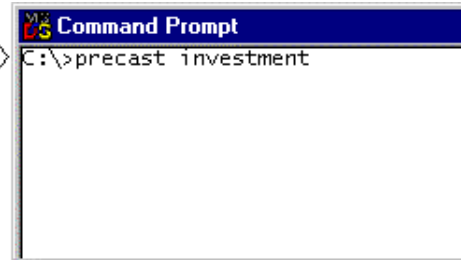


Start PreCAST

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 “investment”.

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

1



-
1. precast investment

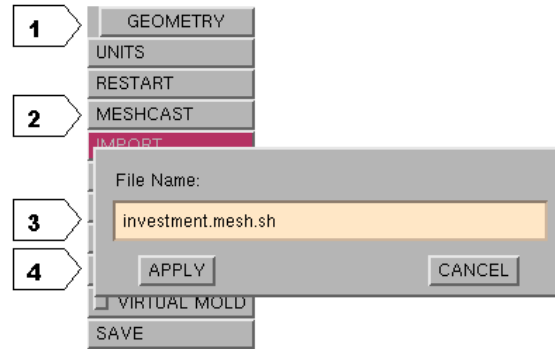
Extra Notes

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 "file.mesh". However, when a layer of shell has been added, the filename has a ".sh" appended.



-
- 1. GEOMETRY**
 - 2. MESHCAST**
 - 3.** Change filename to "investment.mesh.sh"
 - 4. 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.

The MODEL contains:

60496 Elements

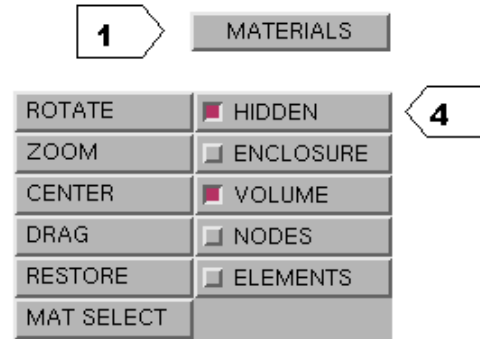
21918 Nodes

2 Material types

There are no steps on this page.

Extra Notes

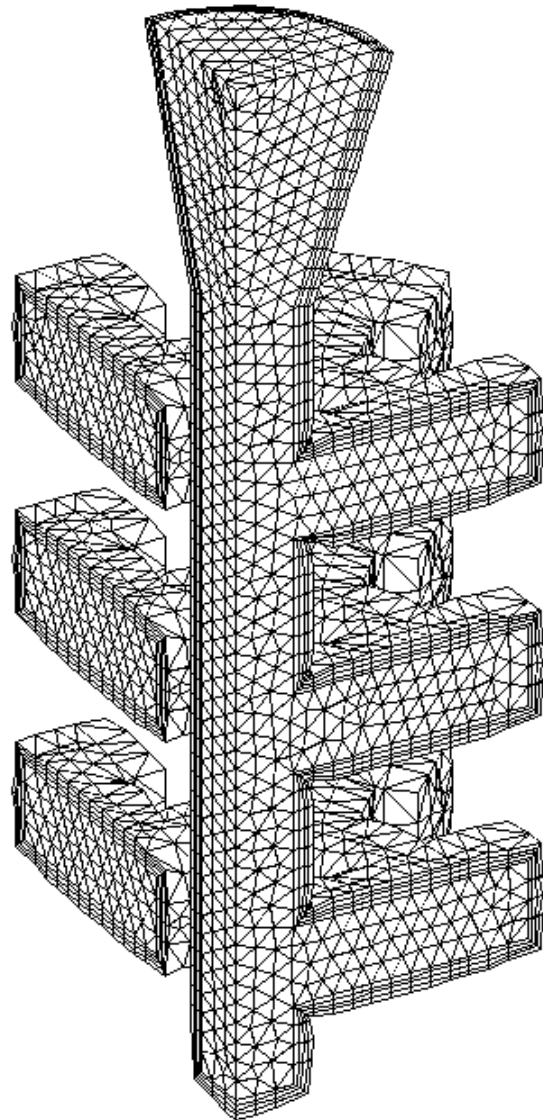
To see the geometry, one simply needs to press any of the other buttons on the top row of the PreCAST window. The control of how the model is viewed is done by hot keys, mouse functions and/or buttons and forms. These controls are described in more detail in Section 1 of the ProCAST Users' Manual.



1. MATERIALS

2. Press CTRL+SHIFT+X twice.
3. Press SHIFT+Z
4. **HIDDEN**

Extra Notes



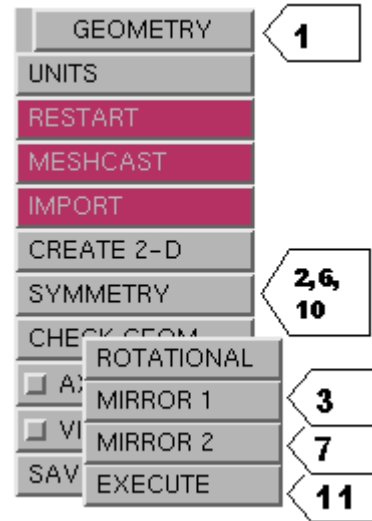
Defining Symmetry

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 depending upon which mesher you are using. 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. MIRROR 2

8. Enter the coordinates of a plane at
 $Y = 0$ (see next page)

9. **APPLY** (see next page)

10. SYMMETRY

11. EXECUTE

Extra Notes

As can be seen on the screen, only 1/4th 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

X0:	<input type="text" value="0"/>	Y0:	<input type="text" value="-1"/>	Z0:	<input type="text" value="-1"/>
X1:	<input type="text" value="0"/>	Y1:	<input type="text" value="-1"/>	Z1:	<input type="text" value="1"/>
X2:	<input type="text" value="0"/>	Y2:	<input type="text" value="1"/>	Z2:	<input type="text" value="1"/>

