Die Casting Tutorial

This example tutorial goes through the steps of setting up and running a high pressure die casting analysis. The tutorial begins with a meshed geometry, assigns the process parameters and completes the solution.

The mesh used in this tutorial is a coincident mesh, in which nodes are shared between regions or materials. There is a slight difference in the setup if a non-coincident mesh (nodes are not shared between regions) is used or if an optimized coincident mesh is used. Whenever differences occur in the setup because of the use of these other meshes, a message concerning these alternative methods will be noted in the Extra Notes section.

Die Casting Analysis Process

When the die casting process is simulated using ProCAST, two simulations are performed. The first simulation looks only at the thermal effects of the process. The aim is to simply heat the dies up to normal operating temperature. In most cases, the typical casting is not produced before a few cycles are run and the dies have allowed some heat to sink in. In order to decrease the amount of time spent on these cycles, the filling is not performed and an instantaneous fill is assumed.

Once the dies have performed an adequate amount of cycles to reach operating temperature, a completely coupled analysis is performed including thermal, fluid, stress, and/or micromodeling effects. This tutorial will demonstrate a thermal / fluid analysis.

Die Casting Cycle Timeline

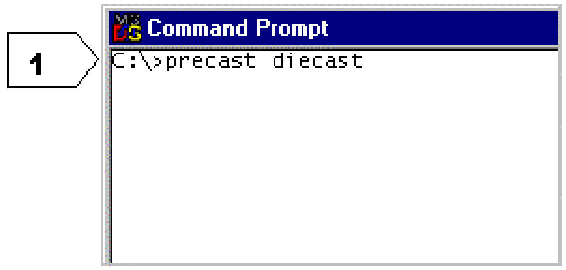
The following sample cycle will be used in this tutorial:

Time (seconds)	Event
0	Begin Filling
25	Open Dies
30	Eject Casting
35	Begin Spray
40	End Spray
50	Close Dies / Begin Next Cycle

Start PreCAST

To begin PreCAST, type the command "precast" followed by the project name inside either a Unix window, or MS-DOS prompt. For this tutorial, use the project name "diecast".

There are other methods for starting any of the ProCAST applications. These are described in Section 1 of the ProCAST manual.



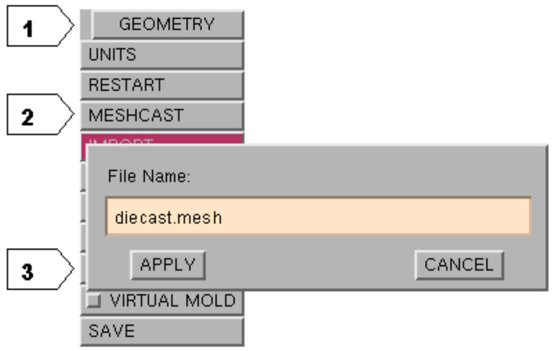
---

1. precast diecast

Extra Notes

If PreCAST fails to appear on your screen, there has been a problem with the installation. Please check the installation instructions, or contact your system administrator or ProCAST customer support representative.

Importing the Mesh



If you are doing a 3-D analysis, a Finite Element mesh needs to be imported into PreCAST. This mesh can be generated and written out in any of the formats listed in the Geometry submenu. The mesh used for this tutorial was generated in ProCAST's own automatic tetrahedral mesher, MeshCAST.

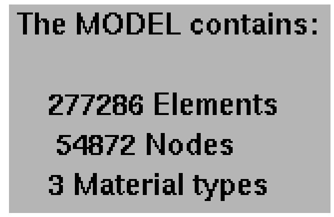
A typical MeshCAST file has the filename "prefix.mesh".

1. **GEOMETRY**
2. **MESHCAST**
3. **APPLY**

Extra Notes

If you are using a mesh generated by MeshCAST, the units of the model are part of the MeshCAST file. Therefore, the units do not have to be set when reading in a MeshCAST mesh. However, any other type of format needs to have the units specified. This is accomplished by selecting a unit in the **Geometry..Units** submenu.

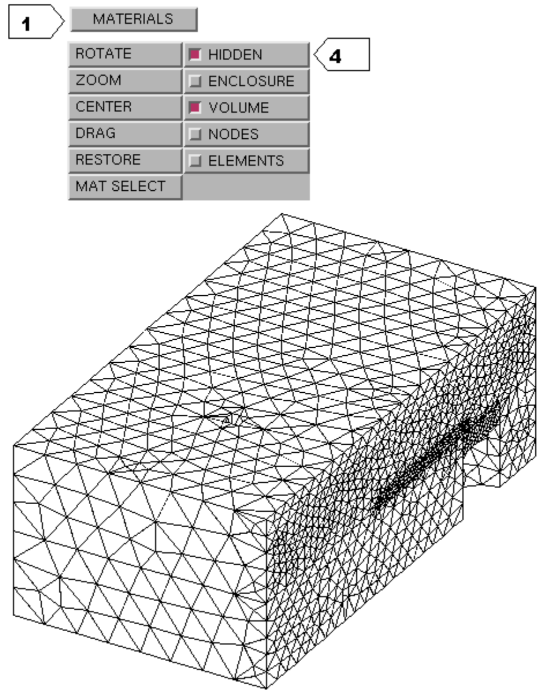
When the mesh is completely read in, you will see a mesh information form like the one shown to the right. This form helps to confirm that the correct number of entities (nodes, elements) are imported.



There are no steps on this page.

Extra Notes

To see the geometry, one simply needs to press any of the other buttons on the top row of the PreCAST window. The control of how the model is viewed is done by hot keys, mouse functions and/or buttons and forms. These controls are described in more detail in Section 1 of the ProCAST Users' Manual.



---

1. MATERIALS

2. Press SHIFT+X.

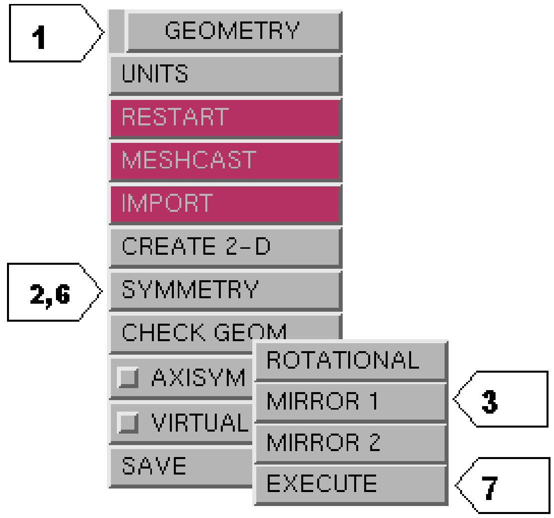
3. Press CTRL+SHIFT+Y

4. HIDDEN

Extra Notes

For information on any of the viewing related options shown on this page, please refer to Section 1 of the ProCAST User's Manual.

Defining Symmetry



In order to define the planes of symmetry of this model, you will list three points that lie on this plane. These points can be retrieved in a variety of ways using the mesher or solid modeler. These points were determined by locating a node on each plane and noting the planar coordinate of the node.

The symmetry condition forces two things:

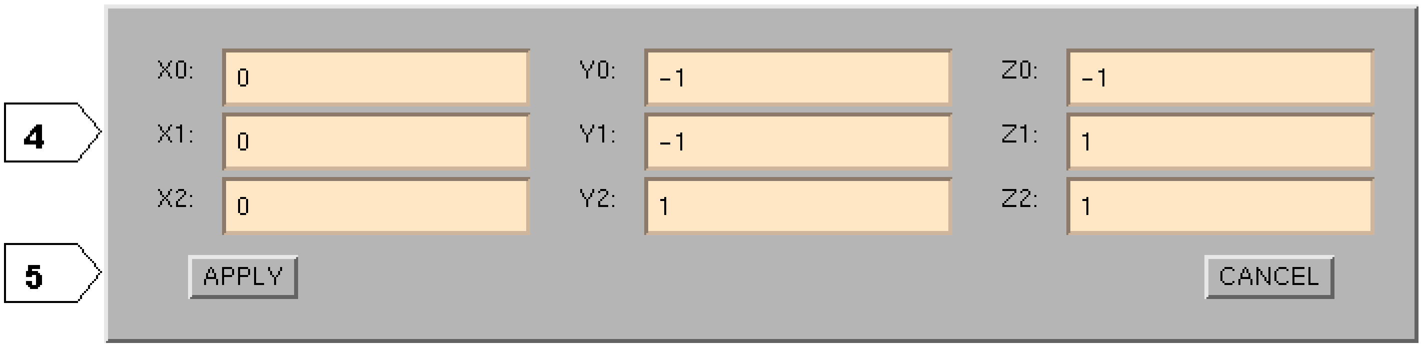
- No heat transfer across the symmetry face.
- No fluid flow across the symmetry face.

Also, a symmetry boundary condition is automatically generated, as you will see in the boundary condition assignment section of this tutorial.

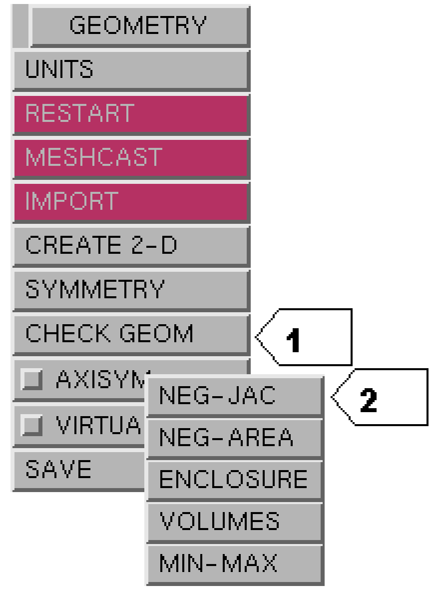
1. **GEOMETRY**
2. **SYMMETRY**
3. **MIRROR 1**
4. Enter the coordinates of a plane at X = 0 (see next page)
5. **APPLY** (see next page)
6. **SYMMETRY**
7. **EXECUTE**

Extra Notes

As can be seen on the screen, only one-half of this casting is being shown. This is done to make use of the symmetry in the model. Instead of meshing and modeling the entire model, the symmetry inherent in the model allows us to only simulate a smaller part. There is no loss of accuracy and the solution time will be greatly decreased.


4
5
X0:
0
Y0:
-1
Z0:
-1
X1:
0
Y1:
-1
Z1:
1
X2:
0
Y2:
1
Z2:
1
APPLY
CANCEL

Checking the Imported Mesh



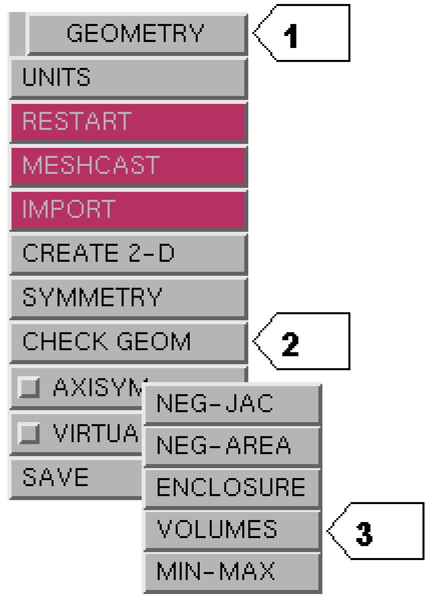
For any mesh that is imported into PreCAST, it is wise to check the integrity of the mesh. Should the mesh have bad elements in it, there is a good chance that the simulation will give poor results.

1. **CHECK GEOM**
2. **NEG-JAC**

Extra Notes

On this page we are looking for Negative Jacobian (Neg-Jac) elements. These are elements which have been turned inside out or are flat. If there are any of these in the mesh, there is a good chance that the simulation will stop due to solver convergence problems.

Here we will check the volume of the different parts of this model. The volume check lets us make sure that the units are correct and gives us information that we can use when we apply the condition that will fill up the cavity with metal.



