

PROCAST USER MANUAL - VERSION 500	3
INTRODUCTION	4
SOFTWARE CAPABILITIES	4
SOFTWARE ORGANIZATION	5
USER MANUAL PRESENTATION	7
GETTING STARTED	8
SOFTWARE PRINCIPLES	8
PROBLEM SET-UP	9
CALCULATION	10
RESULTS DISPLAY	11
SOFTWARE MANAGER	12
FILE MANAGER	13
MODULE CALLS	16
SOFTWARE CONFIGURATION	21
PRE-PROCESSING	24
INTRODUCTION	25
GEOMETRY IMPORT	29
THERMAL	31
GEOMETRY ASSIGNMENTS	31
MATERIALS ASSIGNMENT	31
INTERFACES ASSIGNMENT	35
BOUNDARY CONDITIONS ASSIGNMENT	40
PROCESS CONDITIONS ASSIGNMENT	40
INITIAL CONDITIONS ASSIGNMENT	40
RUN PARAMETERS ASSIGNMENT	40
FLUID FLOW & FILLING	42
RADIATION	45
STRESS	51
DATABASES	52
RUN PARAMETERS	53
ADVANCED FEATURES	54
GEOMETRY MANIPULATION	56
RUN OF THE CALCULATION	60
SOLVER	61
TROUBLESHOOTING	62

RESULTS VIEWING	63
INTRODUCTION	64
FIELD SELECTION	65
DISPLAY PARAMETERS	66
GEOMETRY MANIPULATION	67
CURVES	68
RESULTS ANALYSIS	69
DEFECTS ANALYSIS	70
POROSITY	71
CRITERION FUNCTIONS	72
RESULTS EXPORTS	73
INTRODUCTION	74
FORMATS	75
TIPS & TRAPS	76
TUTORIALS	77
HPDC	78
LPDC	79
GRAVITY-SAND	80
GRAVITY-DIE	81
INVESTMENT	82
TILT	83
FILTER	84
STRESS	85
LOST FOAM	86
THIXO	87
APPENDICES	88
SOFTWARE INSTALLATION	89
WINDOWS	89
UNIX - LINUX	89
FILES STRUCTURE	90
FILE NAMES	90
INPUT FILE STRUCTURE (D.DAT FILE)	90
INPUT FILE STRUCTURE (P.DAT FILE)	90
MODELS & ALGORITHMS	91
PHYSICAL MODELS	91
NUMERICAL MODELS	91

ProCAST User Manual - Version 500

Introduction

Software capabilities

ProCAST is a software using the Finite Elements Method (FEM). It allows the modeling of Thermal heat transfer (Heat flow), including Radiation with view factors, Fluid flow, including mold filling, Stresses fully coupled with the thermal solution (Thermomechanics). Beside that, it includes also microstructure modeling and porosity modeling.

Special models are included in order to account for thixo casting and lost foam. Specific features are included to account for processes such as high pressure die casting, centrifugal, tilt. Finally, customized models for foundry processes, such filters, sleeves are included.