5

APPLY

CANCEL

8

X0:	<input type="text" value="-1"/>	Y0:	<input type="text" value="0"/>	Z0:	<input type="text" value="-1"/>
X1:	<input type="text" value="-1"/>	Y1:	<input type="text" value="0"/>	Z1:	<input type="text" value="1"/>
X2:	<input type="text" value="1"/>	Y2:	<input type="text" value="0"/>	Z2:	<input type="text" value="1"/>

9

APPLY

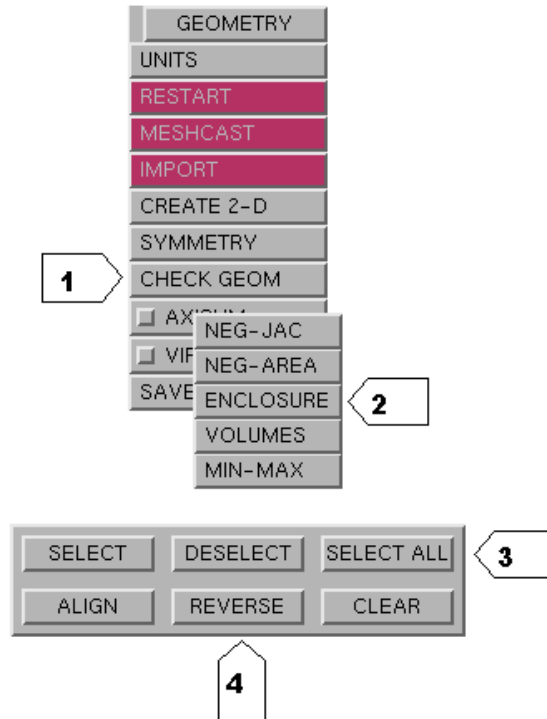
CANCEL

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. Therefore, a problem like this will want to be found as soon as possible.

For an investment casting analysis, there is some kind of enclosure around the casting and shell. This “control volume” is needed for the radiation calculations. Part of the requirement of the enclosure is that the normals of the elements on the enclosure point towards the casting. This section finds that the normals are all pointing away from the casting and fixes this problem.

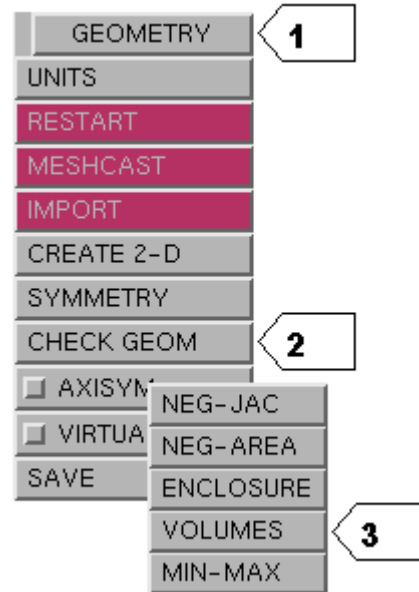
- 1. CHECK GEOM**
- 2. ENCLOSURES**
- 3. SELECT ALL**
- 4. REVERSE**



Extra Notes

An element normal is the direction perpendicular to the element as determined by the elements connectivity, or order of nodes.

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 shell mold with metal.



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

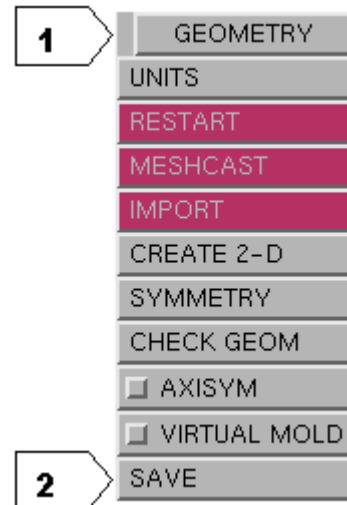
Extra Notes

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 inches (you can see this if you press **GEOMETRY..UNITS**). Therefore, the volume given is in cubic inches.

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 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 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.

Assigning Materials

Now we will define the materials being used in this simulation. For this example, Stainless 316 will be assigned as the metal to be cast, and a Fused Silica shell entry will be assigned as the shell 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 STEEL_Stainless_AISI Type 316 in the Material Database

5. ASSIGN

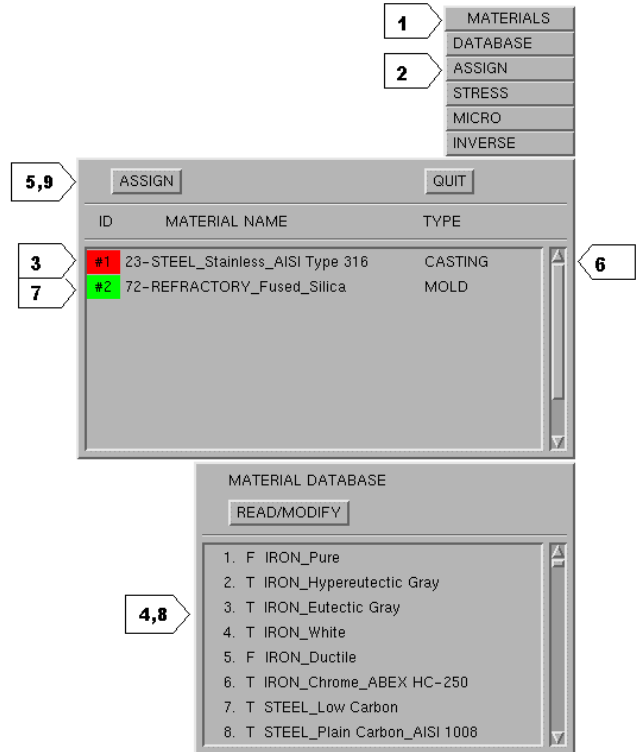
6. Toggle the T to F in the Type column for this material

7. Click on #2

8. Highlight the material "Shell" shell material in the Material Database

9. ASSIGN

10. Toggle the N to Y in the Mold column for this material



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.

