

下面是ProCAST2005软件自带操作手册的部分翻译内容，从75页开始，本人E文水平极为有限，翻译内容必有诸多错漏之处，希望各位不要见笑。

# THERMAL

## Thermal model

The Thermal module allows to perform a heat flow calculation, by solving the Fourier heat conduction equation, including the latent heat release during solidification. The typical results which can be obtained are the following :

- Temperature distribution
- Fraction of solid evolution
- Heat flux and thermal gradients
- Solidification time
- Hot spots
- Porosity prediction

热分析

热分析模块

热分析模块执行热流计算，通过傅立叶热传导方程，包含结晶过程的潜热计算。通过分析可以获得如下结果：

温度分布

结晶过程的凝固状态

热流量和热梯度

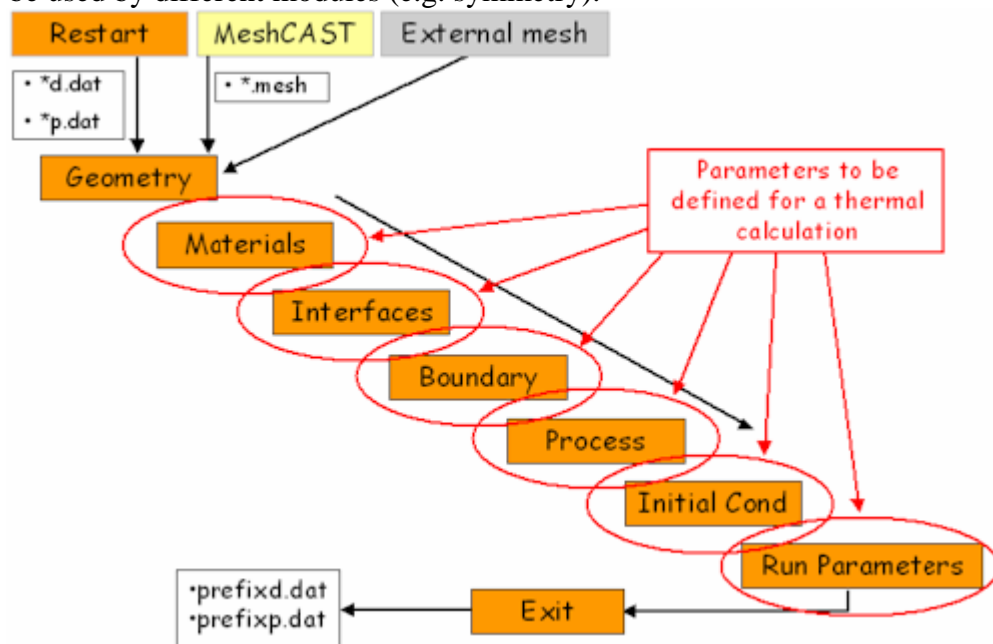
凝固时间

热节

缩松（孔）预测

## Flow chart

This section describes the set-up of a thermal case. It is also the opportunity to introduce the general work flow of the pre-processor, as well as some aspects which may be used by different modules (e.g. symmetry).

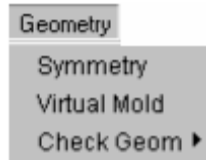


流程图

这张图表描述了热分析案例的设置过程。这也是在 pre-processor 过程通用的设置，在其他模块也同样适用。  
(例如对称性设置)

## Geometry assignments

Once the geometry is loaded, the following operations can be performed on the geometry :



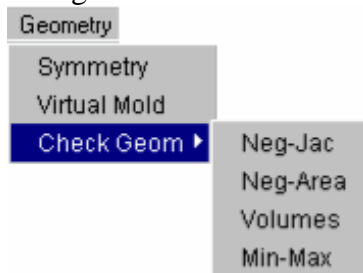
Symmetries can be defined at this stage (see the "Thermal/Radiation" section for more details). The definition of the Virtual mold is also done at this level. (see the "Virtual Mold" section for more details).

几何体分配

当一个几何体打开，下列菜单命令允许被执行

对称性截面可以在此设置。(查看“热辐射”一节以得到详细内容) 虚拟模具也在这里设置(查看“虚拟模具”一节以得到详细内容)

In the "Check Geom" menu, the following features are accessible :



Neg-Jac (negative Jacobian) and Neg-Area correspond to problems in the mesh. These buttons allow to locate where these problems are in the mesh in order to give indication where to modify the mesh in MeshCAST.

Volumes gives access to the volume of each material domain, whereas Min-Max indicates the dimensions of the model.

在“检察几何体”菜单，有如下项目：

负雅可比单元和负雅可比区域指出了网格的错误。这两个命令指出网格的问题在哪里，以便通过 MeshCAST 进行修复。

通过体积命令可以查看指出不同区域的体积大小

Min-Max 可以查看模型各方向的尺寸。

## Materials assignment

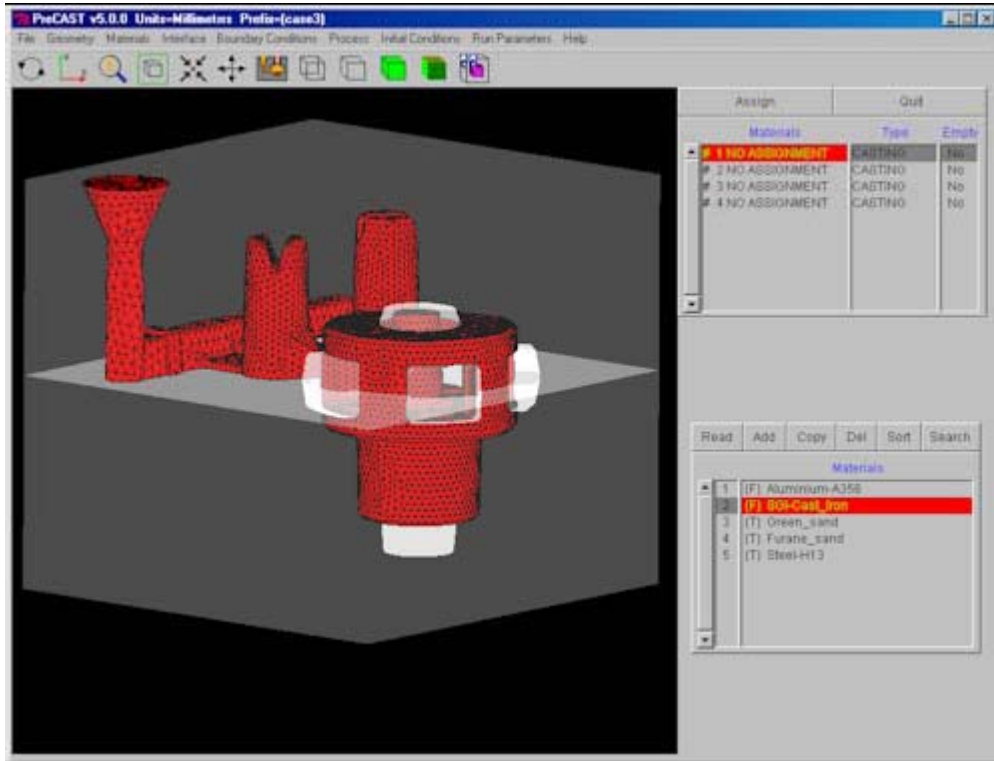
Once the model is loaded (see the "Geometry import" section), the first operation is to define the different materials with their properties and attributes. This is performed in the Material/Assign menu.