Figure under
construction

Thermal calculation

Figure under
construction

Fluid flow calculation

Figure under
construction

Mold filling calculation

Figure under
construction

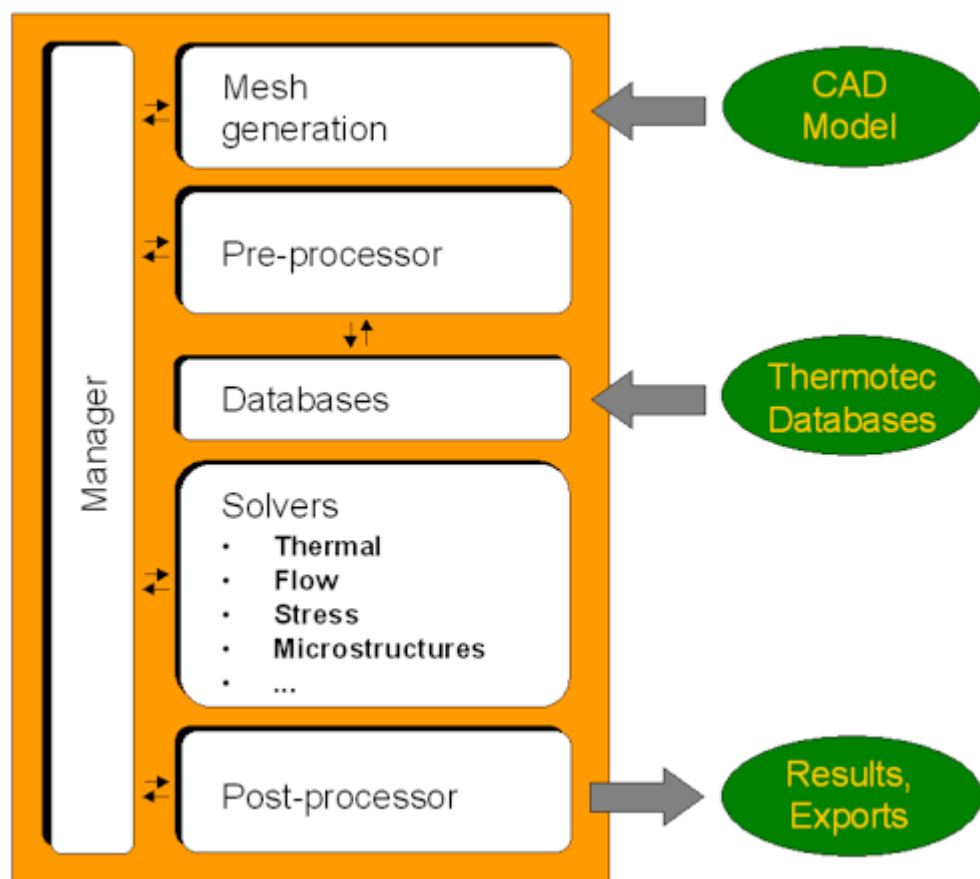
Thermomechanical calculation

Software organization

The software is organized around a Manager, which calls the different modules :

- MeshCAST : the mesh generator
- PreCAST : the pre-processor, coupled with databases
- DataCAST / ProCAST : the solvers
- ViewCAST : the post-processor and data export unit

The following figure is presenting the structure of the software. First, the geometry, in the form of a CAD model is loaded into **MeshCAST**, to generate a FEM mesh. Then, the calculation is configured in **PreCAST**, the Pre-processor. **PreCAST** is linked to **Databases** and in particular with the Thermotec databases for the automatic determination of the material properties from thermodynamic databases. Before the solver **ProCAST** is launched, a "data conditioner" named **DataCAST** is run. Finally, the results can be viewed or exported (for further processing) in the Post-processor **ViewCAST**.



The ProCAST solvers are divided in "Physical modules" with the following capabilities :

Thermal module

- Heat conduction (Fourier equation)
- Latent heat release during solidification
- Cycling in die casting
- Sleeves (insulating and exothermic)
- Non-coincident meshes
- Solidification time
- Secondary Dendrite Arm spacing
- Porosity indicator

Radiation module

- Net radiation method
- Full view factors capabilities
- Mirror and rotational symmetries
- Relative motion of materials
- Solid or surface enclosures

Fluid flow module

- Navier-Stokes equation
- Penalization of the flow in the mushy zone and in the solid
- Mold filling algorithm, with free surface
- Filter model
- Newtonian and Non-Newtonian flow
- Thixo casting models
- Lost Foam model
- Tilt pouring
- Centrifugal casting
- Turbulent models

Stress module

- Elastic, Elastic-plastic, Elasto-visco-plastic
- Rigid or vacant materials
- Automatic calculation of the air gap heat transfer
- Contact algorithm between the different materials
- Contact pressure
- Die Fatigue prediction
- Hot Tearing indicator

The features linked to specific processes (e.g. cycling, tilt, ...) are embedded in the corresponding physical modules.

User Manual Presentation

After a "Getting started" chapter, which presents briefly the set-up of a simple thermal case, the different modules of the software, starting by the "Software Manager".

The "Pre-processing" chapter is probably the most important as it describes the setting up of a case, from the FEM mesh to the run of the calculation. After an introduction on common features, this chapter is divided according to the "Physical modules", i.e. Thermal, Fluid Flow & Filling, Radiation and Stress. Then, the Databases, Run Parameters and Advanced features are presented.