Extra Notes

There are four choices for material type: **T**hermal, **F**luid, **f**ilter, and **f**Oam. When you are performing a thermal-only simulation, you will only have Thermal materials. When you are performing a fluid-only simulation or coupled Thermal-Fluid, the casting (or fluid region) will be designated as a Fluid material.

There are two choices for mold: **Y**es (indicating that the material is part of the mold) and **N**o (indicating that the material is not part of the mold).

Create the Interface Conditions

When heat is transferred across dissimilar materials, it will probably not pass through as 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 a new interface heat transfer coefficient 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

3. ADD

4. STANDARD

5. Enter 0.02

6. Change the units to $\text{Cal}/\text{cm}^2/\text{C}/\text{sec}$

7. Enter “investment” in the Key region.

8. STORE

The screenshot displays a multi-step software interface for creating a new interface condition. The steps are numbered 1 through 8, corresponding to the instructions on the left.

- Step 1:** The 'INTERFACE' menu is selected.
- Step 2:** The 'DATABASE' option is chosen from the dropdown.
- Step 3:** The 'ADD' button is clicked in the 'INTERFACE DATABASE' window.
- Step 4:** The 'STANDARD' option is selected from the 'DIE COMBO' dropdown.
- Step 5:** The value '0.02' is entered into the input field.
- Step 6:** The units 'cal/cm**2/C/sec' are selected from the dropdown.
- Step 7:** The word 'investment' is entered into the 'KEY' field.
- Step 8:** The 'STORE' button is clicked to save the entry.

The interface includes a 'TEMPERATURE' button and a 'TIME' button, as well as a 'COMMENTS' section with a text area and a 'CANCEL' button.

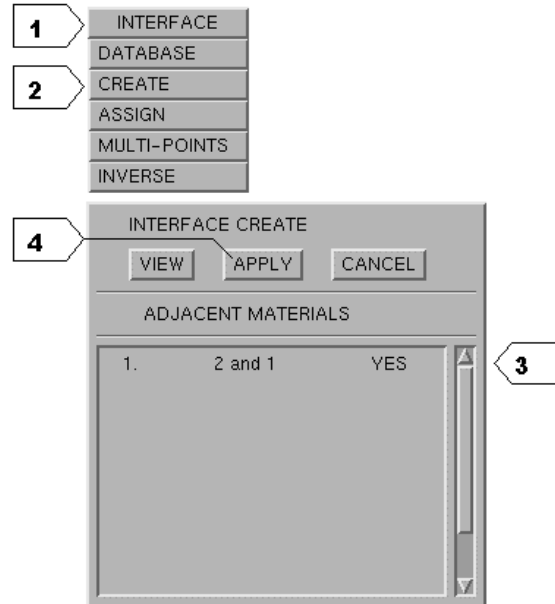
Extra Notes

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.

Create the Interface in the FE Model

The model used in this tutorial is an example of a typical mesh generated by a meshing package. In this coincident mesh, nodes are shared by both materials at the interface. Thus, only one temperature is known. To simulate the heat transfer across the interface, two temperatures are needed, that for the casting material and that for the shell material. By creating an interface in the model, we generate a new set of nodes allowing for the tracking of two temperatures, and thus, the proper simulation of heat transfer across the interface.



-
- 1. INTERFACE**
 - 2. CREATE**
 - 3.** Toggle “No” to “Yes”
 - 4. APPLY**

Extra Notes

Assigning the Interface Condition

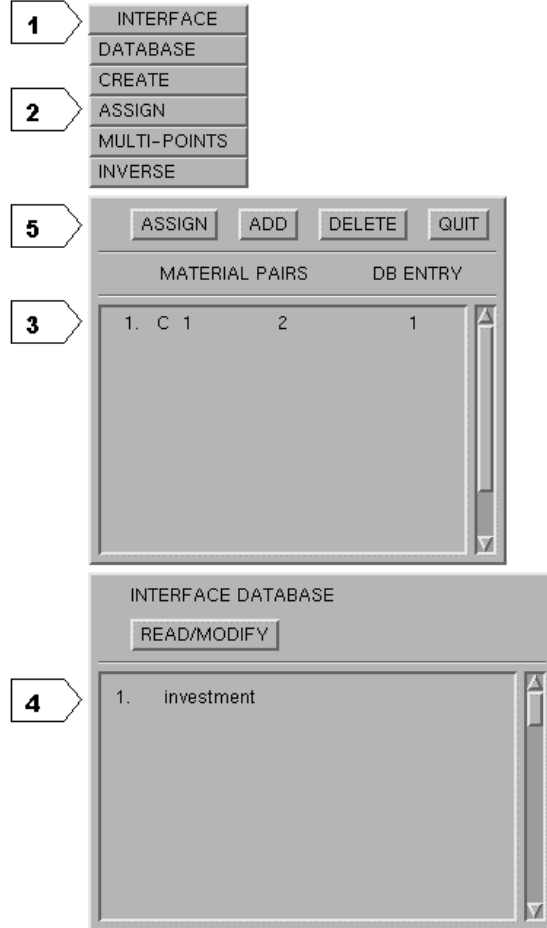
Now that an interface coefficient has been added to the database and an interface has been created, we need to assign that database entry to the interface in this model.

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.