## 材料分配

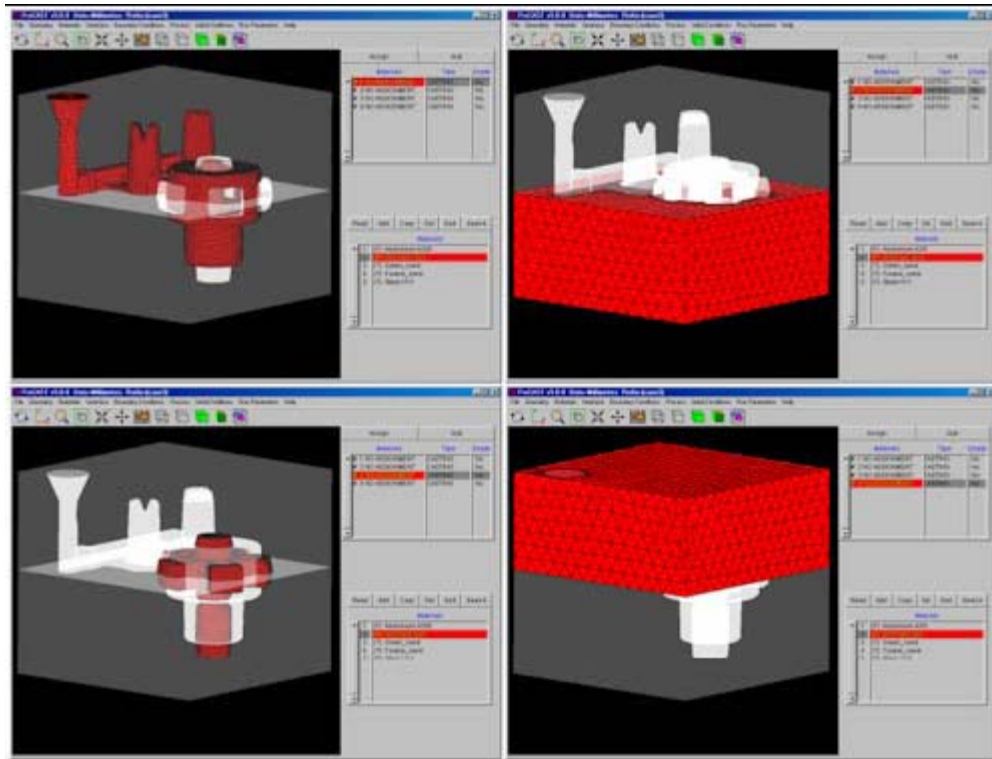
当模型调入后，第一件事情就是为各部分分配材料，通过执行 **Material/Assign** 菜单来执行。

On the right of the window, two frames are shown. The top one contains the material list (or domain list), whereas the bottom one corresponds to the material database.



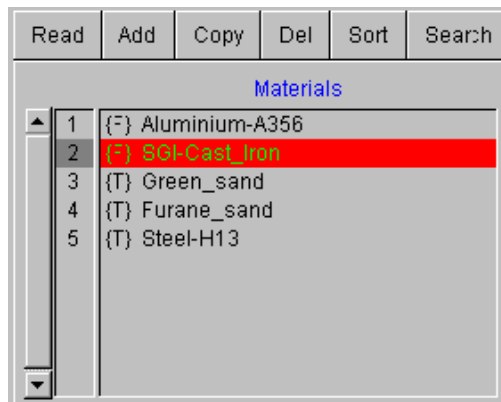
When one clicks on the different materials in the material list, the corresponding domains are highlighted (in the picture below, the hidden mode with mesh was selected - see the "Geometry manipulation" section for the other display modes).

在下图的右侧有两个方框，上面的内容为材料列表（或者叫区域列表），下面的是材料库。当点击材料列表中的某一种材料时，图中相应的区域会高亮显示。（在下面的图中显示为隐藏线模式。查看“几何操作”章节得到更多详情）



In the lower frame ("Material database" list), the list of all available material properties in the material database is displayed. To manage the database entries, please refer the "Databases" sections. The {T} or {F} which are indicated before the material name are telling whether material properties are present in this material for Thermal only calculations (T) or for Thermal and Fluid flow calculations {F}. If a {\*} appears, it means that the material properties definition is incomplete and that this material entry can not be used for a calculation at this stage.

在下面的方框中（材料库列表），软件提供的所有可用的材料均显示在其中。对材料库的详细操作，请参阅“数据库”章节。显示在材料名字之前的{T} 或 {F}表示这种材料时只用于热分析计算{T} 或热分析和流场分析 {F}。如果{\*}显示在名字之前，表示材料还没有完全定义，不能用于当前的分析计算。