1. **GEOMETRY**
2. **CHECK GEOM**
3. **VOLUMES**

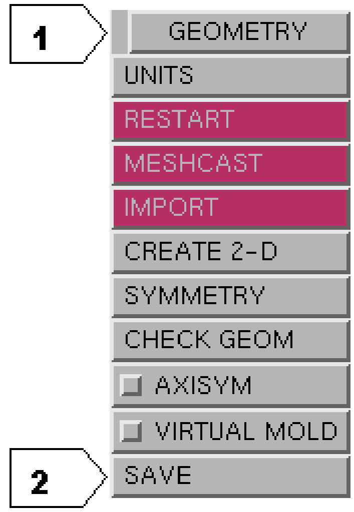
Extra Notes

The unit of the volume is the same as the "working units" of the setup. The "working unit" is determined in one of two ways; either by setting it directly in **GEOMETRY..UNITS**, or by reading it from the file, as we have in this case. The file we have read in used millimeters (you can see this if you press **GEOMETRY..UNITS**). Therefore, the volume given is in cubic millimeters.

Saving the PreCAST Setup

You can save at any point when setting up the simulation in PreCAST. Simply press the **SAVE** button in the **GEOMETRY** submenu.

It is a good idea to save from time to time to prevent losing data due to power failure, incorrect setup or file corruption.



---

1. **GEOMETRY**
2. **SAVE**

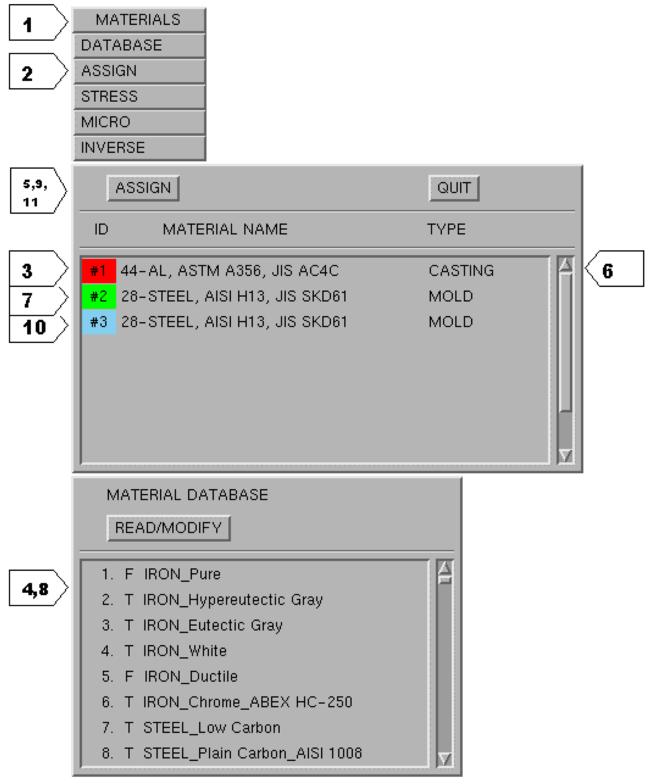
Extra Notes

When you press the **SAVE** button in PreCAST, you are saving all of the data that will be used for the simulation to a file named [prefix]d.dat (or diecastd.dat in this case). This data includes the geometry data, materials being used, process data – anything that is set up or used in PreCAST. A second file will also be written when you save and exit PreCAST, with the file name [prefix]p.dat. This file only contains the Run Parameters, which will be described in more detail at the end of this example.

Assigning Materials

Now we will define the materials being used in this simulation. For this example, Aluminum 356 (Al356) will be assigned as the metal to be cast, and Steel H-13 will be assigned as the die material.

On the Material Assignment form, you will see three columns. The ID is the material identifier for a group of elements. The Material Name is the material assigned to the group of elements that have the same ID. The Type specifies the function of that piece of geometry. This Type designation allows ProCAST to do a certain set of calculations or make assumptions about the model.



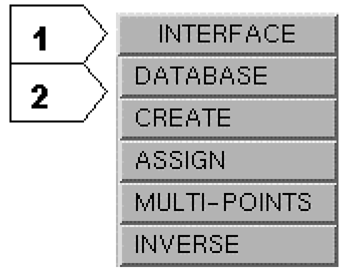
---

1. **MATERIALS**
2. **ASSIGN**
3. Click on #1
4. Highlight the material Al356 in the Material Database
5. **ASSIGN**
6. Toggle Type to “Casting”
7. Click on #2
8. Highlight the material H-13 in the Material Database
9. **ASSIGN**
10. Click on #3
11. **ASSIGN**

Extra Notes

This example uses materials that already exist in the material database. If you would like to create your own material, please refer to the Materials Section of the ProCAST Users’ manual, beginning on page 3-46.

Create the Interface Conditions



When heat is transferred across dissimilar materials, it will probably not pass through as if there is perfect contact. Therefore, an interface heat transfer coefficient needs to be assigned to the region at the interface of the materials. In this section, we will add new interface heat transfer coefficients to the database, which will later be assigned to the interface of the materials.

In all of the databases (Material, Interface, Stress, Boundary Condition, etc.), an individual database entry only needs to be entered once. After it is entered, it is available for assignment in all other simulations from that point on.

---

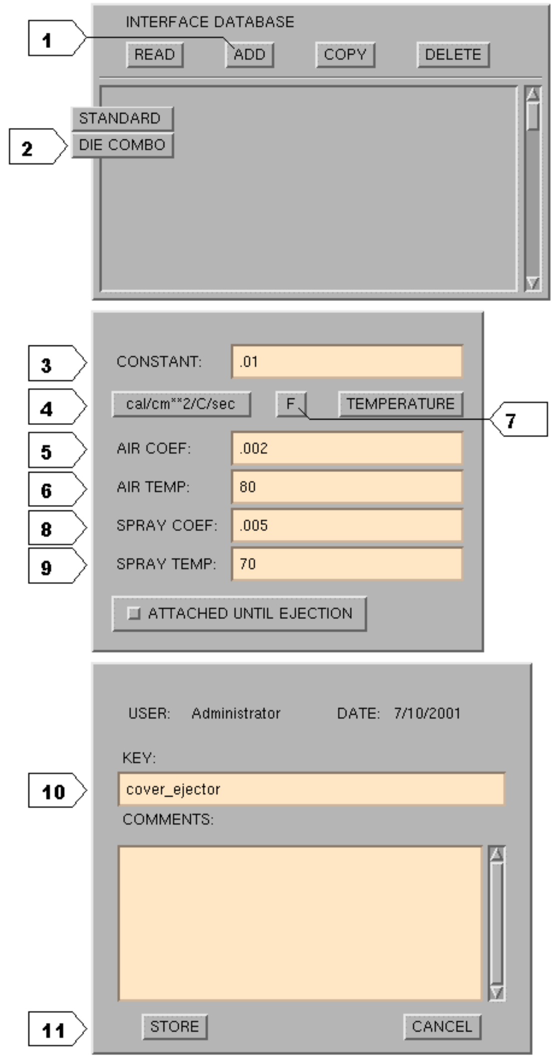
1. **INTERFACE**
2. **DATABASE**

Extra Notes

As you can see in the forms at the right, when you define a new database entry (interface heat transfer coefficient in this case), you have options for defining the data. In this instance, you can enter the heat transfer coefficient as a constant value, time function or temperature function.

A constant value stays the same for the duration of the simulation. A time-dependent function changes the heat transfer coefficient over time. A temperature-dependent function varies the heat transfer coefficient as the temperature of a certain material changes.

A Die Combo interface description is a special method used for describing all the heat transfer properties at the interface during the cycle. This entry describes the heat transfer between the two die halves. The Constant value is the interface heat transfer coefficient between the dies. This may also be temperature dependent by entering data in the Temperature table. The Air Coefficient and Air Temperature is the heat transfer to the air and ambient air temperature around the die when the dies are open. The Spray Coefficient and Spray Temperature describe the heat transfer and ambient spray temperature when the spray occurs during the cycle.



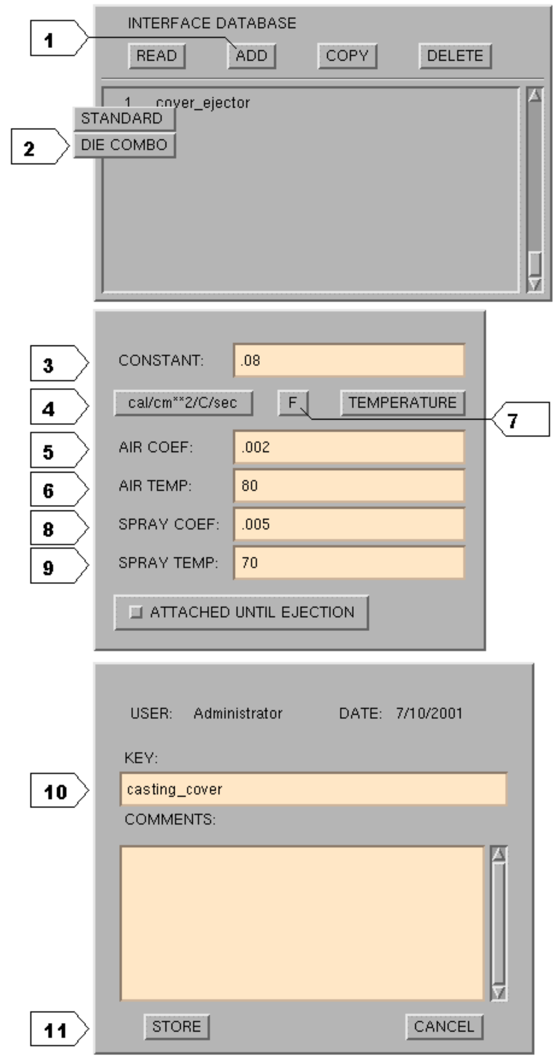
1. **ADD**
2. **DIE COMBO**
3. Enter .01 for Constant
4. Change the units to cal/cm^2/C/sec
5. Enter .002 for Air Coefficient
6. Enter 80 for Air Temperature
7. Toggle Unit to “F”
8. Enter .005 for Spray Coefficient
9. Enter 70 for Spray Temperature
10. Enter “cover_ejector” as the Key
11. **STORE**

Extra Notes

Die Combo may only be used with coincident or shared node meshes. Otherwise, a Standard Interface must be employed using time dependent properties based on the cycle times.

Die Combo combines interface and boundary condition properties together into a single form and assignment. Therefore, the only Boundary Conditions are ones not covered by these Die Combo assignments, such as the heat transfer on the outside of the inserts and cooling lines.

The next interface to be described is between the casting and the cover die. Note the higher rate of heat transfer compared to that between the dies themselves.



