 1), default value =0.8 SOR (1< Relaxation Factor < 2), default value =1.2
Convergence Criterion	Convergence criterion ofiteration method (<0.001, default value=0.001)
Maximum Iteration	Maximum number of iterations (>50, default value =100)
Convection Scheme	Treatment of convective term of Navier-Stokes Equation Central Fully Upwind (default value)

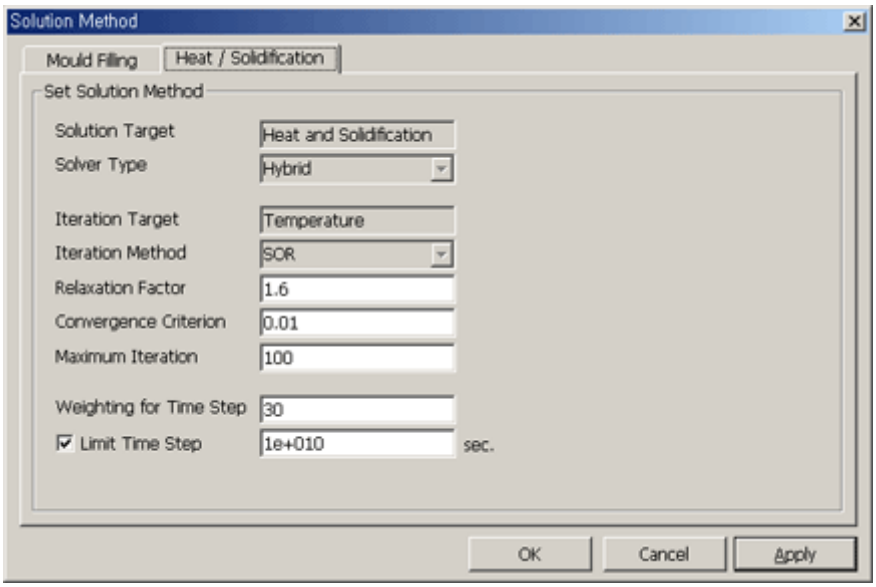
Hybrid (0<Convection Scheme<1)

Ratio for Time Step Time control factor that calculates the progress of free surface
(0 < Weight for Time Step < 0.9, default value = 0.9)

Treat the Solidified Volume as Obstacle In flow analysis, solidified volume is treated as an obstacle.

Heat / Solidification

The heat solver simulates the temperature and solidification. This is where the condition factors for the heat solver are set.



Solution Target	Indicate target to be simulated
Solver Type	Select solver type Explicit,Fully Implicit,Hybrid (0<Value<1, default value=0.5)
Iteration Target	Variable that is solved by iteration method
Iteration Method	Select from iteration. Jacobi (default value), SOR
Relaxation Factor	Relaxation factor to be used during iteration simulation SOR (1<Relaxation Factor<2), default value=1.6
Convergence Criterion	Convergence criterion of iteration method(< 0.01, default value = 0.01)
Maximum Iteration	Maximum number of iterations (> 50, default value=100)
Weighting for Time Step	Weighting for solver acceleration(default value = 30)
Limit Time Step	Limit maximum time step

Note

The following 3 equations need to be analyzed in casting simulation.
Navier-Stokes Equation or Momentum Equation
Continuity Equation
Volume of Fluid Equation

Iteration Method
For simulation of incompressible flow,pressure equations are induced by the continuity equation. The iteration method is used to simulate such pressure equations. There are two types of iteration methods as shown above: the Jacobi and SOR (Successive Over-Relaxation) methods. The Jacobi method generally requires more time till the convergence (the residual of the entire equation falls below the convergence criterion set by the user) than the SOR method. However, this method is useful in predicting the flow of melt that is symmetrical. On the other hand, the SOR method does not guarantee symmetry of results but has a faster convergence speed than the Jacobi method.

Maximum Iteration
If convergence does not occur while carrying out iteration on pressure equations, the simulation can go on to infinity. Thus, in such a case, terminate the iteration when it reaches the maximum iteration number and go onto the next time level. If the value is too small (<50), the errors that have not been converged pile up with the progress of the simulation. This possibly brings down the accuracy of the simulation. If the value is too big(>150), the iteration number increases too much and the overall simulation time takes too long. The value recommended in this program is 80~120.

Ratio for Time Step
Control the time step using the weighting value which is used for VOF equations. If the front of the melt moves more than one grid, then it becomes difficult to accurately predict the free surface. This can also result in the equations not converging. The value recommended in this program is 0.5~0.9.