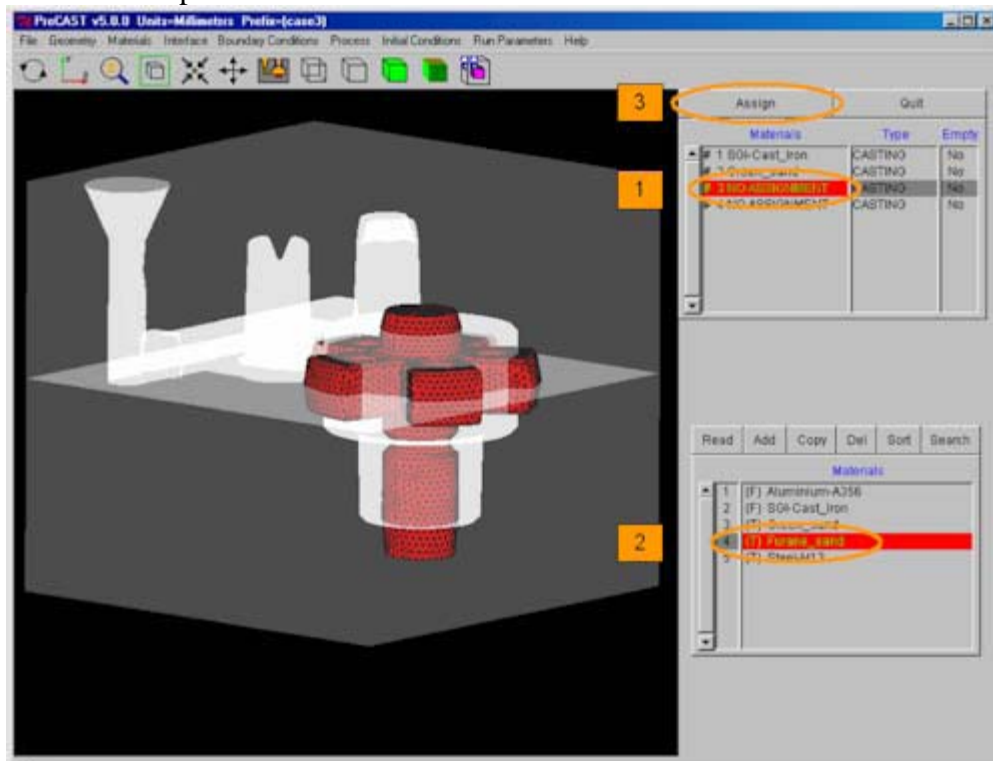
In the top frame ("Domain list"), all the domains (or materials) present in the mesh are listed. When a mesh is loaded, it appears as follows :

在顶上的方框中（区域列表），网格中所有的区域（或材料）都会显示出来，当网格文件打开后，会按如下显示：

Assign		Quit
Materials	Type	Empty
# 1 NO ASSIGNMENT	CASTING	No
# 2 NO ASSIGNMENT	CASTING	No
# 3 NO ASSIGNMENT	CASTING	No
# 4 NO ASSIGNMENT	CASTING	No

Then, one should assign a Material to each domain, to define the type of each domain and to specify whether the domain is empty or not at the beginning of the calculation.

To assign Materials, (1) one should select the desired domain in the upper list, (2) select the desired material in the material database list, and (3) click on the Assign button. This should be repeated for each domain.

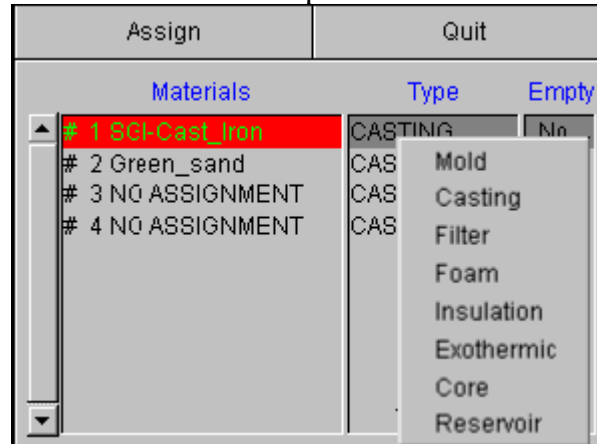


为不同的区域分配材料时，要为相应区域指定材料种类，同时还要指定在计算开始时哪一部分为空腔。

分配材料步骤：（1）首先上面的列表中选择要设定的部分（2）在材料库中选择要设定的材料（3）点击 Assign按钮。重复此步骤以次设定所有的材料。

Then, the "Type" of each material should be defined. To do so, make a right click

on the "CASTING" word and the available list of possible selection will appear :



此外，各种材料在整个铸型中的种类也需要定义，在"CASTING"上右击鼠标，就会显示出可以设定的选项：

**Mold** : the mold material should be set to "Mold". This will be used for cycling calculation (in die casting) in order to allow the calculation of the heating of a die during cycling (i.e. the temperature of the mold domains will not be reset to the initial temperature at the beginning of each cycle).

**Casting** : the casting material should be set to "Casting". This setting is necessary in particular for all the domains where fluid flow will occur. For a cycling calculation, the casting domains initial temperatures will be reset at the beginning of each cycle.

**Filter** : filter domains should be set with the "Filter" type (see the "Filters" section for more details).

**Foam** : for lost foam calculations, the domains where the foam is present at the beginning of the calculation should be set to "Foam". Of course, during the filling, the casting material will replace the foam, as it burns.

**Insulation** : this type has no specific effect on the solver. It will correspond to a "Mold" type of material. At this moment, this is for information purposes.

**Exothermic** : this will activate the Exothermic properties of the sleeve (if they are defined in the corresponding material properties). If the material properties are containing the exothermic information, but the "Exothermic" type is not activated, the exothermic model will not be activated (see the "Exothermic" section for more details).

**Core** : a core type material should be defined in the case of cycling, where cores are placed into the mold at each cycle. This means that unlike mold materials, the initial temperature of the cores will be reset at the beginning of each cycle.

**Reservoir** : a Reservoir type material is a domain where the free surface will always be perpendicular to the gravity. This allows to simplify the free surface computation and it is especially useful in the case of tundish modeling. The "RESERVOIR" domains should have an "EQUIV" interface with the other CASTING materials.

Please note that if there is more than one RESERVOIR domain, they should all touch each other and they should be all full at the beginning. Otherwise, problems may be encountered.

If one want to empty the reservoir, no special BC should be specified. If one

would like to keep the reservoir full, a pressure BC should be set at the surface of the reservoir.

铸型：属于铸型应设为"**Mold**"。这将应用于循环计算。（永久型铸造，例如压铸、金属型等）计算铸型在反复浇注时被加热的情况。

铸件：属于铸件的部分应设为"**Casting**"，在循环计算时，铸件区域在每一个循环开始时会被设为初始温度。

过滤器：属于过滤器的部分应设为"**Filter**"。（关于过滤器的详细内容请参阅"**Filters**"章节）

泡沫：应用于消失模铸造，浇注开始时属于泡沫部分得应设为Foam，当然在浇注进行时，泡沫会被烧掉，金属液会取代泡沫原来得位置。

绝热（保温）层：这种属性不会对计算结果产生特殊得影响，在某种意义上，它相当于“模具”类型。（**At this moment, this is for information purposes.这句话我也不知道怎么译**）。