---

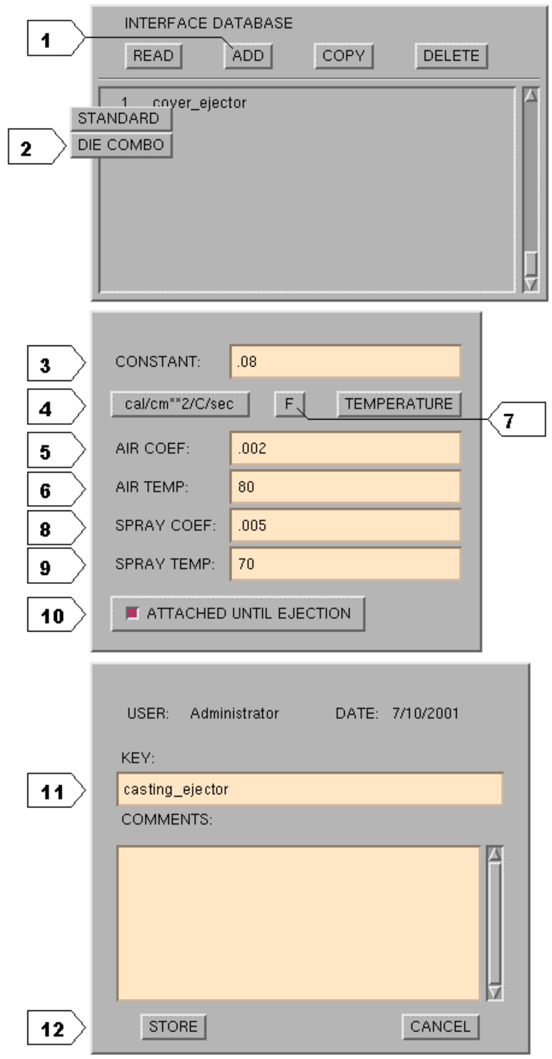
1. **ADD**
2. **DIE COMBO**
3. Enter .08 for the Constant
4. Toggle the Unit to "cal/cm**2/C/sec"
5. Enter .002 for Air Coefficient
6. Enter 80 for Air Temperature
7. Toggle the Unit to "F"
8. Enter .005 for Spray Coefficient
9. Enter 70 for Spray Temperature
10. Enter "casting_cover" as the Key
11. **STORE**

Extra Notes

Notice that on the form where you enter the name of the database entry (the key), there is a listing for the user. In all of PreCAST's databases (Materials, Interface, Boundary, etc.), a specific user owns each database entry. This ownership prevents other users from modifying entries. Any user can make use of an entry not owned by that user, but if there is a need to modify the entry, the entry would first need to be copied to a new name. This new entry would be created an owned by the user who created the entry.

The final interface to be described is between the casting and the ejector die. Since the casting will sit in the ejector until ejection time, specify that the casting is “Attached Until Ejection”.

The interface heat transfer coefficient turns off and switches to the air / spray values at the specified Open time. However, if Attached Until Ejection is specified, this switch is delayed until the Ejection time. These times are entered in the **Run Parameters > Cycles** section of PreCAST, which will be completed later in this tutorial.



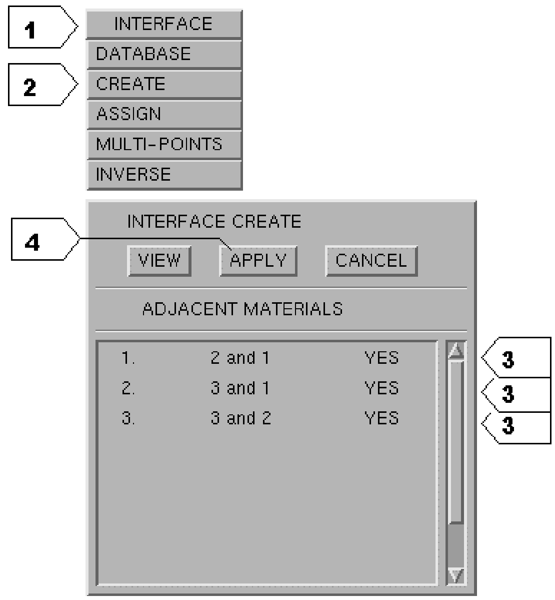
---

1. **ADD**
2. **DIE COMBO**
3. Enter .08 for the Constant
4. Toggle the Unit to “cal/cm**2/C/sec”
5. Enter .002 for Air Coefficient
6. Enter 80 for Air Temperature
7. Toggle the Unit to “F”
8. Enter .005 for Spray Coefficient
9. Enter 70 for Spray Temperature
10. Toggle on **ATTACHED UNTIL EJECTION**
11. Enter “casting_ejector” as the Key
12. **STORE**

Extra Notes

If the Die Combo interface is being used for a permanent mold, where spray does not occur at every cycle, simply leave the Spray Coef and Spray Temp spaces empty.

<u>Create the Interface in the FE Model</u>



The model used in this tutorial is an example of a typical mesh generated by a meshing package. In this coincident mesh, both materials share nodes at the interface.

1. **INTERFACE**
2. **CREATE**
3. Toggle "No" to "Yes" for all three material pairs
4. **APPLY**

Extra Notes

Non-Coincident Mesh:
The **CREATE** button will be highlighted red and is inaccessible as PreCAST cannot determine the material pairs in this type of mesh. Skip this page and go on to the next page, Interface Assignments.

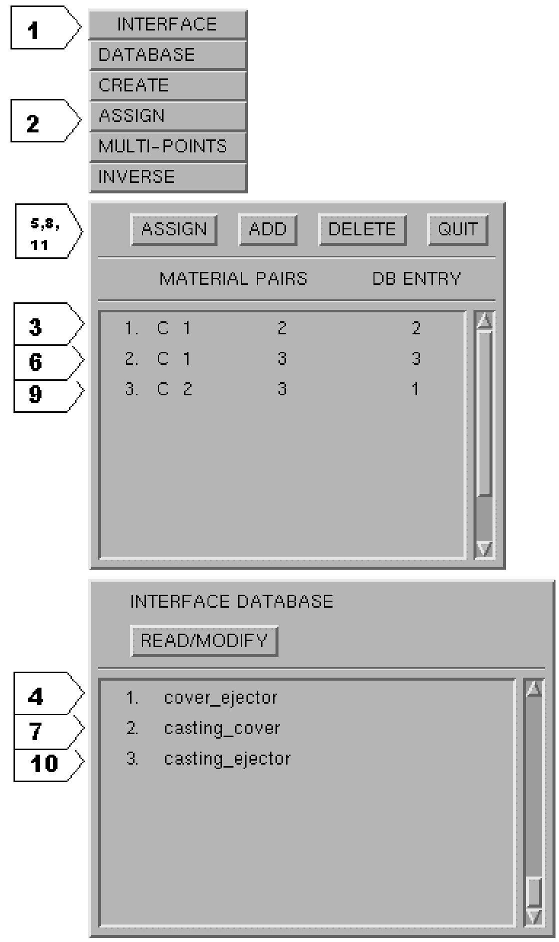
Optimized Coincident Mesh:
When the optimization is performed, the material pairs are already identified. Leave all material pairs marked as NO, press **EXECUTE** and continue on to the next page.

Assigning the Interface Condition

Now that an interface coefficient has been added to the database and an interface has been created, we need to assign that database entry to the interface in this model.

You will notice this "Define then Assign" trend as you go through PreCAST. Most items are first defined by some value like material properties, heat transfer coefficient, or velocity, then they are assigned to some part of the Finite Element Model you have imported.

A "C" in the second column signifies a coincident interface, whereas a "N" specifies a non-coincident mesh interface.



---

1. **INTERFACE**
2. **ASSIGN**
3. Click on the first material ID in material pair 1
4. Highlight the "casting_cover" entry in the database.
5. **ASSIGN**
6. Click on the first material ID in material pair 2
7. Highlight the "casting_ejector" entry in the database.
8. **ASSIGN**
9. Click on the first material ID in material pair 3
10. Highlight the "cover_ejector" entry in the database.
11. **ASSIGN**

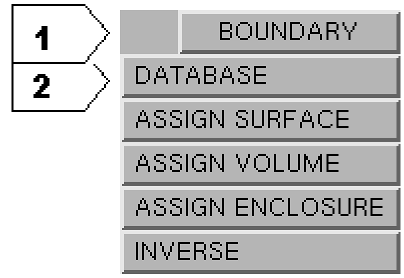
Extra Notes

The order of the materials is important only when you are using a temperature dependent interface heat transfer coefficient. In this case, it is necessary to identify which material to base the calculation of heat transfer. The first material in the pair will be used as this basis. To switch the order, simply click on the second material in the pair.

Non-Coincident Mesh:
Before assigning database entries, you must add the interface pairs. To do this, press ADD, enter the material pair separated by a space (ie, 1 2) and press APPLY. Do this for all materials between which you want to specify an interface.

Creating Boundary Condition Properties

Boundary Conditions enforce the presence of certain process items such as pouring temperature, filling rate, heat transfer and many other items. In this thermal-only simulation we have three parameters to define: heat convection from the dies and cooling line effects.

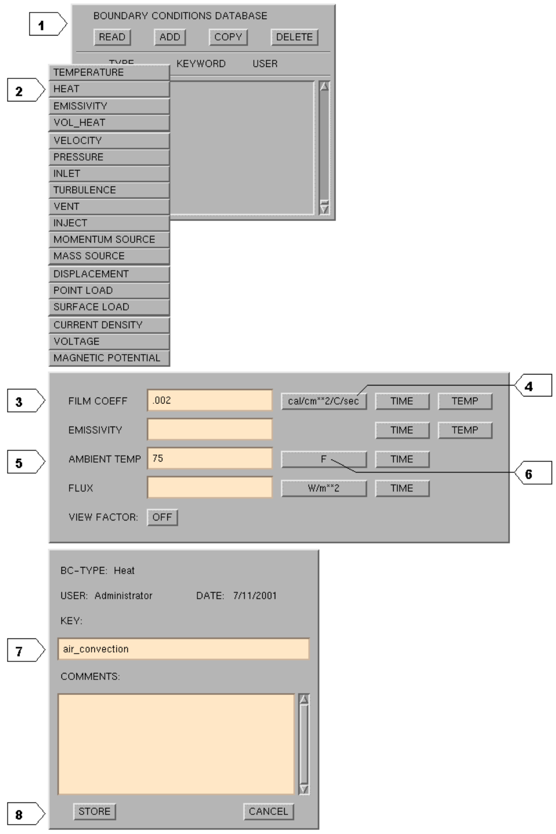


---

1. **BOUNDARY**
2. **DATABASE**

Extra Notes

First, we will define the heat transfer on the outside of the dies. To reduce the size of the model, we typically analyze only the die inserts and not the holders. Modeling the entire system is easily done; however, the size of the model would significantly increase the simulation time and would not offer much more accuracy.

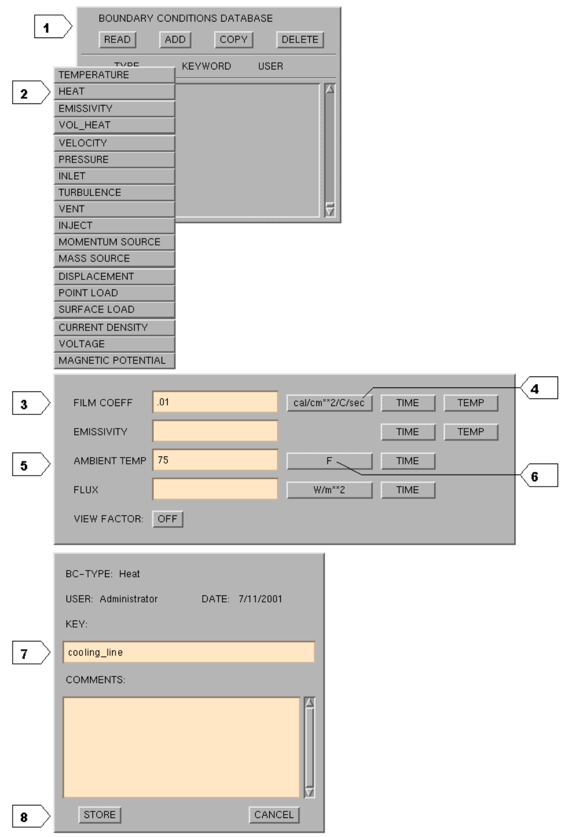


1. **ADD**
2. **HEAT**
3. Enter .002
4. Change the units to cal/cm^2/C/sec
5. Enter 75
6. Change the units to F
7. Enter “air_convection”
8. **STORE**

Extra Notes

Heat transfer can be done in two ways, either by interface heat transfer or by heat boundary condition. The main difference is that heat transfer occurring between items that have been modeled (i.e., die and casting) is simulated using an interface condition. Heat transfer occurring between a modeled item and an unmodeled item must be simulated using a heat boundary condition.

Finally, we will define the heat transfer of the cooling lines. We will assume that the cooling line is always on. You could also simulate cooling and/or oil lines using a heat flux or even meshing the lines and performing a fluid flow analysis of the coolant or oil.