1. INTERFACE

2. ASSIGN

- 3.** Click on the first material ID in the material pair
- 4.** Highlight the “investment” entry in the database.
- 5. ASSIGN**



Extra Notes

You will notice that clicking the second material ID in the material pair will switch the order of the material pair. The order of the materials is important only when you are using an interface heat transfer coefficient that is temperature dependent. In this case, it is necessary to identify which material to base the calculation of heat transfer. The material used will then be the first one listed in the material pair.

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 simulation we have three parameters to define: pouring temperature, heat transfer off of the shell to the air and filling rate. First the pouring temperature will be defined.

-
- 1. BOUNDARY**
 - 2. DATABASE**
 - 3. ADD**
 - 4. TEMPERATURE**
 - 5.** Enter 2750
 - 6.** Change the unit to F
 - 7.** Change the Key to “pouring_temp”
 - 8. STORE**

Extra Notes

The screenshot illustrates the steps to define a boundary condition in a software application. The interface consists of several windows and a list of options.

- Step 1:** The **BOUNDARY** button is selected in the top menu.
- Step 2:** The **DATABASE** button is selected in the top menu.
- Step 3:** The **ADD** button is selected in the **BOUNDARY CONDITIONS DATABASE** window.
- Step 4:** The **TEMPERATURE** option is selected from the list of boundary condition types.
- Step 5:** The value **2750** is entered into the input field.
- Step 6:** The unit **F** is selected from the unit dropdown menu.
- Step 7:** The key **pouring_temp** is entered into the **KEY:** field.
- Step 8:** The **STORE** button is clicked to save the boundary condition.

The **BOUNDARY CONDITIONS DATABASE** window displays a table with columns **TYPE**, **KEYWORD**, and **USER**. The **TEMPERATURE** option is currently selected.

The **BOUNDARY CONDITIONS DATABASE** window also includes buttons for **READ**, **ADD**, **COPY**, and **DELETE**.

The **BOUNDARY CONDITIONS DATABASE** window also includes a **TIME** button.

The **BOUNDARY CONDITIONS DATABASE** window also includes a **BC-TYPE: Temperature** label.

The **BOUNDARY CONDITIONS DATABASE** window also includes a **USER: Administrator** label and a **DATE: 6/26/2001** label.

The **BOUNDARY CONDITIONS DATABASE** window also includes a **KEY:** label.

The **BOUNDARY CONDITIONS DATABASE** window also includes a **COMMENTS:** label.

The **BOUNDARY CONDITIONS DATABASE** window also includes a **STORE** button and a **CANCEL** button.

Next, we will define the heat transfer that radiates off of the shell mold. By toggling View Factor to “ON”, we specify that a calculation be performed that determines a factor of heat transfer from one element to another. For example, if the element faces are toward each other and relatively close, there will be a high rate of heat transfer. However, if two element faces are not pointing towards each other (or one is shadowed from the other), then there will be no heat transfer. An emissivity value is then entered to control the rate of radiation.

The film coefficient and ambient temperature simulation the heat loss to convection.

- 1. ADD**
- 2. HEAT**
- 3.** Enter .0015 for Film Coeff
- 4.** Toggle the Units to cal/cm**2/C/sec
- 5.** Enter .5 for Emissivity
- 6.** Enter 80 for Ambient Temperature
- 7.** Toggle the Units to F
- 8.** Toggle View Factor to ON
- 9.** Enter “shell” for the Key
- 10. STORE**

The image shows two software windows. The top window is the 'BOUNDARY CONDITIONS DATABASE' with buttons for READ, ADD, COPY, and DELETE. It contains a table with columns TYPE, KEYWORD, and USER. The first row is '1. Heat pouring_temp User'. A list of boundary condition types is shown on the left, with 'HEAT' selected. The bottom window is the 'Heat' boundary condition dialog. It has fields for FILM COEFF (.0015), EMISSIVITY (.5), AMBIENT TEMP (80), and FLUX. There are unit toggles for cal/cm**2/C/sec, F, and W/m**2. A 'VIEW FACTOR' toggle is set to 'ON'. At the bottom, there is a 'KEY' field containing 'shell' and a 'COMMENTS' text area. Buttons for 'STORE' and 'CANCEL' are at the bottom right. Numbered callouts 1 through 10 point to specific elements in both windows.

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., shell mold 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.

In the same manner, mold wrap, such as wool, can be modeled in two ways: physically or using emissivity. If you have the wrap in the solid model, you can assign wrap material properties. Or the emissivity can be lowered in a certain area to reduce the heat radiated from a certain area.

Finally, add a velocity boundary condition that will simulate metal entering the pouring cup. In order to determine the rate of filling we will use the volume of the casting that was found using CHECK GEOM..VOLUMES. If we divide the entire volume by the inlet area and approximate filling time, the result is the filling rate. So, assuming a two inch inlet diameter and an approximate fill time of 3 seconds, the velocity will be about 4 inches / second. The direction of the pour will be in the negative-Z direction. This corresponds to a negative value in the W velocity direction.

1. **ADD**
2. **VELOCITY**
3. Enter 0 for U and V
4. Enter -4 for W
5. Change the units to “in/sec”
6. **STORE**

Extra Notes

The U, V and W velocity directions correspond to the X, Y and Z global directions.

Quite often there is significant time between the removal of the shell mold from the oven and the beginning of the filling. In this time, there can be a lot of cooling of the mold. To simulate this, simply use the **TIME** function of the velocity boundary condition to start the flow at a time other than the beginning of the simulation.

BOUNDARY CONDITIONS DATABASE

1. Temperature pouring_temp Administrator
2. Heat shell Administrator

2. VELOCITY

3. U: 0
V: 0 in/sec
W: -4

4. STORE

5. BC-TYPE: Velocity
USER: Administrator DATE: 6/26/2001
KEY: pouring_rate
COMMENTS:

6. STORE

Define the Radiation Condition for the Enclosure

The radiation heat transfer from the shell mold to the air has been defined and assigned in the boundary condition section. Now we need to specify the role of the enclosure.