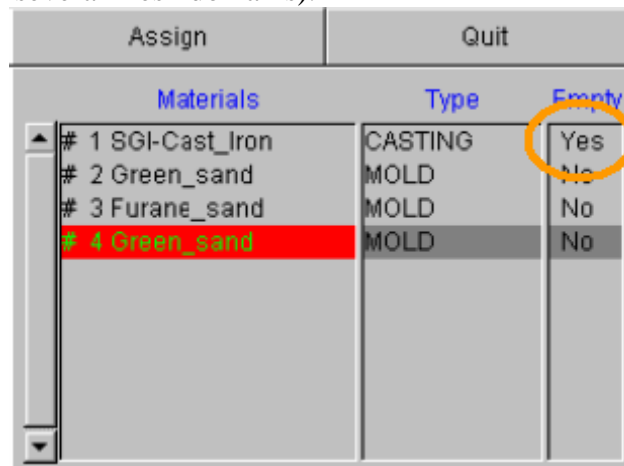
发热剂：设置这项属性将会激活发热冒口得发热属性，（如果设定对其设定了相应的材料属性）如果材料属性包含了发热属性但是没有定义为“**exothermic**”类型，则发热模式不会被激活。（查看“**Exothermic**”章节以得到更多信息）

砂芯：砂芯类型主要应用于循环模拟计算。砂芯在每次浇注前放入铸型，这就意味着此类型不同于“**Mold**”类型，在每个浇注循环的开始时刻，此类部位的温度会被初始化为开始设定的值，而不会随着反复浇注升高。

蓄液池：蓄液池是自由表面始终垂直于重力的区域，允许设定简单的自由表面，经常应用于漏斗形状，设定为“**reservoir**”的区域在设置“**interface**”时应与铸件设定为“**EQUIV**”。需要注意如果设置了多个蓄液池区域，则它们必须互相连接而且在计算开始时它们必须是充满的。否则会出现问题。

如果希望腾空蓄液池，则不需要设置相应的边界条件，反之，如果要保持蓄液池一直处于充满的状态，则需要在相应的表明设置一个压力边界条件。（这一段，我看完也不知道蓄液池到底是什么东东，我猜是不是压铸的活塞前存金属液的位置）。

Finally the user has to specify which domains are empty at the beginning of the calculation (for mold filling calculations). One should make a left click on the "No" to turn it to "Yes" (which means that Yes the domain is empty). On additional click returns to No. Of course, more than one domain may be empty (if the casting is made out of several mesh domains).



When one makes a right click on the material name in the upper list, the lower list is pointing on this particular material. This is very useful if one wants to see the material properties of this material.

最后需要那些区域在计算开始时（也就是充型开始）哪些区域是空的。在其上单击可以转换“**No**”和“**Yes**”（“**Yes**”就表示是空的），当然也有可能多个区域是空的。（如果铸件是由多个网格组装起来



就会产生此种情况)

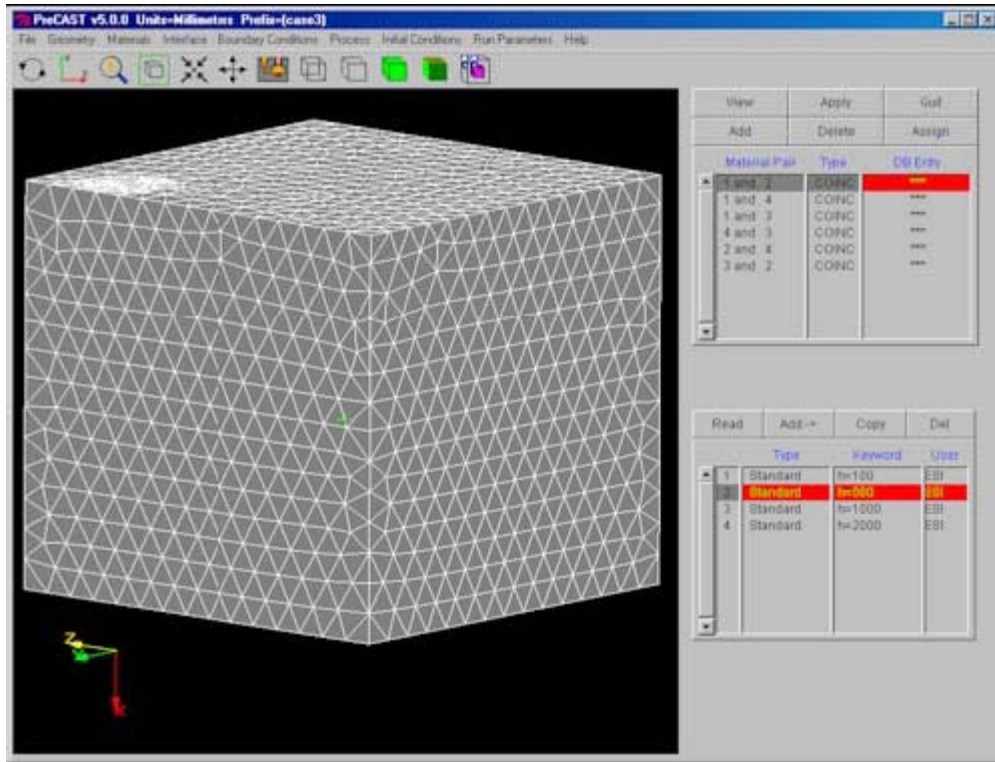
当单击上面材料列表中的某一种材料时，下面的库列表就会加亮显示此种材料，如果要详细查看材料的属性，这种方法可以帮您快速的定位。

## Interfaces assignment

Once the Materials are defined, one should define the Interfaces, with the Interface menu.



As for the Material window, two frames appear on the right of the window. The top one contains the list of all the possible interfaces, whereas the lower one shows the Interface database. To manage the database entries, please refer the "Databases" sections.



材料定义完成后，下一步应该进行界面设定。同材料设定窗口一样，在窗口的右侧有2个方框，上面的显示了目前可能存在的所有界面对，下面的时界面导热系数的数据库。如果要对数据库进行操作，请查看数据库章节。

Firstly, one should define the type of interfaces in the upper right window



View	Apply	Quit
Add	Delete	Assign
Material Pair	Type	DB Entry
1 and 2	COINC	***
1 and 4	COINC	***
1 and 3	COINC	***
4 and 3	COINC	***
2 and 4	COINC	***
3 and 2	COINC	***