1. **ADD**
2. **HEAT**
3. Enter the .01
4. Change the units to cal/cm^2/C/sec
5. Enter 75
6. Change the units to F
7. Enter "cooling_line"
8. **STORE**

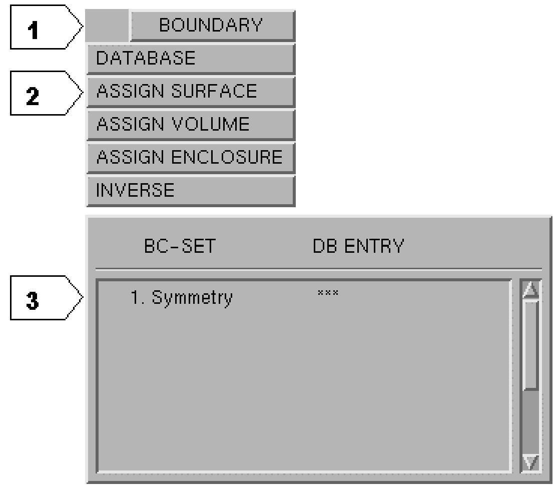
Extra Notes

If the cooling line is not constantly on, as shown in this example, you can turn it on and off by using the **TIME** feature. This feature allows you to adjust the heat transfer coefficient in relation to simulation time.

Assign the Boundary Conditions

Now that the boundary conditions have been defined in the boundary database, we can assign these definitions to locations on the model. We will first check the symmetry boundary condition that was automatically generated when the symmetry planes were defined, then continue with the conditions that were defined in the previous section.

When the symmetry boundary condition is selected, the element faces on the two planes of symmetry should highlight in red.



1. **BOUNDARY**
2. **ASSIGN SURFACE**
3. Highlight the Symmetry boundary condition

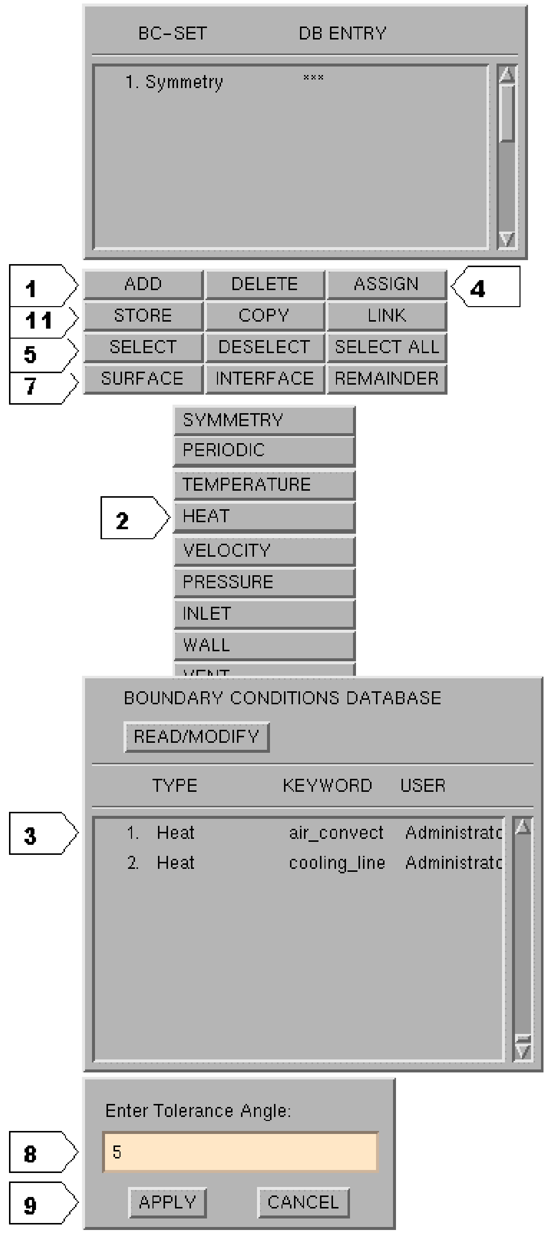
Extra Notes

If the selected faces automatically assigned to the Symmetry boundary condition are not correct, you can simply add or remove the necessary elements using **SELECT** and **DESELECT**.

Assign the heat boundary condition of the outside of the dies.

Using the **SURFACE** button makes selecting a large area very easy. After selecting an entity and pressing **SURFACE**, you enter an angular tolerance. All neighboring entities within this tolerance will then be selected. Thus, when selecting just a plane, use a small tolerance. But when the selection needs to go across a varying terrain of geometry, use a larger angle.

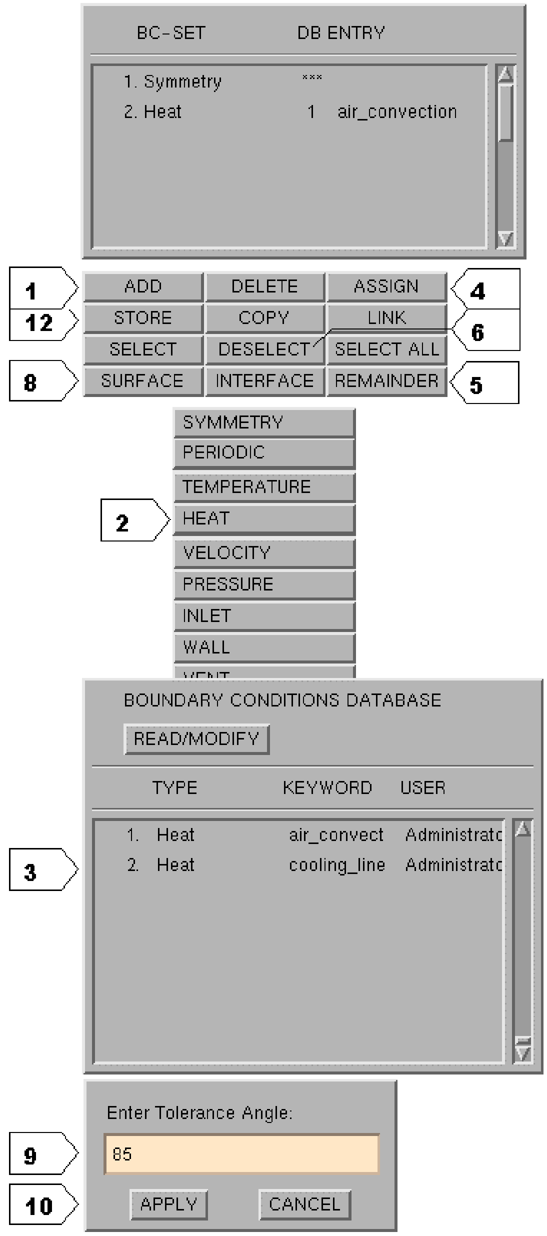
You can also select multiple entities by dragging a window with the right mouse button.



1. **ADD**
2. **HEAT**
3. Highlight the "air_convection" Heat database entry
4. **ASSIGN**
5. **SELECT**
6. Select an element on the outside of the die
7. **SURFACE**
8. Enter "5"
9. **APPLY**
10. Repeat Steps 5 – 9 for the other outer die surfaces
11. **STORE**

Extra Notes

Finally, assign the heat boundary condition to the cooling lines. Cooling lines and other features that go into a volume can be difficult to select. To ease selection, we make use of the fact that when working with heat boundary conditions, an element can be assigned to only one heat condition. Therefore, it is very easy to know which elements have not been selected for a heat condition. In this example we have the symmetry face selected, the outer faces selected and the interface elements selected. This leaves the cooling lines and the shot cylinder as the only faces left unselected. The **REMAINDER** button picks all of these with one click. All that needs to be done is to deselect the shot cylinder and biscuit inlet.

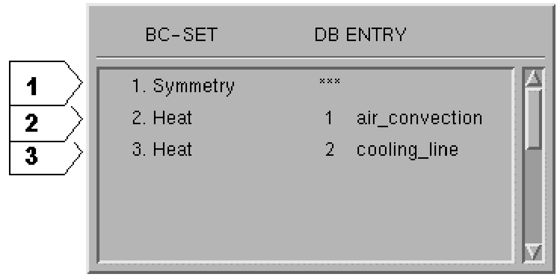


1. **ADD**
2. **HEAT**
3. Highlight the "cooling_line" Heat database entry
4. **ASSIGN**
5. **REMAINDER**
6. **DESELECT**
7. Deselect an element on the shot cylinder
8. **SURFACE**
9. Enter "85"
10. **APPLY**
11. Repeat for biscuit inlet surface.
12. **STORE**

Extra Notes

When setting up this model, you may notice that the shot cylinder and biscuit will have no heat transfer. In practice, this surface would be in contact with the shot sleeve. A heat boundary condition could have been placed in this location to emulate the heat transfer between these two regions, but was not added in order to keep this example simple.

To make sure that the correct entities have been assigned to the correct boundary condition, click on each boundary condition in the bc-set column. When a boundary condition is selected, the entities assigned to that condition highlight.

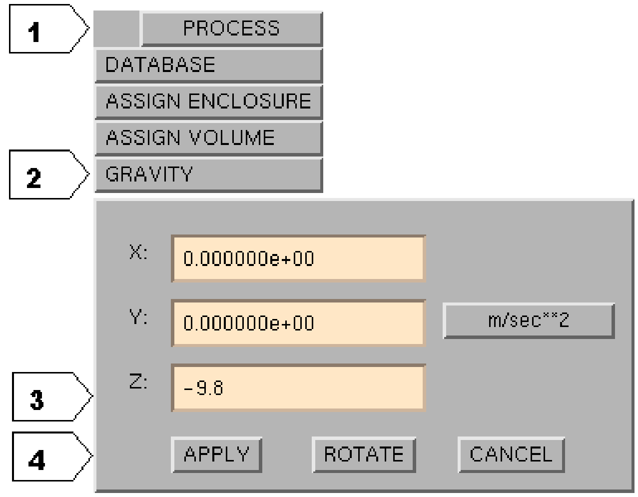


1. Click on the “Symmetry” BC
2. Click on the first “Heat” BC
3. Click on the second “Heat” BC

Extra Notes

Define Gravity Direction

For the definition of the gravity direction, we simply enter the value into the appropriate direction, the negative-Z direction in this case.



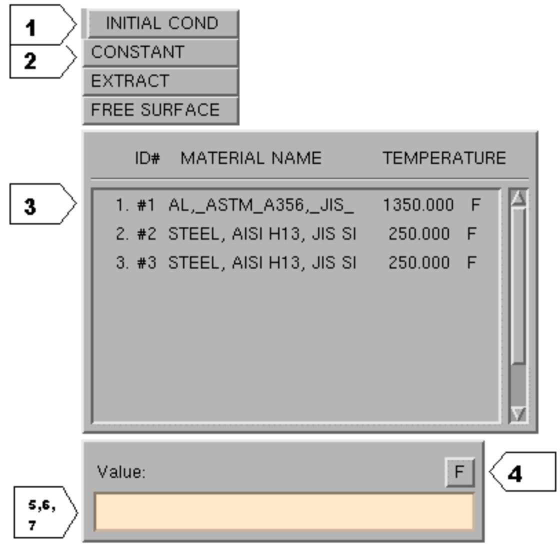
1. **GRAVITY**
2. Enter –9.8 as the Z value.
3. **APPLY**

Extra Notes

For a gravity direction that is off of an axis, simply resolve the gravity vector into its X, Y and Z components and enter those values into the X, Y and Z value boxes.

Initial Condition Definition

The last process parameter we need to define for this thermal-only analysis is the initial condition of the simulation. This consists of the initial temperatures, which are specified in the **CONSTANT** form. The initial temperature designates the pre-heat temperature of the mold and pouring temperature.