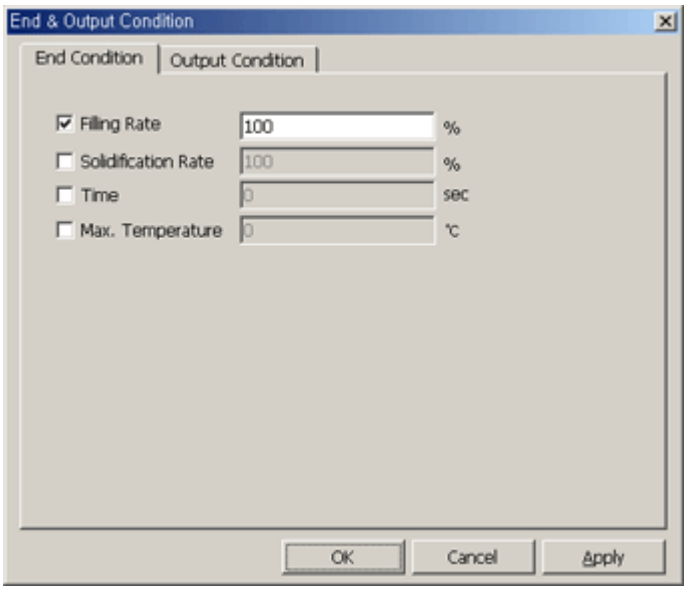
Weighting for Time Step
Usually the time step is set up by stability condition for simulation. The heat solver of anySOLVER can use more than the stable time step. This value is ratio to the stable time step

Simulation End and Result Output Conditions

This is where the simulation end time and result storage conditions are set.

End Condition

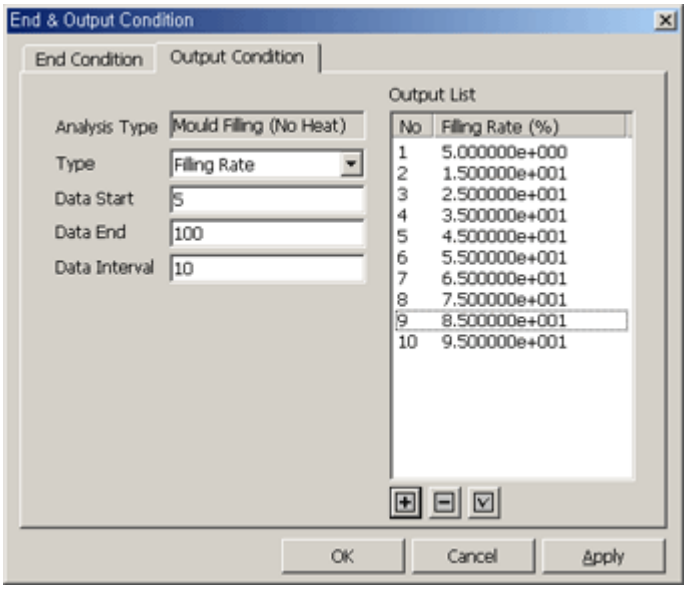
This is where the conditions to end the simulation are set. The conditions are Filling Rate, Solidification Rate, Time, and Max. Temperature. Select only one from these.



Filling Rate	Simulation ends when filling rate exceeds the input value
Solidification Fraction	Simulation ends when solidification fraction exceeds the input value
Time	Simulation ends when simulation time exceeds the input value
Max. Temperature	Simulation ends when maximum temperature of the entire domain falls below the input value

Output Condition

This is where the storage conditions of the simulation result files are set when anySOLVER is launched.



Type	Select a criterion variable for result storage(Filling Rate, Solidification Rate, Time)
Data Start	Initial storage point
Data End	Final storage point
Data Interval	Storage interval

Add/Delete/Edit Data Interval

Move to mouse to output list field and Click on the right mouse button to load the fast menu. In the fast menu, select **Add**/**Edit**/**Delete**/**Delete** or click on the below buttons.

Add data interval
Possible to add one or more

Delete data interval

Edit data interval

Note When anySOLVER saves result files in the data intervals designated in the **Output Conditions**, middle files(*.mid) are automatically created

Launching anySOLVER

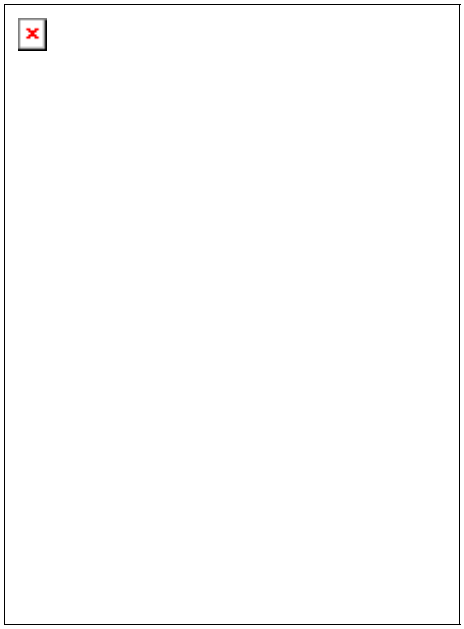
Click on **Run** in **Launching Conditions** and the following anySOLVER window will load.

Click on the activated **Start** button and the simulation will start.

Note Refer to the anySOLVER manual for specifics on the program.

Setting Preferences

This is where you preferences including unit, user database, background color, etc. are set.



Unit

Unit System	Select one from MKS, CGS, BTU
Measurement Unit	Select measurement unit (inch, mm)
Importer Unit	Set unit of STL file (inch, mm)

Database

User Database	Click [r] button and select user database file
Default Material Spec.	Select default material specification displayed when database materials are shown

Background

Color	Select background color
Gradient Background	Turn on/off background gradient

Misc.

Show Report After Building Mesh	Show report after building mesh
Max User View Point Number	Set maximum number of user view points