After the Run of the calculation chapter, three chapters are dedicated to the Results viewing, the Results analysis and the Results exports.

Finally, Tips & Tricks and Tutorials will illustrate the use of the software.

Appendices are presenting the Software installation, the Files structure and the Models & Algorithms.

MeshCAST, the mesh generator, is presented in a separate manual.

Remarks

This is the first edition of the Manual of ProCAST (version 500). Please note that this manual does not yet contain a comprehensive description of all the features of the manual. Some sections will thus be "empty" at this stage. Further editions will complement this current version.

Getting started

Software principles

Enter topic text here.

Problem set-up

Enter topic text here.

Calculation

Enter topic text here.

Results display

Enter topic text here.

Software Manager



File Manager

When the ProCAST software is started, the Manager is launched, as seen in the figure below. The Manager allows to browse in the desired working directory, to create new directories and cases, to copy files from one directory to an other and to launch the different modules of the software.



ProCASTManager






The default starting directory can be set in the "Installation settings" (see the "Software configuration" section).

Firstly, the working directory should be defined (with the "browse"  and "create directory"  icons) and secondly, the name of the case (prefix name) should be defined.



Working directory and case name (prefix) definitions

The File Manager allows to perform some actions on the files, such as "Refresh"

 (to refresh the file list), "create new text file" , "Rename file" , "Duplicate file"  and "Delete file" .

A set of files can be copied from one directory to the other, using the

 button.

Then, two windows are opening (see figure below) and files can be copied from the left window to the right one. If the "Keep Prefix" check box is activated, the prefix remains unchanged, otherwise, the prefix is changed to the one defined under "Case" in the lower orange section.



Copy of files from one case to an other one

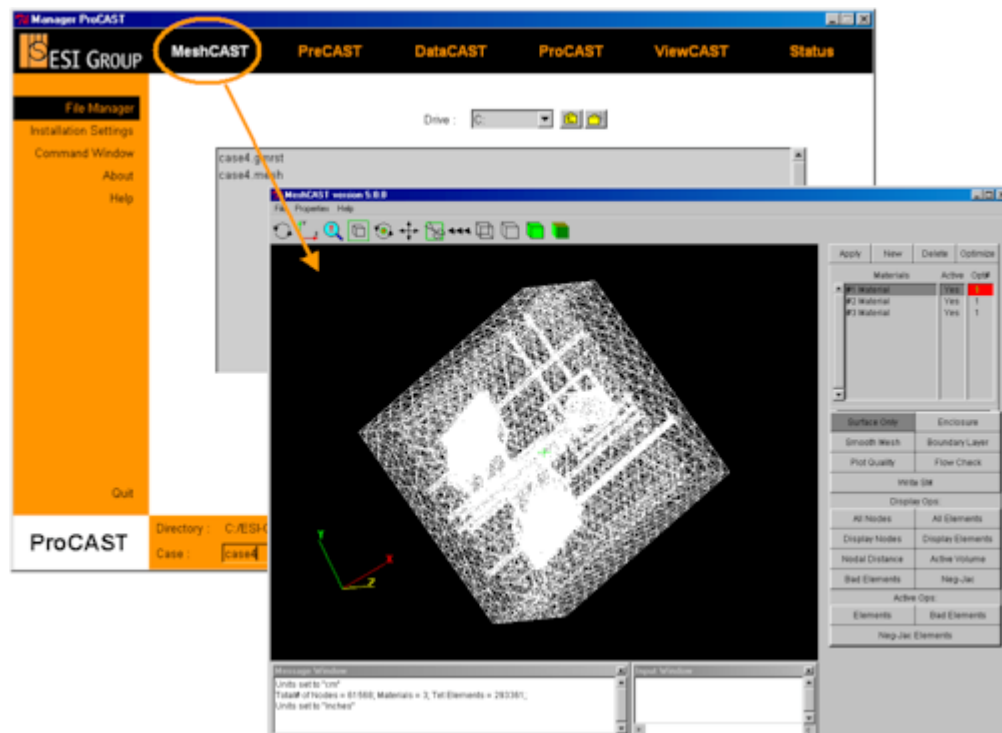
Module calls

The different modules of ProCAST can be called from the top bar buttons :