---

1. **INITIAL COND**
2. **CONSTANT**
3. Highlight Material 1
4. Change the Units to F
5. Enter 1350 (pressing "Enter" moves the highlight bar to the next material)
6. Enter 250
7. Enter 250

Extra Notes

Modify the Run Parameters

The run parameters provide control over the simulation. In these forms you can change the time to be simulated, which solver to use and many other items. In most cases, the default choices will provide the best simulation; but, as you become more experienced with the software, you may wish to change some of these and observe the effects they have on the simulation.

In the General form, we control basic simulation characteristics. In this example, we are changing the number of steps to run to 2000.

---

1. **RUN PARAMETERS**
2. **GENERAL**
3. Change the value of NSTEP to 2000
4. **APPLY**

See next page for illustrations.

Extra Notes

You will notice that some of the run parameters are listed in black while others are listed in red. The black parameters are ones that are typically changed for a simulation. Red parameters are "advanced" parameters and should only be modified by experienced users.

Anytime help is needed for these parameters, simply click into the value related to the parameter in question and press the help button.


1
2
3
4
RUN PARAMETERS
UNITS
GENERAL
CYCLES
THERMAL
RADIATION
FLOW
TURBULENCE
STRESS
ELECTROMAGNETICS
INVERSE
CAFE
INILEV 0
NSTEP 2000
NRSTAR 5
NPRFR 1
PRNLEV 0
SDEBUG 1
AVEPROP 0
CGSQ 0
LUFAC 1
DIAG 16384
NEWTONR 0
USER 0
DT 1.00000e-03 sec
DTMAX 5.00000e+00 sec
TFINAL 0.00000e+00 sec
TMODS 2.00000e+00
TMODR 5.00000e-01
CONVTOL 1.00000e-04
HELP
APPLY
CANCEL

Now we will modify the parameters which define the cycle. The cycle parameters for this analysis may be found at the beginning of this document.

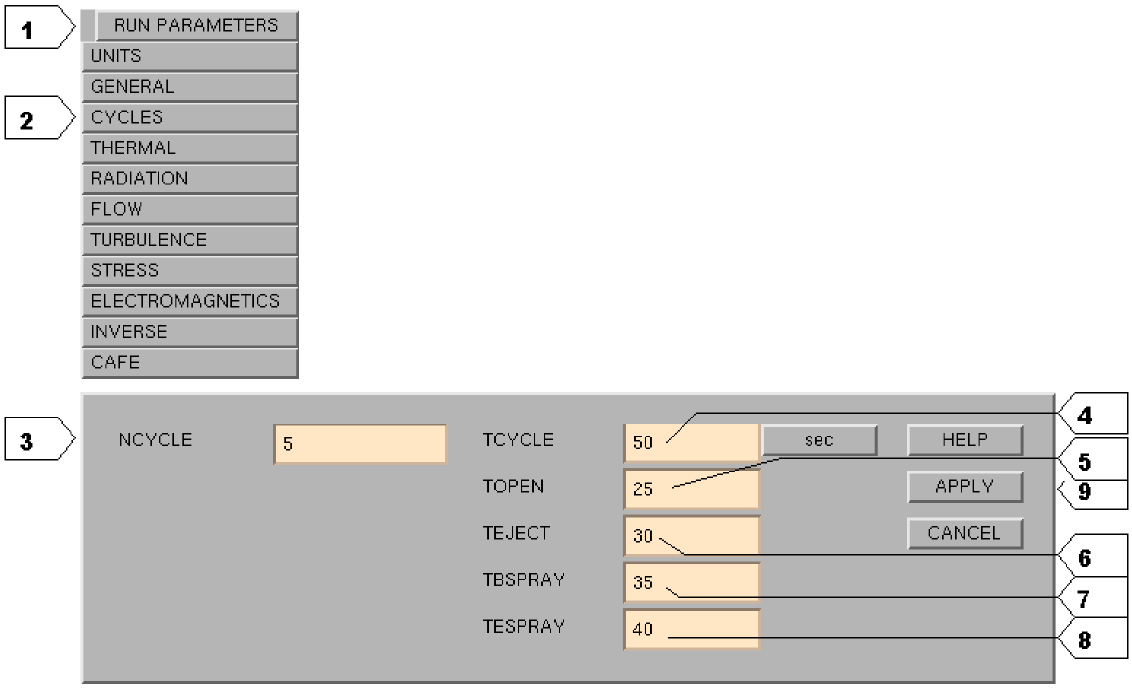
1. **RUN PARAMETERS**
2. **CYCLES**
3. Change the value of NCYCLE to 5
4. Change the value of TCYCLE to 50
5. Change the value of TOPEN to 25
6. Change the value of TEJECT to 30
7. Change the value of TBSPRAY to 35
8. Change the value of TESPRAY to 40
9. **APPLY**

See next page for illustrations.

Extra Notes

By defining these parameters, we are also specifying when the simulation will stop. The simulation will normally stop when any of the following happen:

- Number of simulated steps equals the value for NSTEP in the General Run Parameters form.
- Simulated time meets the value for TFINAL in the General Run Parameters form.
- All specified cycles have been completed as indicated by NCYCLE in the Cycles Run Parameter form.


1
2
3
RUN PARAMETERS
UNITS
GENERAL
CYCLES
THERMAL
RADIATION
FLOW
TURBULENCE
STRESS
ELECTROMAGNETICS
INVERSE
CAFE
NCYCLE
5
TCYCLE
50
sec
HELP
TOPEN
25
APPLY
TEJECT
30
CANCEL
TBSPRAY
35
TESPRAY
40
4
5
9
6
7
8

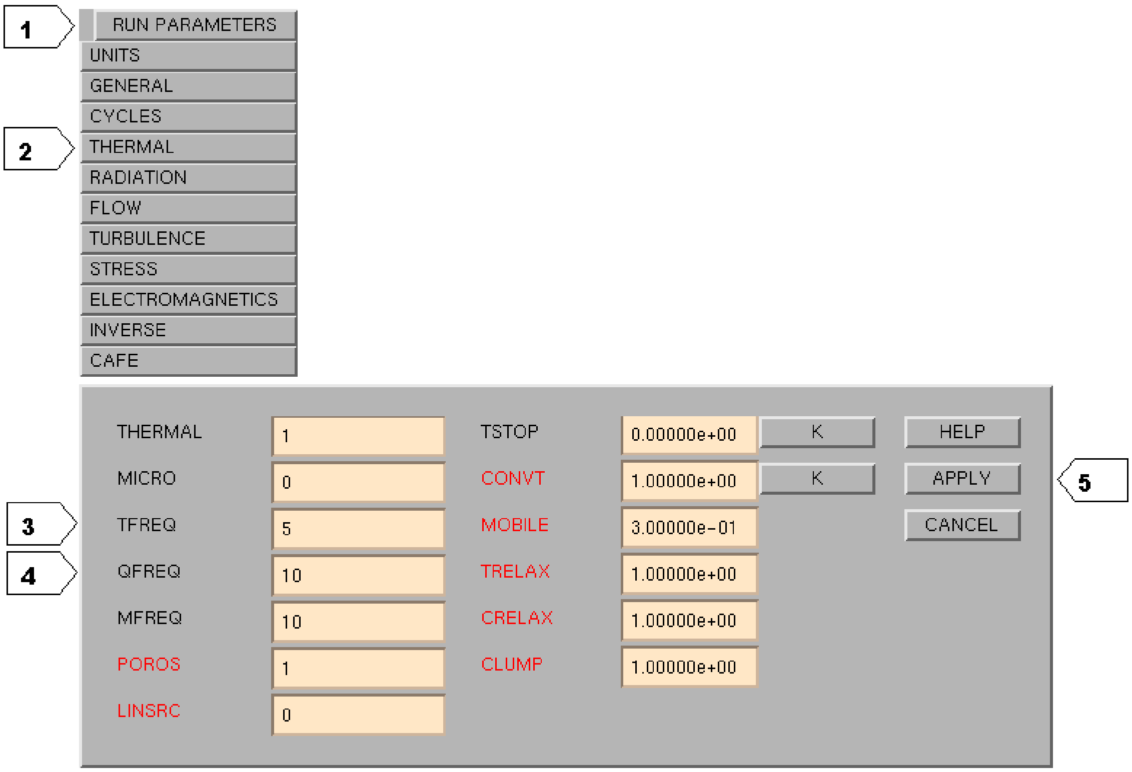
Now we will modify the parameters which control the thermal part of the simulation. Here, we are simply changing the output frequency of the temperatures to save every 5 steps, and the output frequency of heat flux to save every 10 steps.

---

1. **RUN PARAMETERS**
2. **THERMAL**
3. Change the value of TFREQ to 5
4. Change the value of QFREQ to 10
5. **APPLY**

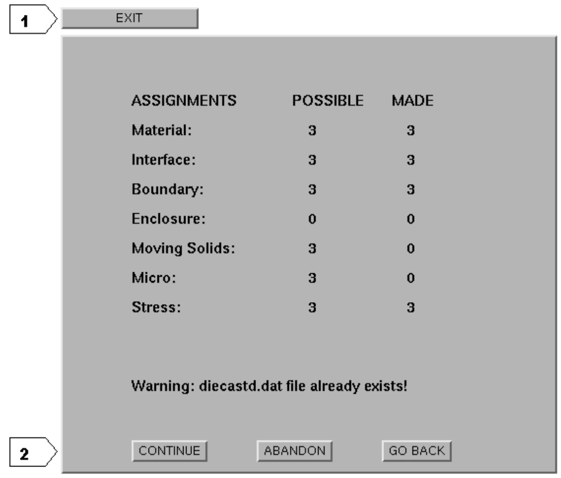
See next page for illustrations.

Extra Notes


1
RUN PARAMETERS
UNITS
GENERAL
CYCLES
2
THERMAL
RADIATION
FLOW
TURBULENCE
STRESS
ELECTROMAGNETICS
INVERSE
CAFE
THERMAL 1
MICRO 0
3
TFREQ 5
4
QFREQ 10
MFREQ 10
POROS 1
LINSRC 0
TSTOP 0.00000e+00 K
CONVT 1.00000e+00 K
MOBILE 3.00000e-01
TRELAX 1.00000e+00
CRELAX 1.00000e+00
CLUMP 1.00000e+00
HELP
APPLY
CANCEL
5

Save and Exit PreCAST

Everything should now be setup for this high pressure die casting simulation. After pressing **EXIT**, a summary screen appears showing the number of possible assignments and the number of assignments made. For this solution, the numbers for Materials, Interface and Boundary should be same in each column. Should there be any difference, **GO BACK** to the section in question and check your setup. If the numbers are matching, then **CONTINUE**.



---

1. **EXIT**
2. **CONTINUE**

Extra Notes

When you press the **EXIT..CONTINUE** button in PreCAST, you are saving all of the data that will be used for the simulation to a file named [prefix]d.dat (or investmentd.dat in this case). This data includes the geometry data, materials being used, process data – anything that is set up or used in PreCAST. A second file is also written, with the file name [prefix]p.dat. This file only contains the Run Parameters, which will be described in more detail at the end of this example.

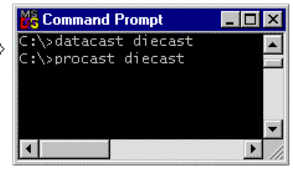
Pressing **ABANDON** simply exits without saving. Use this when browsing the set up or when changes made to the set up are not wanted.

Start Solving the Analysis

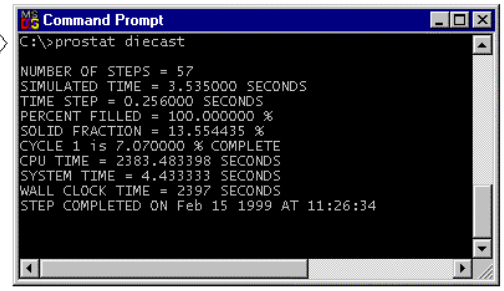
The next step in getting results is to run DataCAST. This non-interactive script simply converts the units to a consistent system (CGS) and does some simple error checking. Should any problems exist, warning or error messages will be shown on the screen.