In this example, we are using the enclosure to simulate a control volume consisting only of air and will be specifying only an emissivity and ambient temperature. In this case, there will be little heat radiated from the air back onto the mold. However, if there were some other item in the process, such as covering the casting with a can during solidification, there would be a much more radiation of heat back onto the mold.

- 1. ADD**
- 2. TEMPERATURE**
- 3.** Enter 80
- 4.** Toggle Unit to F
- 5.** Enter "enclosure" as the Key
- 6. STORE**

Extra Notes

The screenshot shows the 'BOUNDARY CONDITIONS DATABASE' window. At the top are buttons for 'READ', 'ADD', 'COPY', and 'DELETE'. Below these is a table with columns 'TYPE', 'KEYWORD', and 'USER'. The 'TEMPERATURE' type is selected, and the 'ADD' button is highlighted. The table contains three entries: 'HEAT' with keyword 'pouring_temp' and user 'Administrator', 'EMISSIVITY' with keyword 'shell' and user 'Administrator', and 'VOL_HEAT' with keyword 'pouring_rate' and user 'Administrator'. Below the table is a list of boundary condition types: 'VELOCITY', 'PRESSURE', 'INLET', 'TURBULENCE', 'VENT', 'INJECT', 'MOMENTUM SOURCE', 'MASS SOURCE', 'DISPLACEMENT', 'POINT LOAD', 'SURFACE LOAD', 'CURRENT DENSITY', 'VOLTAGE', and 'MAGNETIC POTENTIAL'. The 'TEMPERATURE' type is selected. Below the list is a form with a text input field containing '80', a unit dropdown menu set to 'F', and a 'TIME' button. The 'STORE' button is highlighted. Below the form is a section for 'BC-TYPE: Temperature' with fields for 'USER: Administrator', 'DATE: 7/9/2001', 'KEY:', and 'COMMENTS:'. The 'KEY:' field contains 'enclosure'. The 'COMMENTS:' field is empty. At the bottom are 'STORE' and 'CANCEL' buttons.

TYPE	KEYWORD	USER
HEAT	pouring_temp	Administrator
EMISSIVITY	shell	Administrator
VOL_HEAT	pouring_rate	Administrator

VELOCITY
PRESSURE
INLET
TURBULENCE
VENT
INJECT
MOMENTUM SOURCE
MASS SOURCE
DISPLACEMENT
POINT LOAD
SURFACE LOAD
CURRENT DENSITY
VOLTAGE
MAGNETIC POTENTIAL

80 F
TIME

BC-TYPE: Temperature
USER: Administrator DATE: 7/9/2001
KEY: enclosure
COMMENTS:

STORE CANCEL

Now enter the emissivity of the enclosure.

1. **ADD**
2. **EMISSIVITY**
3. Enter .99
4. Enter “enclosure” as the Key
5. **STORE**

Extra Notes

BOUNDARY CONDITIONS DATABASE

1. **ADD**

TYPE	KEYWORD	USER
TEMPERATURE		
HEAT	pouring_temp	Administrator
EMISSIVITY	shell	Administrator
VOL_HEAT	pouring_rate	Administrator
VELOCITY	enclosure	Administrator
PRESSURE		
INLET		
TURBULENCE		
VENT		
INJECT		
MOMENTUM SOURCE		
MASS SOURCE		
DISPLACEMENT		
POINT LOAD		
SURFACE LOAD		
CURRENT DENSITY		
VOLTAGE		
MAGNETIC POTENTIAL		

2. **EMISSIVITY**

3. .99

TEMP

BC-TYPE: Emissivity

USER: Administrator DATE: 7/9/2001

KEY:

4. enclosure

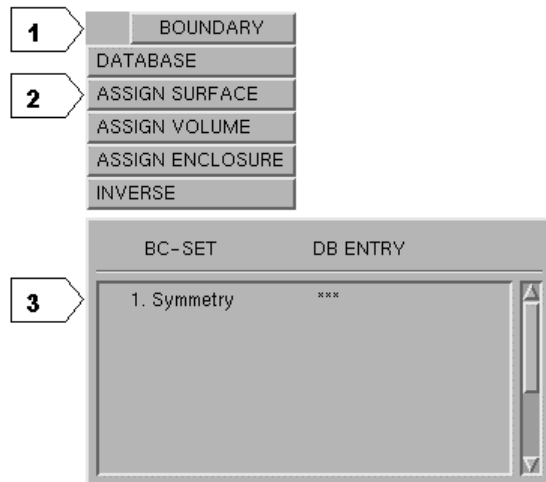
COMMENTS:

5. **STORE** **CANCEL**

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. Should you not see any selected faces, go back to **GEOMETRY > SYMMETRY** and redefine the planes of symmetry as listed on page 5 of this tutorial.