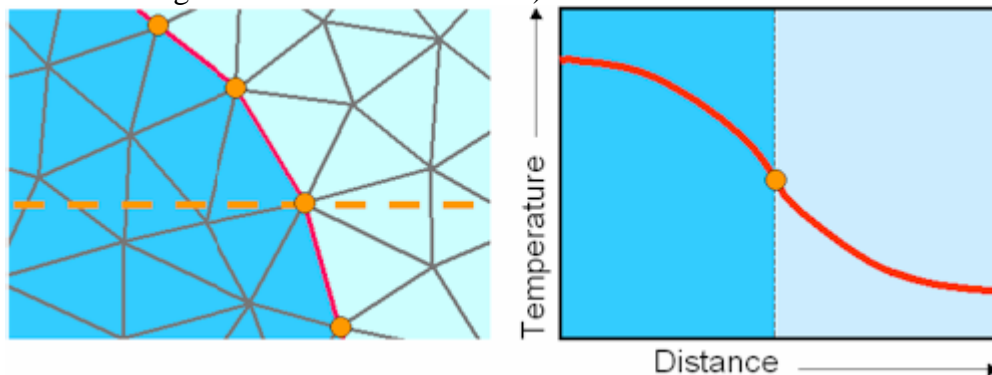
On the left, the "Material Pair" are shown. "1 and 3" means that there is an interface between material 1 and material 3. By default, the Type of the interface is set to "COINC". By clicking on the "COINC" text, one can toggle between "COINC", "NCOINC" and "EQUIV".

首先，要在右上角的方框中设定界面的类型。

在方框的左边显示者材料对，"1 and 3"表示在材料1和材料3之间存在一个界面。默认情况下所有的类型会被预设为"COINC"，在其上单击可以在"COINC", "NCOINC" 和 "EQUIV"之间切换。

### EQUIV option

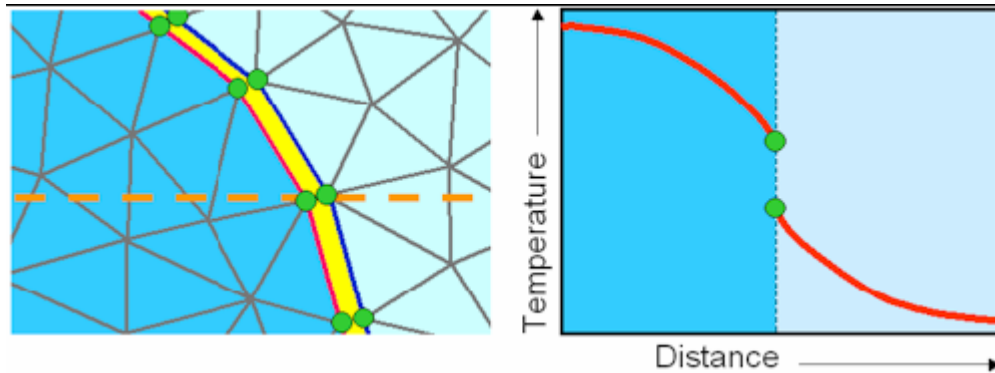
When two domains are part of the same entity (i.e. they both belong to the casting with the same material properties, but they were meshed separately for technical reasons), one will set an "equivalenced" interface between them (EQUIV). It means that there will be a continuum between the two domains, with a continuous temperature profile across the interface, as well as continuous velocity field. In such a case, the nodes at the interface (shown in orange in the figure below) are shared by the elements on both sides. This EQUIV option can also be used if one has different materials in the two domains, but the materials are welded together (i.e. with a total bounding between the two materials).



当两个区域是同种材料一个实体时（比如他们拥有同种材料都属于铸件，但是由于技术上的原因在划分网格时被分开），则在他们之间应该设置为“等价”的界面关系。设置为**EQUIV**表明两个区域是连续的，同时拥有连续的温度场和速度场。在这种情况下，界面上的节点（途中橙色的部分）同时属于界面上的两种材料。**EQUIV**选项也可用于虽然材料不同但焊接在一起的两个区域。

### COINC option

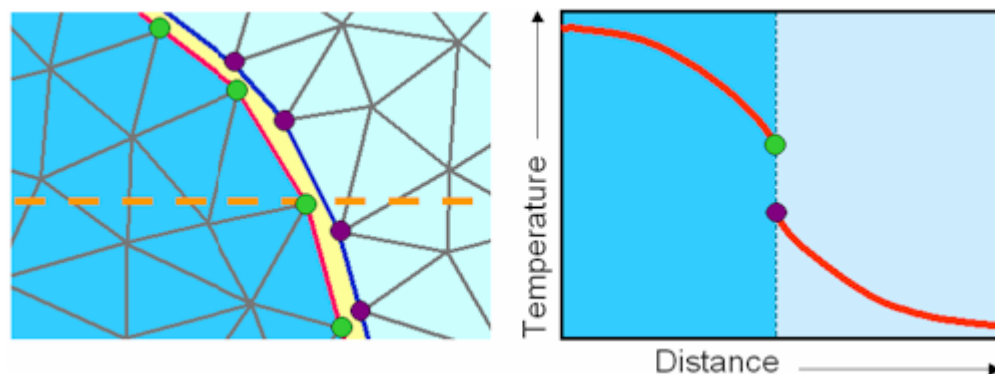
At an interface between two different materials, such as the casting and the mold, there is usually a temperature drop. In this case, the nodes at the interface should be doubled (for a coincident interface), in order to distinct temperature on each side of the interface. As during the mesh generation, there is one node at the interface, it is necessary at this stage to duplicate all the interface nodes (as shown in green in the figure below). This duplication operation is performed when "COINC" is selected (for "coincident nodes"). The interface, which is shown in yellow in the figure below has in fact a zero thickness.




当界面两边的材料不同时，比如说铸件和铸型之间，通常会有一个温度差。在这种情况下，为了区别界面两边的温度，界面上的节点应该是双倍的（在一致的界面上）。在划分网格的过程中，界面上只有单倍的节点，所以必须对界面上所有的节点进行复制（图中显示为绿色的节点）。这个复制操作就依靠设置为"COINC" 类型来实现。图中黄色的部分（也就是界面）实际厚度为零。