If DataCAST runs without warning or error messages, then the setup is ready to be solved using the ProCAST executable. While the simulation is running, you can check on the progress by running the ProSTAT script in another window. This summary lists the current simulated time, time step, solve time and other statistics. This solution is complete when cycle 5 is 100% complete.

1,2



3



---

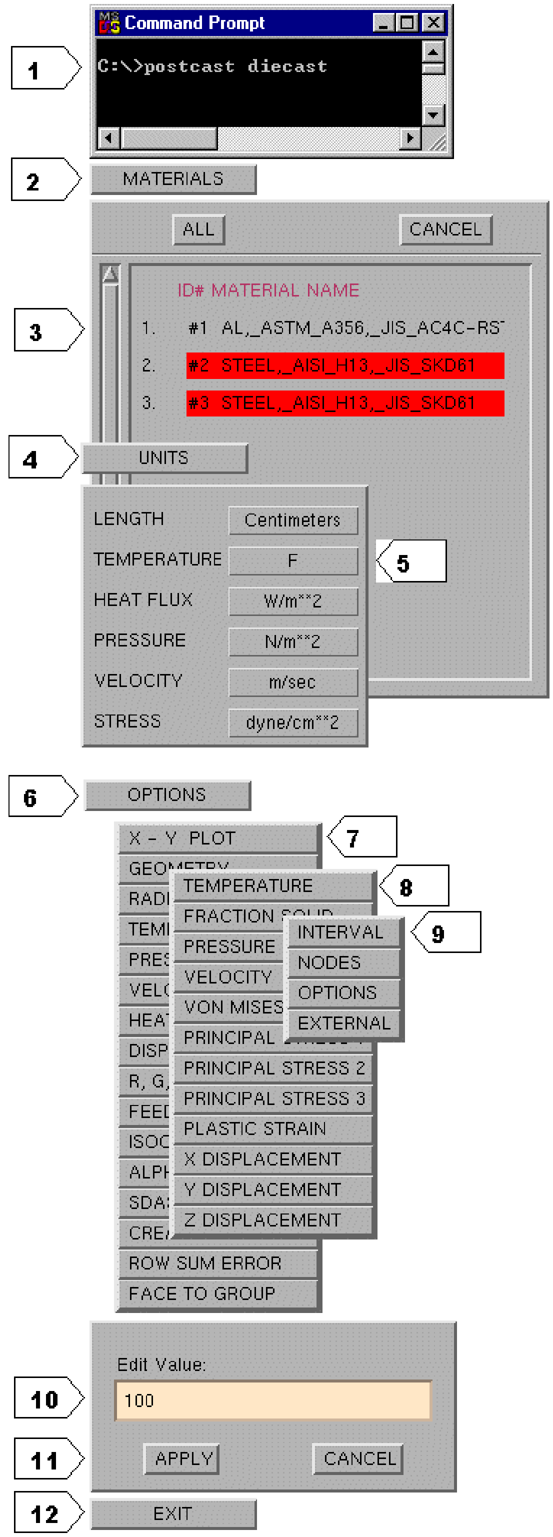
1. datacast diecast
2. procast diecast
3. prostat diecast (in another window)

Extra Notes

If DataCAST or ProCAST list any warnings or errors, please return to PreCAST, **RESTART** the model and examine the setup as specified by the warning / error message.

Check the Analysis

After the solution has finished, check to see if enough cycles have been simulated to reach the normal operating temperature. To accomplish this, we will use PostCAST. PostCAST will allow us to view the temperatures of many nodes at the same time and determine the normal operating temperature. We know we have performed enough cycles if the peaks of the temperature curves begin to level off.



1. postcast diecast
2. **MATERIALS**
3. Highlight only the die materials
4. **UNITS**
5. Change the temperature units to F
6. **OPTIONS**
7. **X-Y PLOT**
8. **TEMPERATURE**
9. **INTERVAL**
10. Enter 100
11. **APPLY** to view the graph
12. **EXIT** when finished

Extra Notes

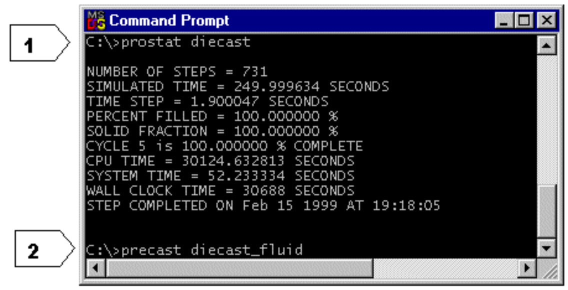
PostCAST performs many functions. These include:

- X-Y Graphing
- Extra Calculations for Porosity Determination
- Isochron Calculation
- Result Format Conversion

Instructions for using PostCAST can be found in Section 6 of the ProCAST User's Manual.

Start PreCAST

When it is determined that enough cycles have been run to reach the steady state temperature in the dies, we are ready to set up and run the coupled fluid / thermal analysis. Before we do this, run prostat on the thermal analysis and make a note of the number of steps that were run.



The setup for the thermal-only analysis that we did in the first half of this tutorial contains much of the information needed for the fluid / thermal simulation. So, to avoid having to redo some work, we will read in the thermal setup and modify some parameters. When we save, we do not want to overwrite the thermal analysis, so we will start precast with a new name.

1. prostat diecast
2. precast diecast_fluid

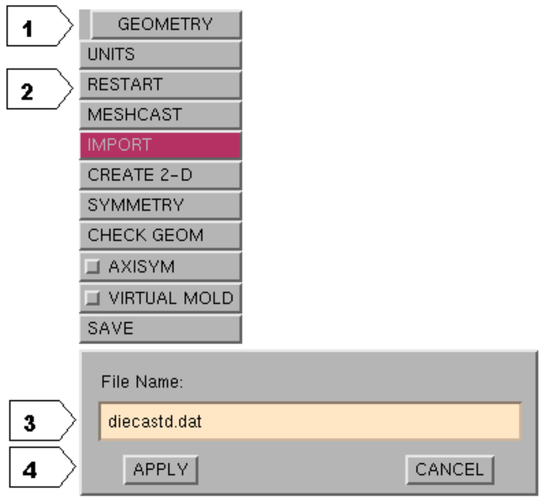
Extra Notes

For organizational reasons, it is better to have the thermal and fluid solutions in separate directories. For this tutorial it is easier if the two simulations occupy the same directory.

Reading the PreCAST Restart File

Now we will read in the file that contains the setup for the thermal-only analysis. Anytime you wish to read in a PreCAST file (d.dat file), simply start PreCAST, with an appropriate prefix specified, and use **RESTART** to read in the data.

When the file has been read, a form will appear listing information about the analysis setup, including entity data, number of materials and a listing of boundary conditions.

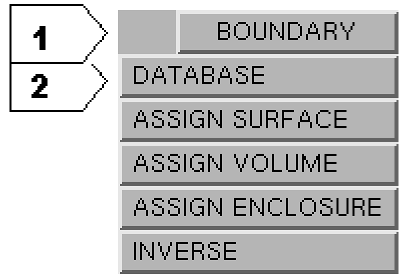


1. **GEOMETRY**
2. **RESTART**
3. Change the filename to "diecastd.dat"
4. **APPLY**

Extra Notes

Creating Boundary Condition Properties

For the fluid simulation, we need to specify the rate of metal entering the cavity and the pouring temperature of the metal.

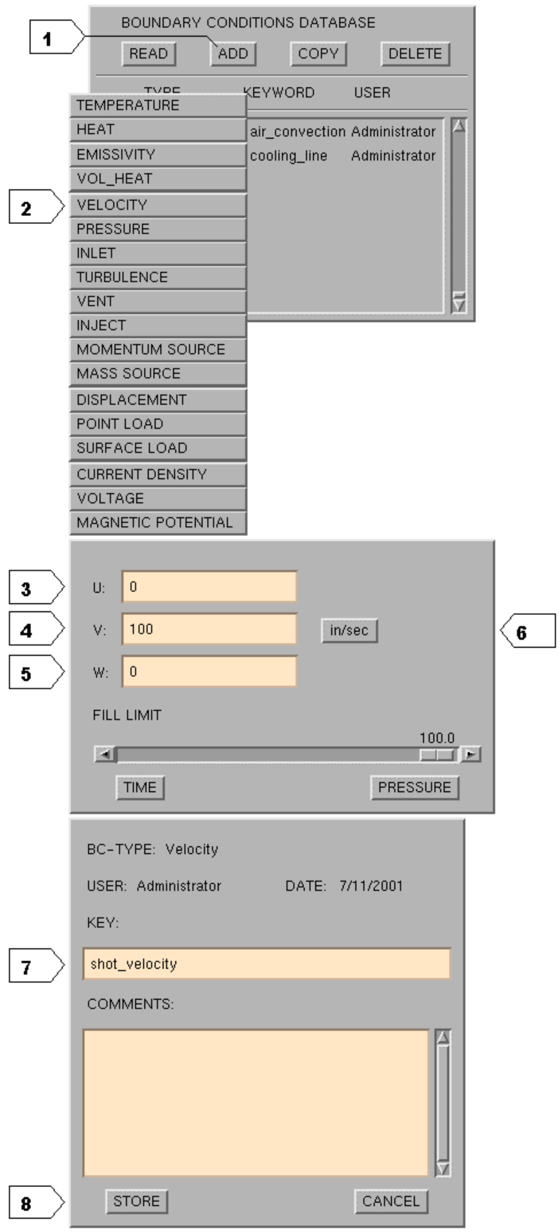


1. **BOUNDARY**
2. **DATABASE**

Extra Notes

This simulation will demonstrate using a velocity boundary condition to introduce metal into the cavity. The value used for the velocity condition can be found in different ways. If the volume of the cavity, filling time and inlet area are known, the inlet velocity is equal to the volume divided by the product of the filling time multiplied by the area (V/tA). The velocity of the shot piston can also be used as the value for the velocity boundary condition.

Entering metal into the cavity can also be done with the pressure, turbulence and inject boundary conditions



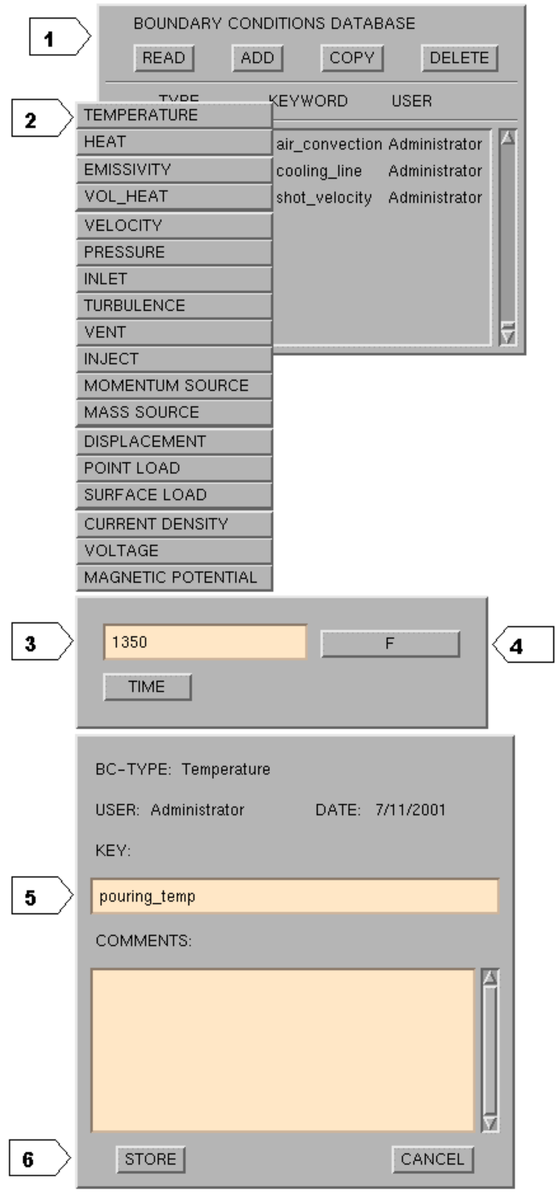
1. **ADD**
2. **VELOCITY**
3. Enter 0 for U
4. Enter 100 for V
5. Enter 0 for W
6. Change units to in/sec
7. Enter “shot_velocity”
8. **STORE**

Extra Notes

U, V and W are velocities corresponding to the X, Y and Z directions, respectively.