1. BOUNDARY

2. ASSIGN SURFACE

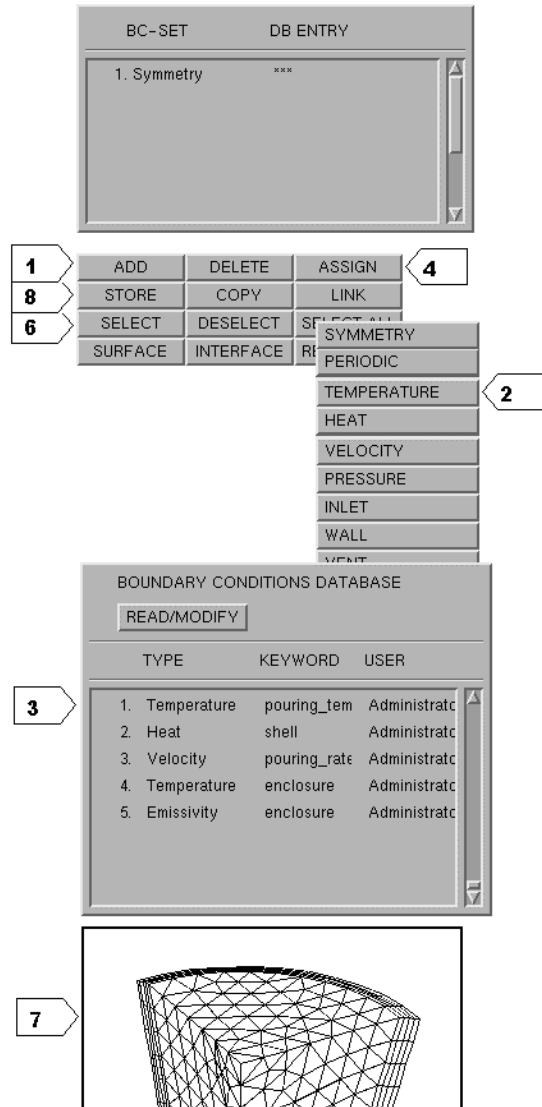
3. Highlight the Symmetry boundary condition

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**.

A temperature boundary condition needs to be defined wherever the metal will be entering the casting model. In this example, we will assume that the metal enters the center of the pouring cup with a stream diameter of two inches. Therefore, the overall area of the stream at the top of the pouring cup is πr^2 , or 3.14 in². However, since only 1/4 of the pouring cup is modeled, the area to be selected is 3.14/4, or 0.785 in². When selecting these inlet nodes for this boundary condition, begin at the corner of symmetry planes and pick the nodes closest to the center until the area indicator in the top-right corner of the screen shows a value close to 0.785.

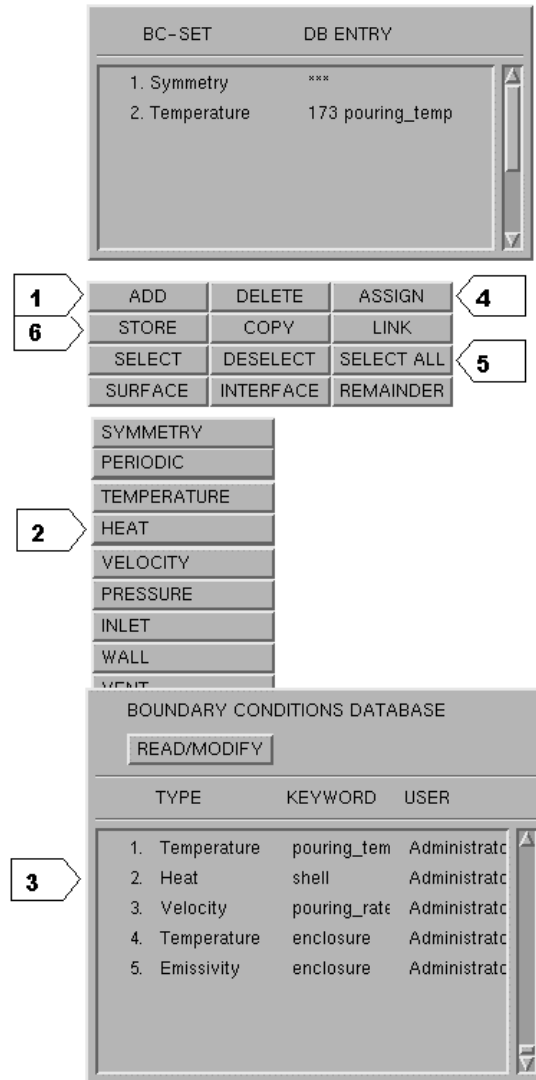
1. **ADD**
2. **TEMPERATURE**
3. Highlight the “pouring_temp” temperature boundary condition entry
4. **ASSIGN**
5. Zoom into the pouring cup for ease of selection using the middle mouse button
6. **SELECT**
7. Pick the nodes for the temperature condition
8. **STORE**
9. Zoom out by clicking the right mouse button in the graphics region



Extra Notes

Now assign the heat boundary condition to the outside of the shell, allowing the shell to radiate and convect heat to the air.

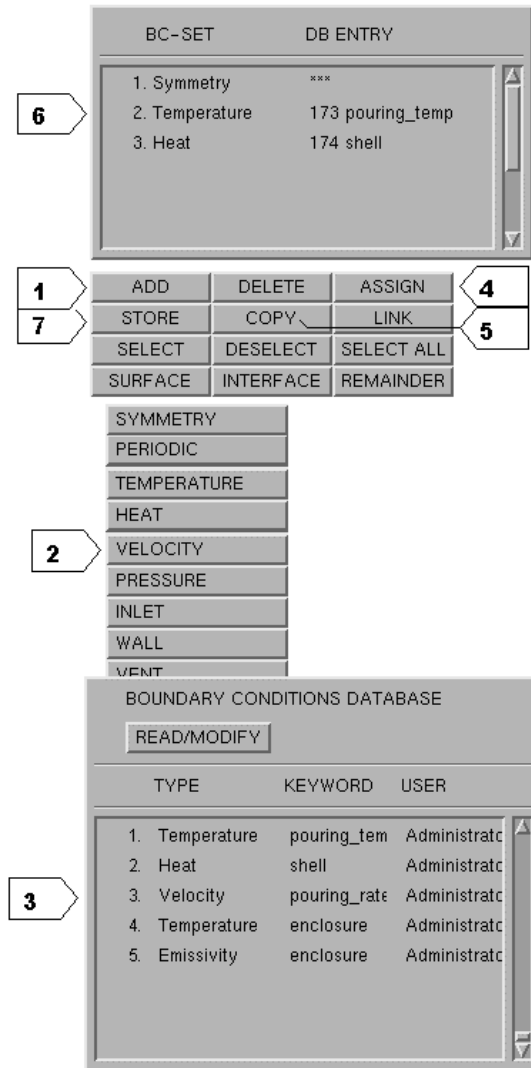
1. **ADD**
2. **HEAT**
3. Highlight the “shell” Heat database entry
4. **ASSIGN**
5. **SELECT ALL**
6. **STORE**



Extra Notes

Finally, add the Velocity boundary condition. Since the Temperature boundary condition occupies the same nodes as the inlet velocity, simply copy the temperature nodes to the velocity condition.