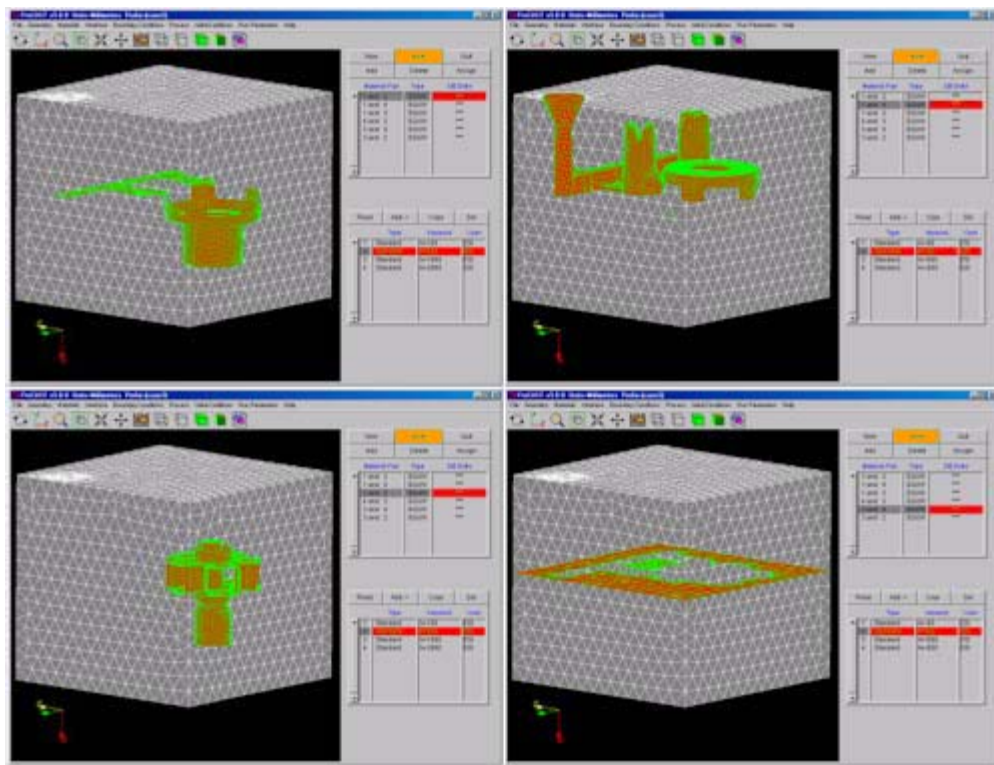
### NCOINC option


It is also possible to generate a non-coincident mesh (i.e. where the elements on both sides of the interface are not matching, which means that they are not sharing the same nodes), by adding different meshes together (see the "Advanced features" section for more details on non-coincident meshes). In this case, one has to specify that the interface is non-coincident, with the "NCOINC" option.

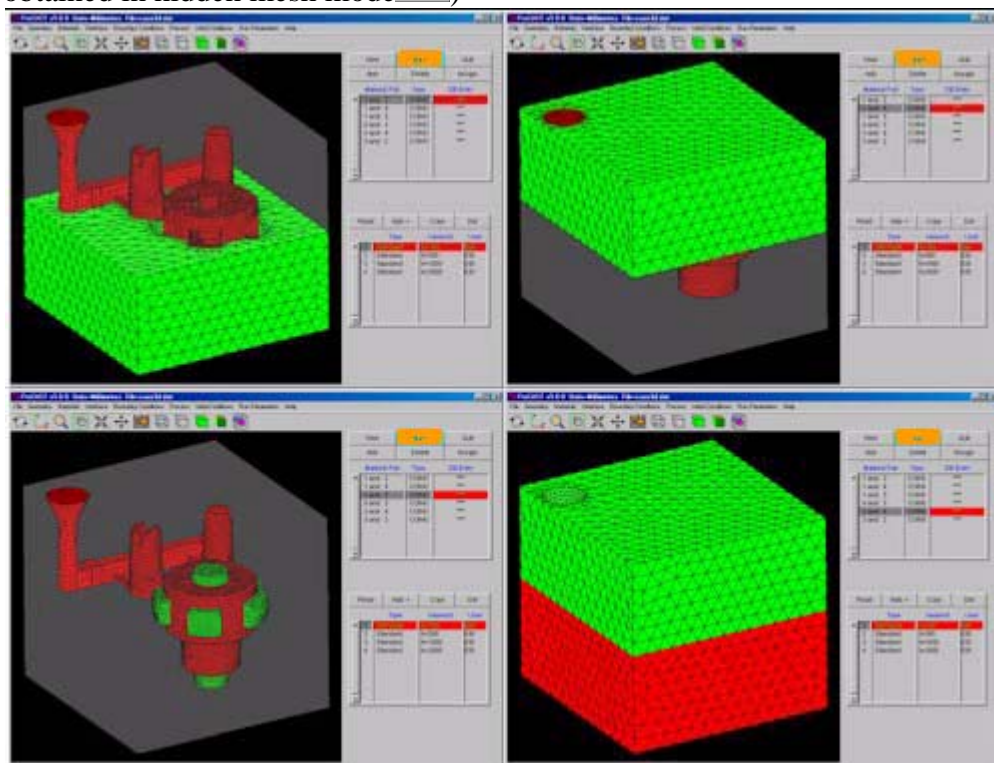


也有可能会出现不一致的网格，（例如界面两侧的元素不匹配，它们之间不能共用节点）将不同的网格文件组合在一起时可能会出现这种情况，（查看“高级特征”章节得到更详细的信息）在这种情况下，需要将界面设定为"NCOINC"选项。

When one toggles between the different options, the interface appears in red and green. It is thus possible to well identify whether it corresponds well to the desired interface (see figure below, which was obtained in hidden mesh mode )




An other way to view the desired interfaces is to click on the "Material Pair" and the material on both sides will be highlighted in red and green respectively - the first material in the list is in red and the second one in green (see figure below, which was obtained in hidden mesh mode )



当在不同的项目间切换，屏幕上会以红绿色显示出界面，可以直观的观察出界面（图中是在实体模式

下显示)

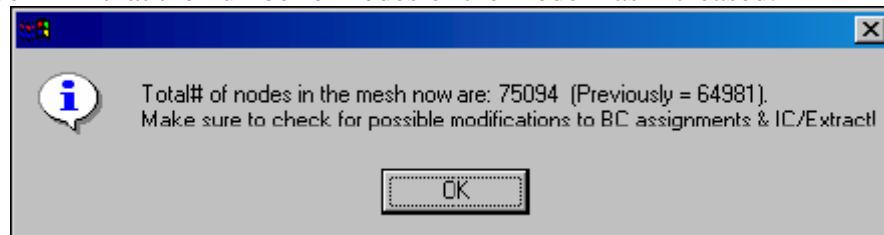
另外一种方法是通过单机材料对下的选项，则相应的界面两边的材料会以红绿色显示，排在前面的为红色，后面的为绿色（看下面的图，图以实体模式显示)

Once the desired selections (between COINC, EQUIV and NCOINC) are done for each possible interface, the STORE button (which is highlighted in orange) should be pressed.