Non-Coincident Mesh:
After entering the velocity boundary condition described above, ADD a new velocity boundary condition that has U, V and W set to 0. Name this boundary condition “no_slip”. This is used to define the region that the flow is allowed to occupy. This boundary condition is automatically defined when using a Coincident mesh.

Enter the temperature of the metal that will be entering the inlet (pouring temperature).

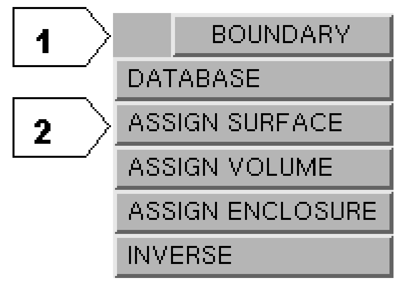


1. **ADD**
2. **TEMPERATURE**
3. Enter 1350
4. Change units to F
5. Enter "pouring_temp"
6. **STORE**

Extra Notes

Assign the Boundary Conditions

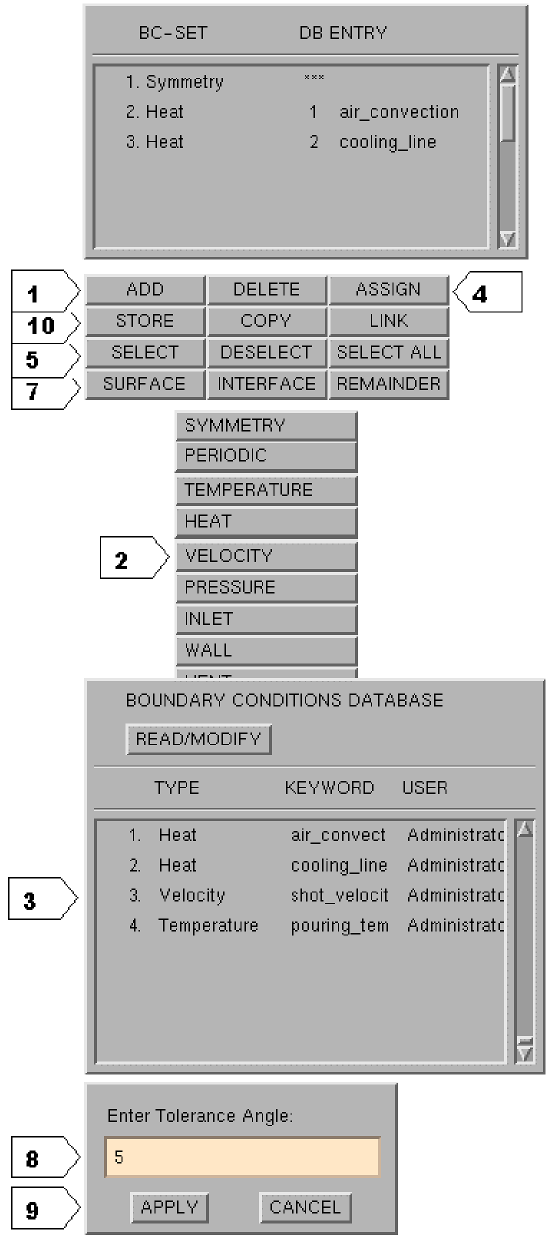
With the new boundary conditions defined, they need to be assigned to the appropriate regions on the mesh. The inlet velocity and pouring temperature both describe the metal entering the cavity and will be assigned to the location where the metal enters – the biscuit surface.



1. **BOUNDARY**
2. **ASSIGN SURFACE**

Extra Notes

Assign the velocity condition to the inlet surface.



1. **ADD**
2. **VELOCITY**
3. Highlight the “shot_velocity” Velocity database entry
4. **ASSIGN**
5. **SELECT**
6. Select an element on the biscuit surface
7. **SURFACE**
8. Enter “5”
9. **APPLY**
10. **STORE**

Extra Notes

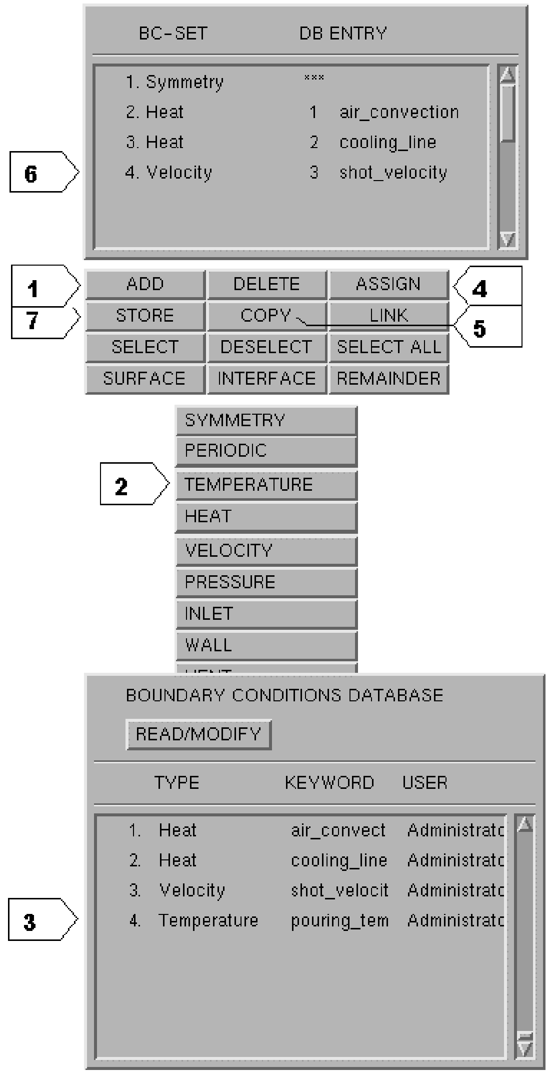
Non-Coincident Mesh:
Before completing the steps above:

- **ADD** a **VELOCITY** condition
- Highlight the “no_slip” velocity boundary condition in the database
- **ASSIGN** the database entry
- Use **MAT SELECT** and highlight only the Al356 material, press **APPLY**
- Press **SELECT ALL**
- **STORE** the selection

Note that when the same region has been selected for separate boundary conditions of the same type (ie, velocity), the selection of the last boundary condition will override previous selections. Thus, even though all of the casting nodes are stored for the “no slip” velocity, the “shot_velocity” selection of the biscuit surface removes those same nodes from the “no slip” condition.

Assign the pouring temperature condition to the inlet surface.

Since the temperature condition is assigned to the inlet surface, we can simply **COPY** the velocity selection to the temperature condition. Once the casting is full, the inlet velocity is turned off and the temperature boundary condition is also terminated. This no longer forces the nodes on the biscuit surface to be equal to the pouring temperature and allows for the biscuit to cool off, should a heat boundary condition or some other means of heat transfer be assigned to that region.

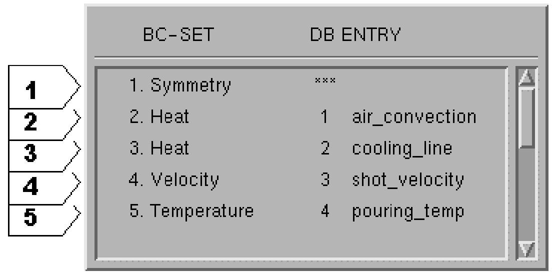


---

1. **ADD**
2. **TEMPERATURE**
3. Highlight the "pouring_temp" Temperature database entry
4. **ASSIGN**
5. **COPY**
6. Highlight the inlet velocity boundary condition in the BC Set Form (not the database form)
7. **STORE**

Extra Notes

To make sure that the correct entities have been assigned to the correct boundary condition, click on each boundary condition in the bc-set column. When a boundary condition is selected, the entities assigned to that condition will highlight.



---

1. Click on the “Symmetry” BC
2. Click on the first “Heat” BC
3. Click on the second “Heat” BC
4. Click on the “Velocity” BC
5. Click on the “Temperature” BC

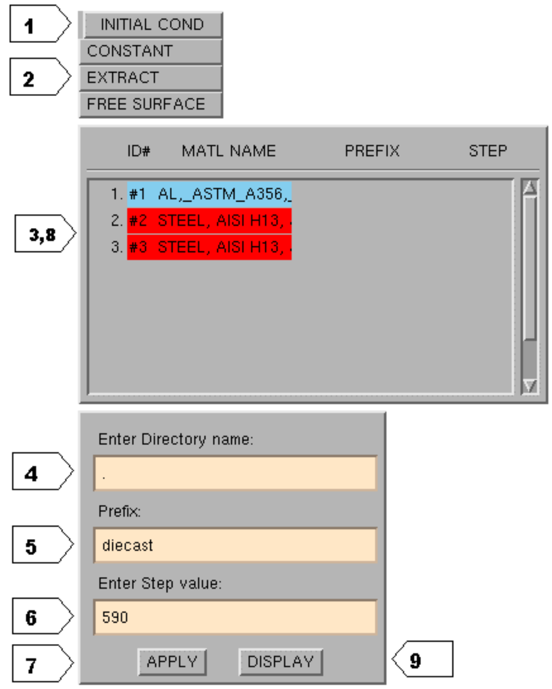
Extra Notes

Non-Coincident Mesh:
There will be two Velocities listed, one for the inlet velocity, one for the no slip velocity. The inlet velocity will be on the biscuit surface.   The no slip velocity will cover the entire casting except for the biscuit surface.

Initial Condition Definition

Again, this fluid / thermal analysis simulates an entire cycle that occurs after the dies heat up. Since we want to use dies that have some heat built up in them, we will set the initial die temperature to be the same as that at the end of the thermal-only analysis. Therefore, we will **EXTRACT** the die temperatures out of that analysis and use them for this fluid / thermal analysis.

The last written step refers to the results of the last step written out from the previous analysis. For example, if the simulation ran 594 steps and every 5 steps were saved, you would enter 590 as the last written step.



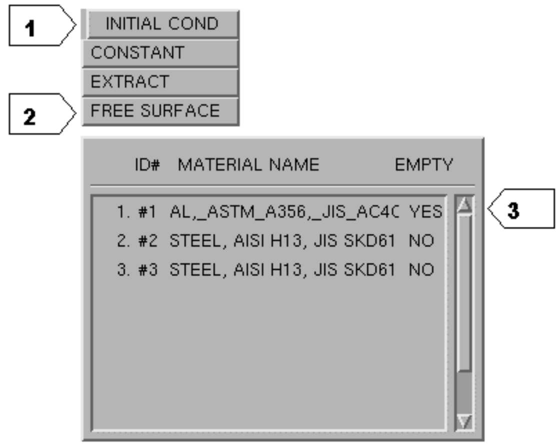
---

1. **INITIAL COND**
2. **EXTRACT**
3. Highlight the die materials
4. Enter "." (a "." means current directory)
5. Enter "diecast"
6. Enter the number of the last written step
7. **APPLY**
8. Highlight the die materials again
9. **DISPLAY**

Extra Notes

When you press **DISPLAY**, the temperature of each node is displayed. The units for the temperature is always degrees Kelvin.

The last process parameter we need to define is specifying the empty cavity.