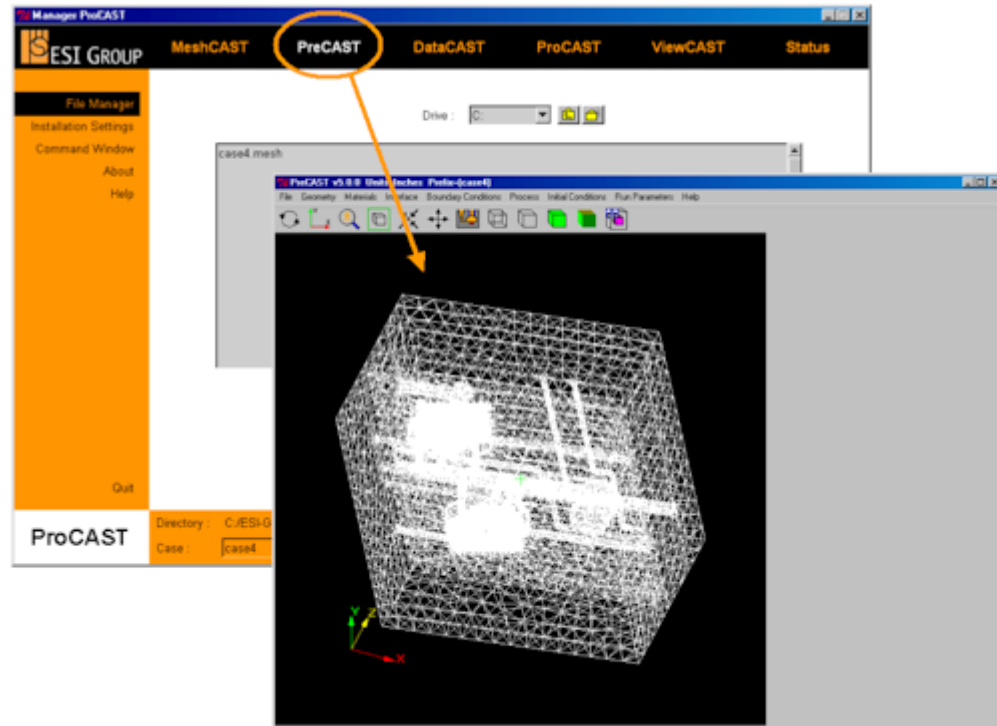


Firstly, the FEM mesh should be generated with **MeshCAST**. Then the case should be set-up with the pre-processor (**PreCAST**). Once all the parameters and conditions are defined, the calculation files should be prepared with **DataCAST**. The calculation can be run at this time by calling the **ProCAST** solver. Finally, the results can be visualized with the post-processor **ViewCAST**. At any time during and after the modeling, the status of the calculation can be monitored with **Status**.

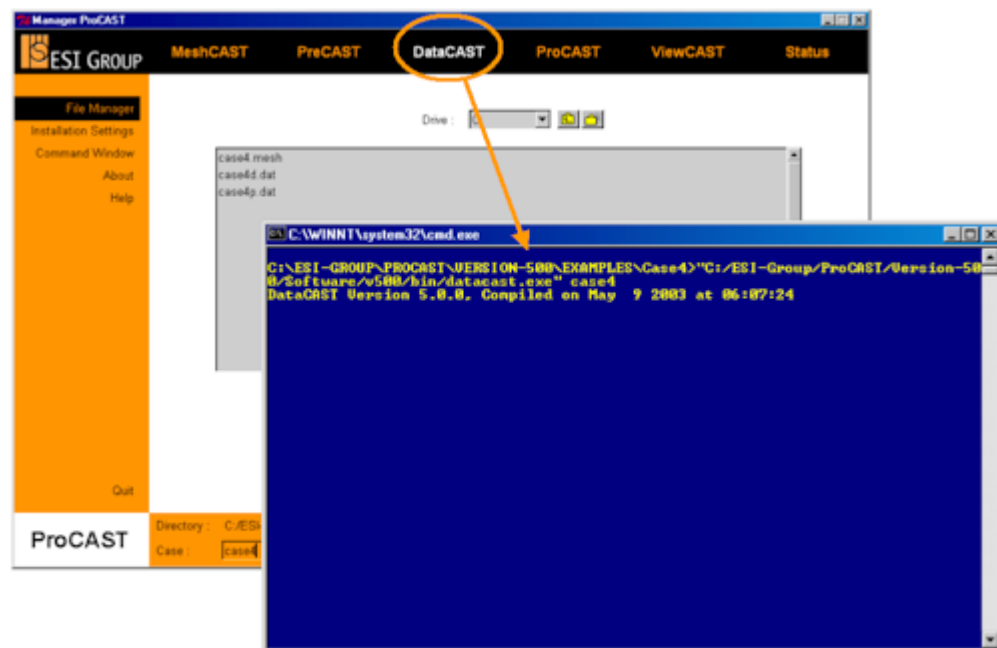
MeshCAST, the meshing module of ProCAST, is launched with the "MeshCAST" button. If a case name is defined, the corresponding "gmrst file" will automatically be loaded. Please refer to the MeshCAST User Manual for further details.