View	Apply	Quit
Add	Delete	Assign
Material Pair	Type	DB Entry
1 and 2	COINC	***
1 and 4	COINC	***
1 and 3	COINC	***
4 and 3	COINC	***
2 and 4	COINC	***
3 and 2	COINC	***

当选定了界面种类后，（在COINC, EQUIV 和 NCOINC之间切换）需要点击Apply按钮（加亮为橙色的）

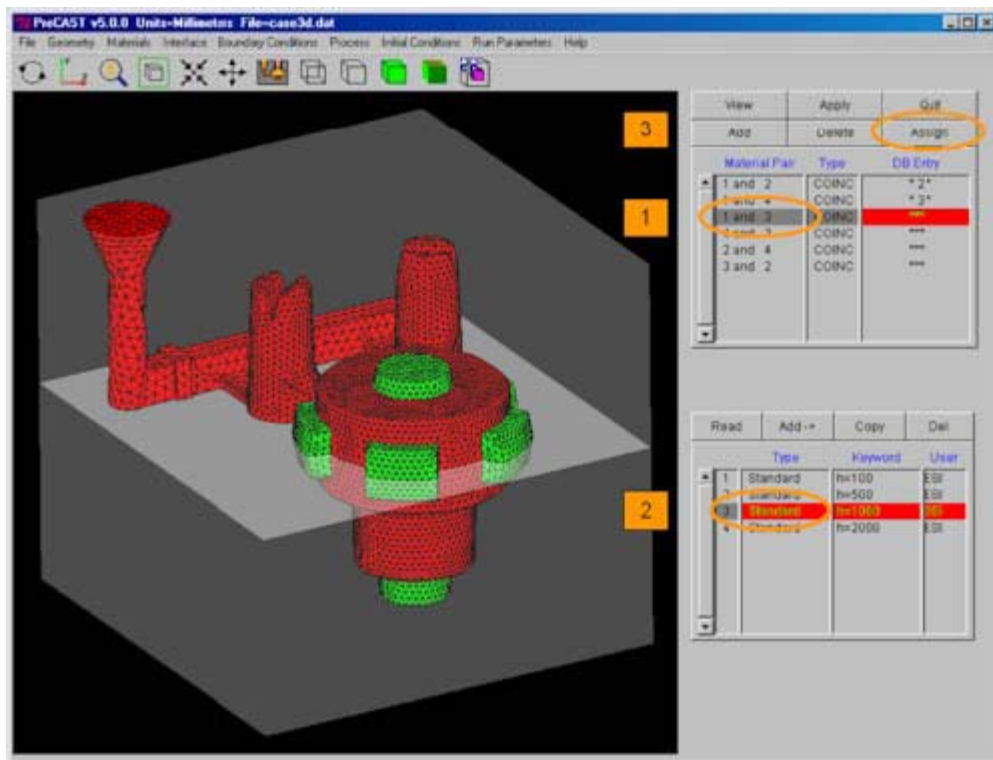
Then the pre-processor will automatically create the double nodes and a message will appear to confirm that the number of nodes of the model has increased.



As it is now possible in version 2005.0 to go from EQUIV to COINC and viceversa, the user should be careful that as nodes were duplicated or removed, some boundary condition assignments may be corrupted, as well as the extracted initial conditions. Thus, in this case, they should be re-assigned.

执行了以上操作后，程序会自动创建双倍的节点并弹出一个信息窗口，窗口中显示出现有及原有的节点总数。

Once the types of interfaces are defined, one has to apply the corresponding heat transfer coefficients (for COINC and NCOINC only, as nothing as to be specified for EQUIV). To assign Interface heat transfer coefficients, (1) one should select the desired Material Pair in the upper list, (2) select the desired interface heat transfer coefficient in the interface database list, and (3) click on the Assign button. This should be repeated for each coincident or non-coincident interface.



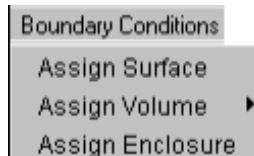
当界面的种类选定后，需要为其指定合适的界面导热系数，（为COINC及NCOINC模式，EQUIV模式不需要指定）分配界面导热系数按下图操作（1）选择要设置的材料对；（2）在数据库中选择相应的值；（3）单击Assign按钮。重复上述过程为每一个界面设定合适的值。

For non-coincident meshes, it is possible to have access to the non-coincident tolerances with a right click on the NCOINC label. A window will open with the two tolerances.

对于非一致的网格，可以通过在NCOINC标签上右击来设置偏移量，点击后会弹出一个设置窗口。

## Boundary conditions assignment

After the definition of the interfaces, the Boundary conditions should be specified. This is done in the "Boundary Conditions" menu.



Three types of boundary conditions can be applied :

- Surface boundary conditions ("Assign Surface"), which correspond to all the conditions applied to the outside of the model or the outside of a given material domain. This is the most commonly used type of boundary conditions.
- Volume conditions ("Assign Volume"), which corresponds to conditions which are applied in a whole volume (e.g. volumetric heat or mass source).
- Boundary conditions assigned to Enclosures ("Assign Enclosure") in case of radiation problem (see the "Radiation" section for more details).



## 边界条件设置

界面导热系数设置完成后，下一步需要设置边界条件，下面是“边界条件”菜单  
在菜单中可以看出共有三大类边界条件设置。

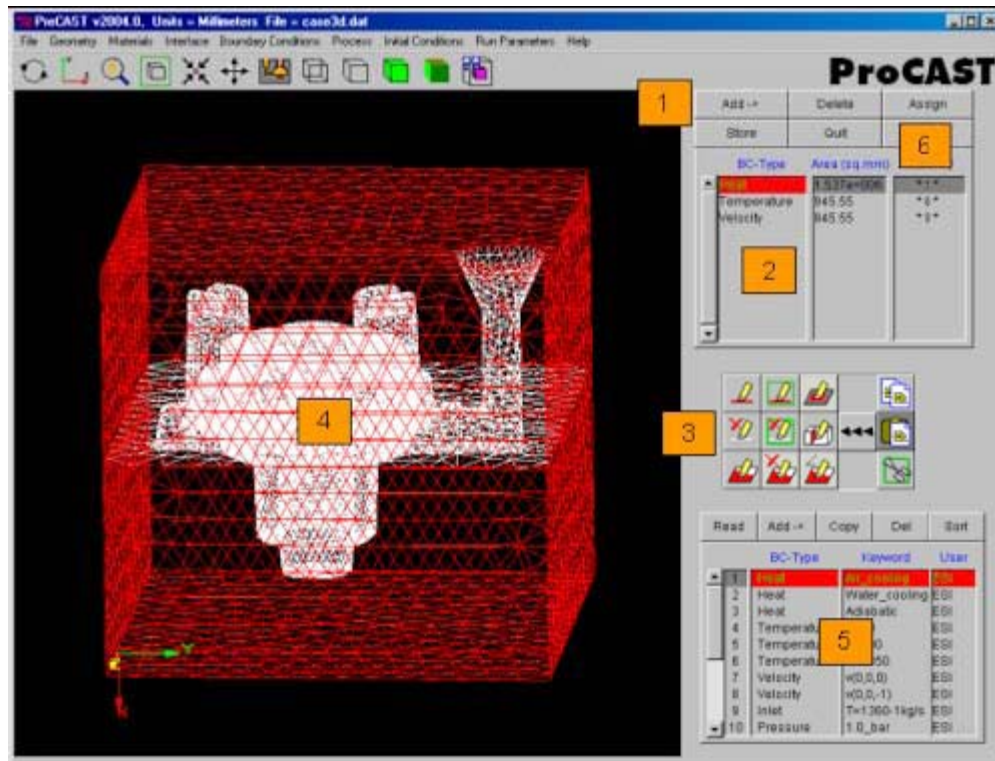
表面边界条件（Assign Surface），用来设定模型（此处指几何模型，而不是单指铸型）或给定材料的外表面，这是最常用的边界条件设置类型。