1. **ADD**
2. **VELOCITY**
3. Highlight the “pouring_rate” velocity database entry
4. **ASSIGN**
5. **COPY**
6. Click on the temperature boundary condition
7. **STORE**



Extra Notes

There are other locations where velocity boundary conditions may automatically be applied, based on the interfaces. In this example, we are working with a Coincident Interface mesh. This means that the casting and mold share nodes along their common interface. In **INTERFACE .. CREATE**, we split the two regions to be independent. At the same time, a “No Slip” (all velocities equal 0) condition was automatically placed at all of the nodes in the interface.

When an interface is not created and a fluid solution is performed, this “No Slip” condition must be manually placed on all nodes of the fluid region except where the fluid enters.

Assigning the Enclosure Radiation Conditions

When you assign these enclosure conditions to the enclosure, you assign the conditions to sets of elements on the enclosure. By default, there is one set, which contains all of the enclosure elements. You can have more than one grouping of elements so that different events can occur on different regions of the enclosure.

In the Enclosure Assign form, you see three columns, the Set column, the **E**missivity column and the **T**emperature column. When you assign an item from the database to an enclosure set, the database entry number is placed into the appropriate column for that set.

1. BOUNDARY

2. ASSIGN ENCLOSURE

3. Highlight the “enclosure” temperature database entry

4. ASSIGN

5. Highlight the “enclosure” emissivity database entry

6. ASSIGN

The screenshot shows the 'BOUNDARY' form with a list of options: DATABASE, ASSIGN SURFACE, ASSIGN VOLUME, ASSIGN ENCLOSURE, and INVERSE. Below this is a table with columns ENCL SET, E, and T. The first row contains the values 1, 1, 5, and 4. To the right of the table are buttons: ADD, DELETE, ASSIGN, SELECT, DESELECT, SELECT ALL, STORE, CANCEL, and SURFACE. Below the buttons is a 'BOUNDARY CONDITIONS DATABASE' window with a 'READ/MODIFY' button and a table with columns TYPE, KEYWORD, and USER. The table contains three rows: 1. Temperature pouring_tem Administratc, 4. Temperature enclosure Administratc, and 5. Emissivity enclosure Administratc. Callout boxes 1 through 6 point to specific elements in the interface: 1 points to the BOUNDARY button, 2 points to the ASSIGN ENCLOSURE button, 3 points to the '1' in the ENCL SET column, 4 points to the '5' in the E column, 5 points to the '4' in the T column, and 6 points to the ASSIGN button.

ENCL SET	E	T	
1.	1	5	4

TYPE	KEYWORD	USER
1. Temperature	pouring_tem	Administratc
4. Temperature	enclosure	Administratc
5. Emissivity	enclosure	Administratc

Extra Notes

The emissivity assignment defines the emissivity from the enclosure to the casting and shell. The temperature specifies the ambient temperature of the enclosure.

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. ProCAST also has the ability to handle a rotating gravity direction using the **ROTATE** option. This simulates processes such as rollover.

The screenshot shows the 'GRAVITY' dialog box in ProCAST. On the left, a vertical list of buttons includes 'PROCESS', 'DATABASE', 'ASSIGN ENCLOSURE', 'ASSIGN VOLUME', and 'GRAVITY'. A callout box with the number '1' points to the 'PROCESS' button. A callout box with the number '2' points to the 'GRAVITY' button. The main area of the dialog box contains three input fields: 'X:' with the value '0.000000e+00', 'Y:' with the value '0.000000e+00', and 'Z:' with the value '-9.8'. A unit label 'm/sec**2' is positioned to the right of the Y and Z fields. At the bottom, there are three buttons: 'APPLY', 'ROTATE', and 'CANCEL'. A callout box with the number '3' points to the 'Z' input field. A callout box with the number '4' points to the 'APPLY' button.

1. PROCESS

2. GRAVITY

3. Enter -9.8 as the Z value.

4. 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 are the initial conditions of the simulation. These consist of the initial temperatures (**CONSTANT**) and a designation of the empty cavity (**FREE SURFACE**). The initial temperature designates the pre-heat temperature of the mold.

This example assumes that the metal is poured directly after the mold is removed from the pre-heating device. However, if there is a significant period of time between the removal of the mold from heating and the beginning of the pour, it is quite easy to simulate this initial cooling of the mold. See the Extra Notes section on page 16 of this tutorial for details.

1. **INITIAL COND**
2. **CONSTANT**
3. Highlight Material 1
4. Enter 2750 (pressing “Enter” moves the highlight bar to the next material)
5. Change the Units to F
6. Enter 2000
7. **INITIAL COND**
8. **FREE SURFACE**
9. Toggle the “NO” for Material 1 to “YES” specifying that this material is initially empty.

The screenshot shows the 'Initial Condition Definition' dialog box. It has a tabbed interface with 'INITIAL COND' selected. Below the tabs are three main sections:

- CONSTANT**: A table with columns 'ID#', 'MATERIAL NAME', and 'TEMPERATURE'. It contains two rows: '1. #1 STEEL_Stainless_AISI T 2750.000 F' and '2. #2 REFRACTORY_Fused_Silica 2000.000 F'. Callout 3 points to the first row.
- Value:**: A text input field with a unit dropdown set to 'F'. Callout 5 points to the unit dropdown.
- EMPTY**: A table with columns 'ID#', 'MATERIAL NAME', and 'EMPTY'. It contains two rows: '1. #1 STEEL_Stainless_AISI Type 3 YES' and '2. #2 REFRACTORY_Fused_Silica NO'. Callout 9 points to the 'YES' value for Material 1.