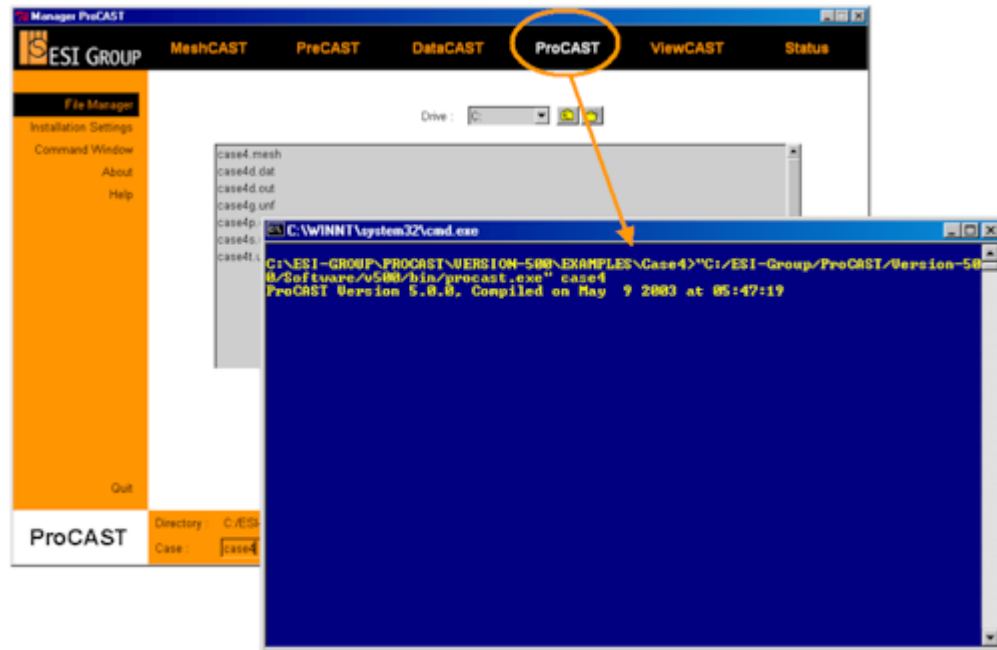
When PreCAST is launched, if a ".dat" file is present, it will automatically be loaded. Otherwise, the browser will prompt in order to select a ".mesh" file.