体积条件（Assign Volume）为整个体积来设置条件（括号里的东东我也不知道是什么意思）

外壳条件（Assign Enclosure）用来解决辐射问题（查看“热辐射”以了解更多相关信息）

## Assign Surface

The principles of Boundary conditions definition are described in the following figure.



Firstly, the desired boundary conditions should be "Added" (1) in the upper list  
(2). With the "Add ->" button, one has the following choices :





The boundary conditions corresponding to Thermal problems are "Symmetry", "Periodic", "Temperature" and "Heat".

Then the "location" where the boundary condition should be applied on the geometry should be specified. The selection tools (3) allow to "paint" the desired area on the geometry (4).

The values to be assigned to the boundary conditions should be selected in the database (5) and then they are assigned to the corresponding boundary condition (6). See the "boundary conditions database" section for more details about the different type of boundary conditions, as well as the database management.

It is possible to have a quick access to the database entries in the following way.

When one makes a right click on the boundary condition entry in the upper list (2), the lower list is pointing on this particular boundary condition entry in the lower list (5).

#### 表面分配

设定边界条件的步骤大致均如下图:

首先通过点击右上方方框中(2)的Add(1)按钮增加需要设定的边界条件类型, 点击后会弹出如下  
的下拉式菜单, 选择要设置的类型。

应用与热分析的边界条件类型有对称性、周期性、温度和冷却条件。

通过使用选择工具(3)设定相应的边界条件应用与模型的什么位置。

对相应边界条件设定的值在下面的数据库中选取(5)并分配给相应的边界条件(6)。查看边界条件数据库可以了解各种边界条件以及对数据库的操作。

通过在上面列表中右击某一种边界条件(2)则下面列表中(5)对应的条件会加亮显示在列表中, 用这种方法可以快速的找到数据库中的值一般查看和修改。

The "Selection tools" allow to "Select", "Deselect", "Propagate", "Clip", "Copy" and "Paste" faces or nodes on the geometry.



For some boundary conditions, such as "Heat" or "Symmetry", faces of the Finite Element Mesh are selected. For other boundary conditions, such as "Temperature", "Velocity", ... the boundary conditions are applied on nodes. The choice between faces or nodes is automatic.

选择工具包括下图所示:

有些边界条件如Heat或Symmetry需要选择单元的表面, 另外一些如Temperature、Velocity等需选择节点, 在使用选择工具使系统会自动根据所设定的边界条件选择选取类型。



Selection of individual faces or nodes. 选择单个的表面或节点



Deselection of individual faces or nodes. 取消选择单个的表面或节点



Select all. 选择所有



Deselect all.取消全部选择



Select and propagate. All the faces or nodes which have an angle with the neighbors smaller than the specified propagation angle will be selected.

选择并扩展选择, 选择节点及附近与该节点所在面角度小于指定值的节点均会被选择



Deselect and propagate. All the faces or nodes which have an angle with the neighbors smaller than the specified propagation angle will be deselected.

上一项的反操作



Definition of the propagation angle.定义扩展角度值

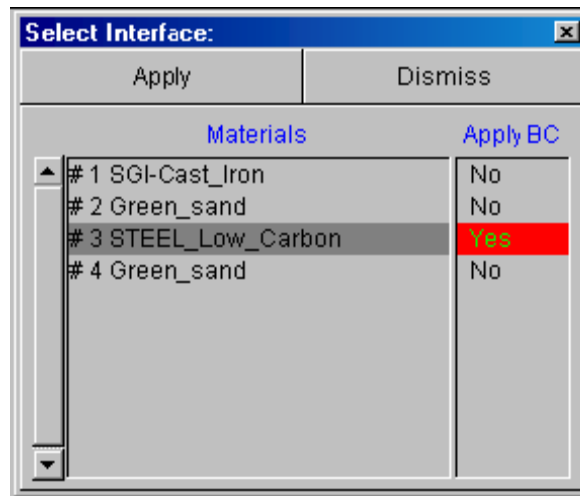


Select remainder. The remaining faces or nodes which have not yet been selected are selected.

选择没有所有其它没有被选择到的节点



Select interface. For "Heat" boundary conditions, the selections are applied on external faces only. However, for cycling, one would like sometimes to apply a Heat BC on faces which lie at interfaces. This allows to select those interfaces automatically. In this case, the following panel is opened. It is proposing the list off all active materials. One could select one or more material and all the faces of the selected materials which are lying on an interface will be selected.



选择界面，一般来说，Heat边界条件只应用与外表面，但是对于类似金属型循环浇注来说，Heat边界条件也可以应用在界面上，应用这个按钮，可以自动选择界面。这是会弹出如下的窗口，可以选择其中的一种或多种材料，则这些相关的界面会被选择。



Button to clip a model. This allows to perform a selection inside a model, where is it not accessible from the outside.

对模型进行切除，这样做使为了选择一些不易选择的杳晃拐角的位置。



Button to "Backtrack" the clip.返回切除前的状态



Copy of selection. All the nodes or faces which are applied on the geometry for the active boundary condition will be copied in the memory.

复制当前所做的选择



Paste of selection. The copied selections (above button) can be pasted on a different boundary condition (i.e. on the active boundary condition when the Paste button is pressed). Please note that one can not copy the node selection of a Temperature BC to a Heat BC (as faces are expected).

粘贴刚才所复制的选择内容。注意，不能把对温度条件的选择应用与冷却条件。

未完待续.....