1. **INITIAL COND**
2. **FREE SURFACE**
3. Toggle No to Yes for the Al356 Material

Extra Notes

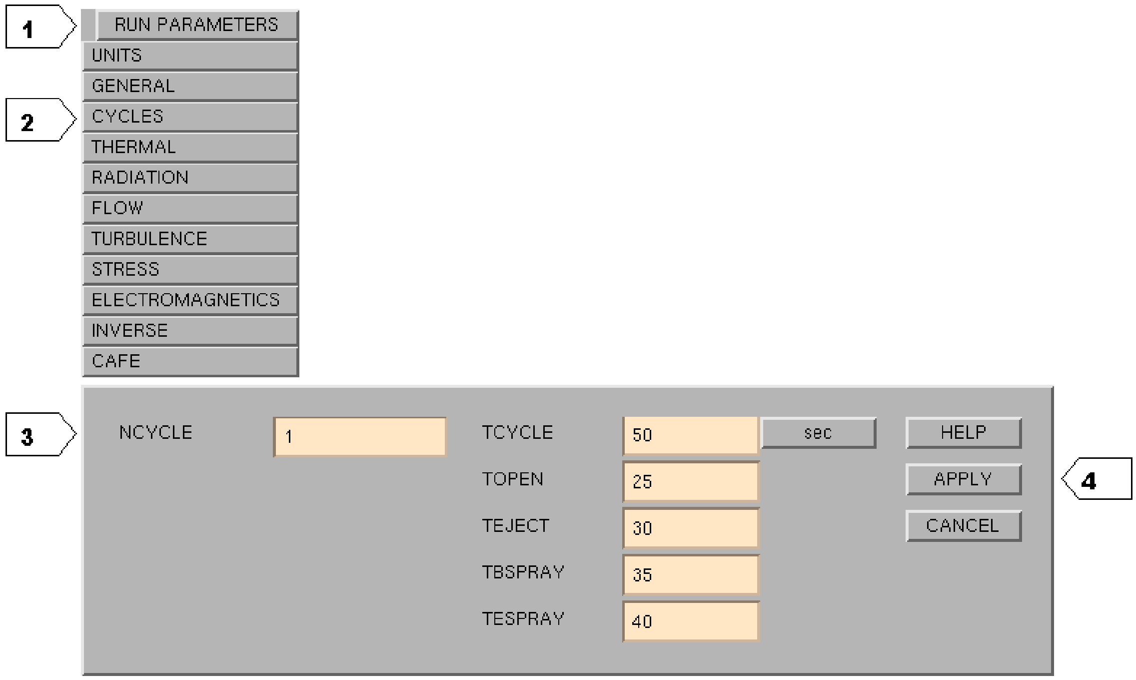
Modify the Run Parameters

Now we will modify the parameters which control the solving of the analysis. In the cycle form, we are simply changing the number of cycles to run from 5 to 1.

1. **RUN PARAMETERS**
2. **CYCLES**
3. Change the value of NCYCLE to 1
4. **APPLY**

See next page for illustrations.

Extra Notes


1
RUN PARAMETERS
UNITS
GENERAL
2
CYCLES
THERMAL
RADIATION
FLOW
TURBULENCE
STRESS
ELECTROMAGNETICS
INVERSE
CAFE
3
NCYCLE
1
TCYCLE
50
sec
TOPEN
25
TEJECT
30
TBSPRAY
35
TESPRAY
40
HELP
APPLY
CANCEL
4

Modify the parameters that control the fluid analysis. The first item to change will be fluid solution control, FLOW. If set to 1, the fluid solution will be performed throughout the solve. Unless you are interested in natural convection effects, it saves a lot of time to set FLOW to 3. This turns off the fluid solution when the cavity has filled. Another parameter to be changed is VFREQ. Set this to 5 to save fluid results every 5 steps. Setting FREESF to 1 turns on the Free Surface Solver. Increasing the COURANT Limit to 100 allows for a faster solve. Set LVSURF to 1. This parameter determines the percent filled the cavity must achieve before the fluid solution can turn off (when using FLOW option 3). An LVSURF value of 1 corresponds to 100%. Finally, set WSHEAR and EDGE both to 1. These parameters control the description of the flow. In this scenario, they are set to high speed settings.

1. **RUN PARAMETERS**
2. **FLOW**
3. Change the value of FLOW to 3
4. Change the value of VFREQ to 5
5. Change the value of FREESF to 1
6. Change the value of COURANT to 100
7. Change the value of LVSURF to 1
8. Change the value of WSHEAR to 1
9. Change the value of EDGE to 1
10. **APPLY**

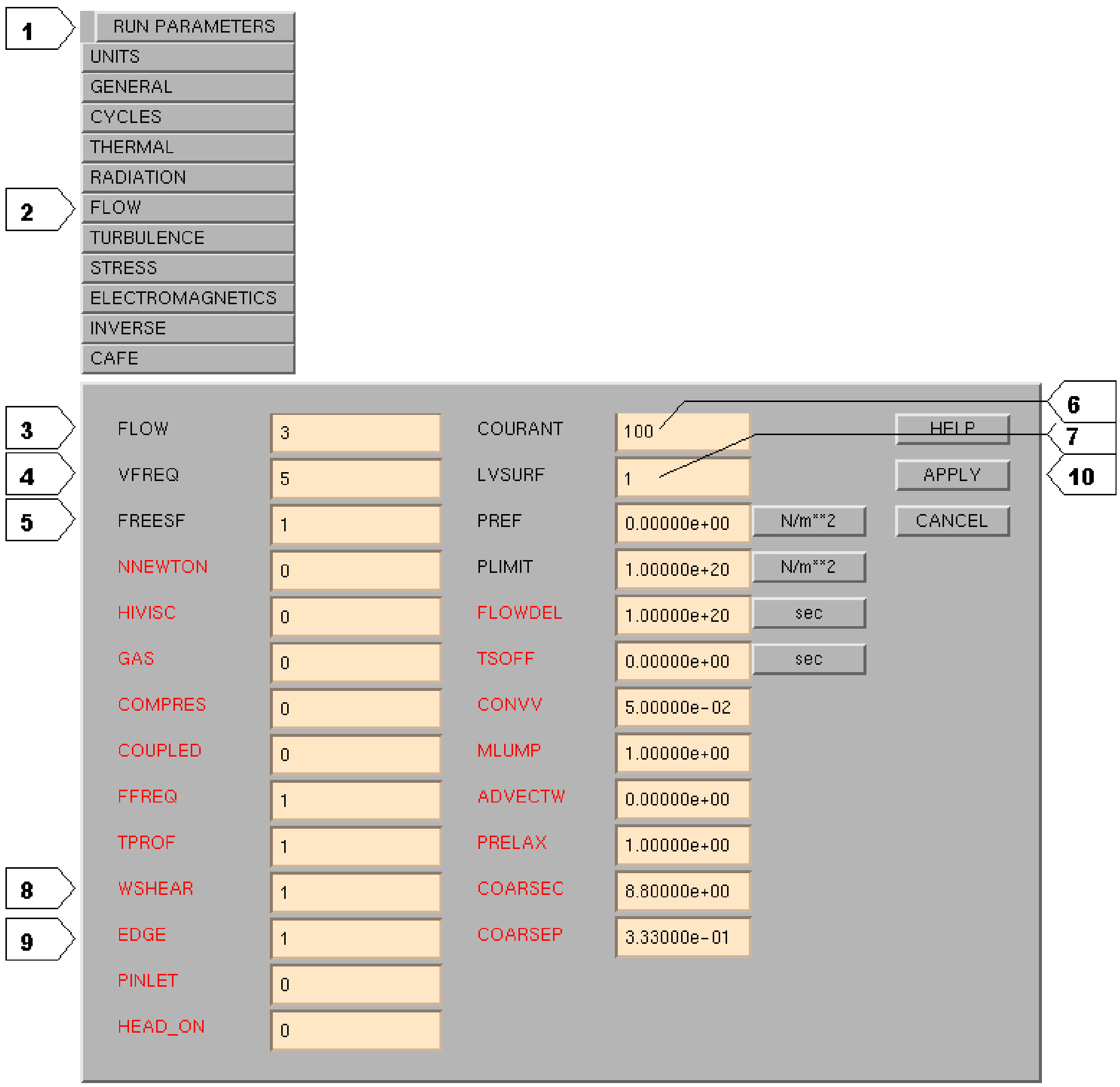
See next page for illustrations.

Extra Notes

There are a few other important flow run parameters:

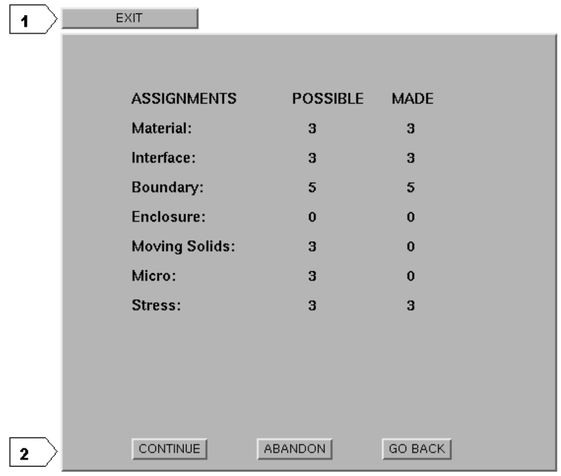
GAS – turns on or off the solver that determines back pressure. The default is off (0), which simulates the filling of an evacuated cavity.

PINLET – if you are using a Pressure boundary condition to introduce metal into the cavity, this parameter must be turned on (1).


1
2
3
4
5
6
7
8
9
10
RUN PARAMETERS
UNITS
GENERAL
CYCLES
THERMAL
RADIATION
FLOW
TURBULENCE
STRESS
ELECTROMAGNETICS
INVERSE
CAFE
FLOW 3
VFREQ 5
FREESF 1
NNEWTON 0
HIVISC 0
GAS 0
COMPRES 0
COUPLED 0
FFREQ 1
TPROF 1
WSHEAR 1
EDGE 1
PINLET 0
HEAD_ON 0
COURANT 100
LVSURF 1
PREF 0.00000e+00 N/m**2
PLIMIT 1.00000e+20 N/m**2
FLOWDEL 1.00000e+20 sec
TSOFF 0.00000e+00 sec
CONVV 5.00000e-02
MLUMP 1.00000e+00
ADVECTW 0.00000e+00
PRELAX 1.00000e+00
COARSEC 8.80000e+00
COARSEP 3.33000e-01
HELP
APPLY
CANCEL

<u>Save and Exit PreCAST</u>

Everything should now be setup for this coupled fluid / thermal high pressure die casting simulation. After pressing **EXIT**, a summary screen appears showing the number of possible assignments and the number of assignments made. For this solution, the numbers for Materials, Interface, Boundary and Enclosure should be same in each column. Should there be any difference, **GO BACK** to the section in question and check your setup. If the numbers are matching, then **CONTINUE**.



---

1. **EXIT**
2. **CONTINUE**

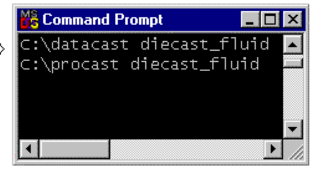
Extra Notes

Start Solving the Analysis

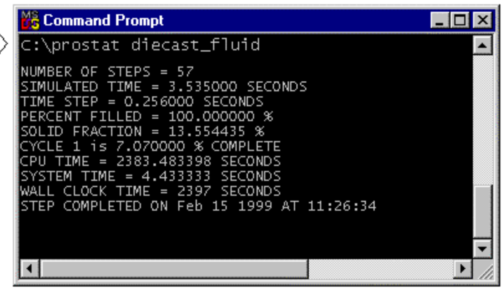
The next step in the simulation process is to run DataCAST. This non-interactive script simply converts the units to a consistent system (CGS) and does some simple error checking. Should any problems exist, warning or error messages will be shown on the screen.

If DataCAST runs without messages, then the setup is ready to be solved using the ProCAST executable. While the simulation is running, you can check on the progress by running the ProSTAT script in another window. This summary lists the current simulated time, time step, solve time and other statistics. This solution is complete when cycle 1 is 100% complete.

1,2



3



---

1. datacast diecast_fluid
2. procast diecast_fluid
3. prostat diecast_fluid (in another window)

Extra Notes