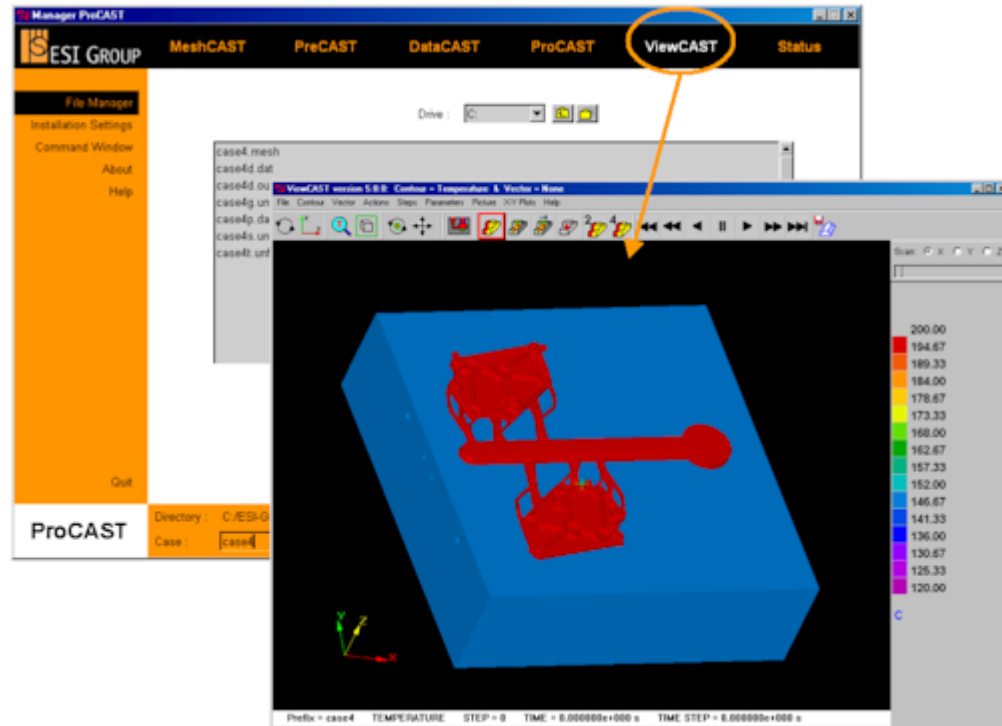
The DataCAST module is translating one input file generated by the pre-processor PreCAST (i.e. the d.dat file) into binary files (*.unf) which will be used by the calculation. Some error checking is also performed at this stage. When the DataCAST button is activated, a Command Window appears. This allows to view potential warnings or error messages. The operation is finished when the PAUSE appears on the window. Then, one should just hit any key (e.g. RETURN) in order to close this Command window and proceed with the calculation itself.



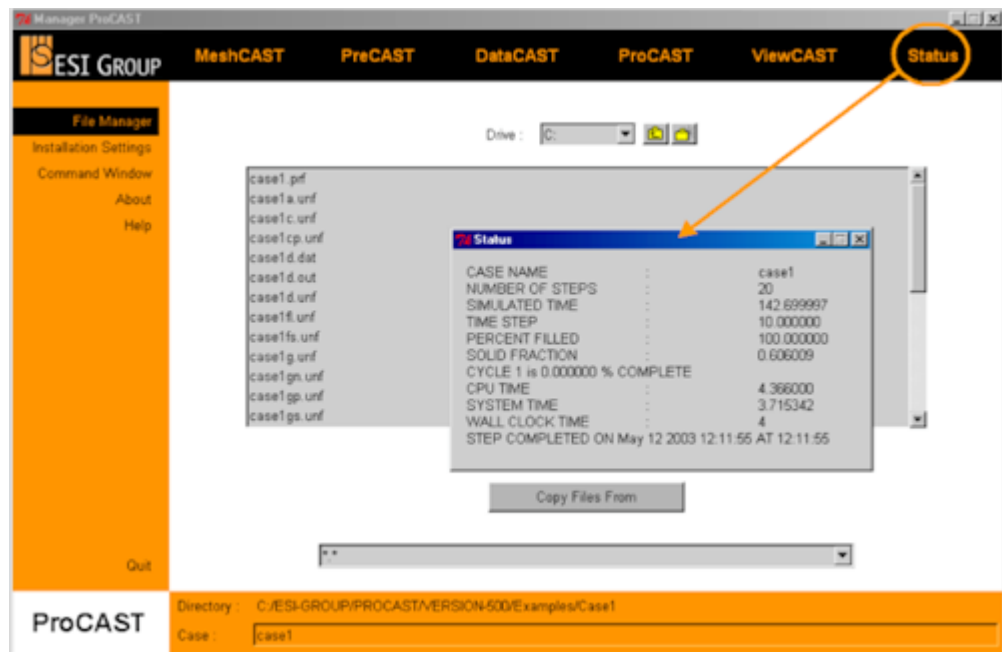
Once the data are ready (after PreCAST and DataCAST), the ProCAST solver can be launched, with the "ProCAST" button. A Command Window will be automatically opened and the solver will be started. As this window is independent from the Manager, it is possible to use the Manager (and the other modules, such as the post-processing for instance) while the calculation is running. When the calculation is finished, a PAUSE will be prompted and one has to hit any key (e.g. RETURN) to make the window disappear. If there is a problem during the execution, a warning or an error message may be printed in this window.