Callout 1,7 points to the 'INITIAL COND' tab. Callout 2 points to the 'CONSTANT' tab. Callout 8 points to the 'FREE SURFACE' tab. Callout 4,6 points to the 'Value:' input field.

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, the maximum time step to 20 seconds per step and stopping the analysis after 30 minutes of simulation time.

1. RUN PARAMETERS

2. GENERAL

- 3.** Change the value of NSTEP to 2000
- 4.** Change the value of DTMAX to 20
- 5.** Change the value of TFINAL to 30
- 6.** Toggle the Unit to “min”
- 7. 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

RUN PARAMETERS

2

UNITS
GENERAL
CYCLES
THERMAL
RADIATION
FLOW

3

INILEV	0	DT	1.00000e-03	sec	HELP	4
NSTEP	2000	DTMAX	20	sec	APPLY	7
NRSTAR	5	TFINAL	30	min	CANCEL	6
NPRFR	1	TMODS	2.00000e+00			5
PRNLEV	0	TMODR	5.00000e-01			
SDEBUG	1	CONVTOL	1.00000e-04			
AVEPROP	0					
CGSQ	0					
LUFAC	1					
DIAG	16384					
NEWTONR	0					
USER	0					

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

For investment casting, there are three ways to stop the analysis. In the Run Parameters General form, NSTEP and TFINAL will halt the analysis. This occurs if the number of steps completed equals NSTEP or if the simulation time reaches TFINAL. In the Run Parameters Thermal form, TSTOP may be used to designate the temperature of which the entire casting region must fall below. Therefore, a value of just below Solidus may be entered, and the solution will stop once the casting has completely solidified.

1

RUN PARAMETERS

UNITS

GENERAL

CYCLES

2THERMAL

RADIATION

FLOW

TURBULENCE

3

4

THERMAL	1	TSTOP	0.00000e+00	K	HELP
MICRO	0	CONVT	1.00000e+00	K	APPLY
TFREQ	5	MOBILE	3.00000e-01		CANCEL
QFREQ	10	TRELAX	1.00000e+00		
MFREQ	10	CRELAX	1.00000e+00		
POROS	1	CLUMP	1.00000e+00		
LINSRC	0				

5

Finally we will modify the parameters that control the fluid analysis. The first item to change will be fluid solution control, FLOW. There are a couple of options for this parameter. 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. The other parameter to be changed is VFREQ. Set this to 5 to save fluid results every 5 steps. Set FREESF to 1 turns on the Free Surface Front solver. Finally, modify COURANT to 100 for a faster solve.

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. APPLY

See next page for illustrations.

Extra Notes

The COURANT parameter is a control on the free surface front to keep it stable and accurate. However, for this type of analysis, the default value of 10 is very limiting and slows the solution. Therefore, setting it to 100 allows the solution to run faster with very minimal loss of accuracy.

1

RUN PARAMETERS

UNITS

GENERAL

CYCLES

THERMAL

RADIATION

2

FLOW

TURBULENCE

STRESS

ELECTROMAGNETICS

INVERSE

CAFE

3

4

5

FLOW	3	COURANT	10		6
VFREQ	5	LVSURF	9.80000e-01		7
FREESF	1	PREF	0.00000e+00	N/m**2	
NNEWTON	0	PLIMIT	1.00000e+20	N/m**2	
HIVISC	0	FLOWDEL	1.00000e+20	sec	
GAS	0	TSOFF	0.00000e+00	sec	
COMPRES	0	CONVV	5.00000e-02		
COUPLED	0	MLUMP	1.00000e+00		
FFREQ	1	ADVCTW	0.00000e+00		
TPROF	1	PRELAX	1.00000e+00		
WSHEAR	0	COARSEC	8.80000e+00		
EDGE	0	COARSEP	3.33000e-01		
PINLET	0				
HEAD_ON	0				

HELP

APPLY

CANCEL

Save and Exit PreCAST

Everything should now be setup for this investment 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

ASSIGNMENTS	POSSIBLE	MADE
Material:	2	2
Interface:	1	1
Boundary:	4	4
Enclosure:	1	1
Moving Solids:	2	0
Micro:	2	0
Stress:	2	0

Warning: investmentd.dat file already exists!

2

CONTINUE

ABANDON

GO BACK

-
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.

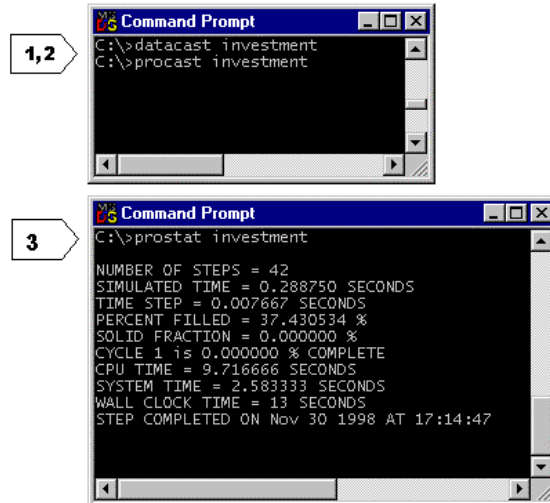
Pressing **ABANDON** simply exits without saving. Use this when browsing the set up or when changes made to the set up are not wanted.

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 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 the casting is 98% full and 100% solid.

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



Extra Notes