Results can be viewed at any time (even during the run of the calculation), with the ViewCAST button. If the case name is set, as well as the current working directory, the case is automatically loaded in the post-processor, as shown below.



To monitor the calculation, the "Status" button can be used at any time. A window will be prompted with the relevant information concerning the calculation (i.e. timestep, current time, filled and solidified fractions, cpu time, ...) corresponding to the case name indicated in the lower orange window.



Finally, all these modules can be called manually from a Command Window. The "Command Window" button opens automatically a Window in the working directory.



Then, the modules can be called in the following way (or via ad-hoc aliases) :

For Windows :

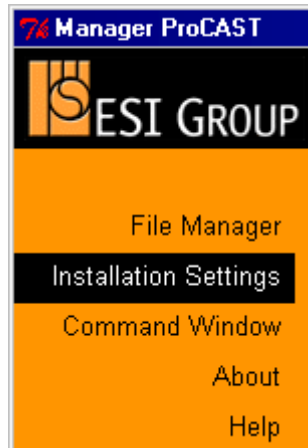
Manager	%ProCAST500%\bin\manager.exe
MeshCAST	%ProCAST500%\bin\meshcast prefix
PreCAST	%ProCAST500%\bin\precast prefix
DataCAST	%ProCAST500%\bin\datacast prefix
ProCAST	%ProCAST500%\bin\procast prefix
ViewCAST	%ProCAST500%\bin\viewcast prefix
Status	%ProCAST500%\bin\status prefix

For Unix (tcsh, csh) :

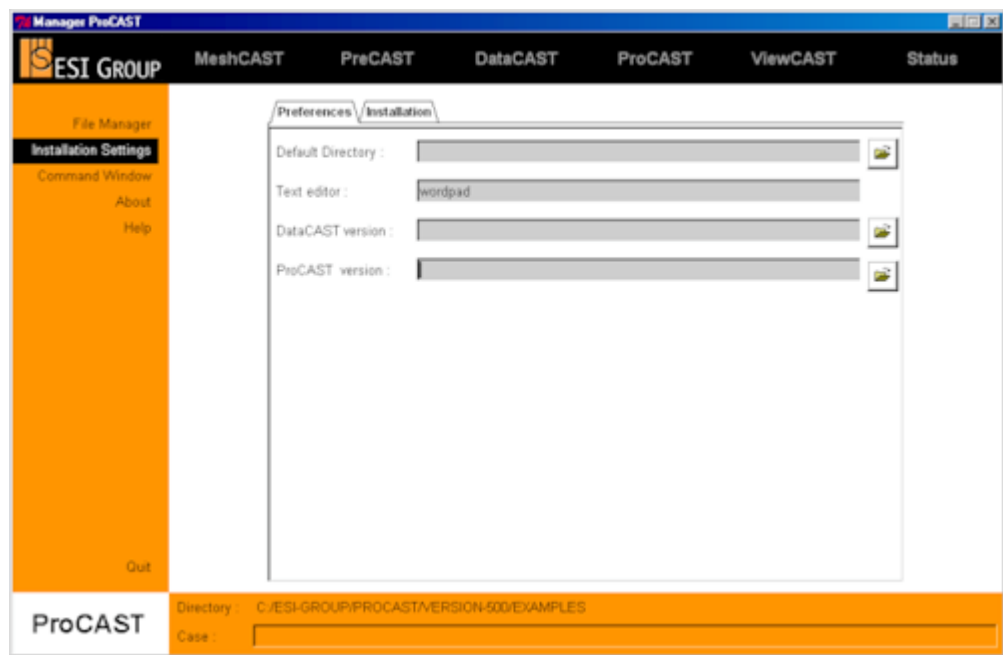
Manager	\$ProCAST500/bin/manager.exe
MeshCAST	\$ProCAST500/bin/meshcast prefix
PreCAST	\$ProCAST500/bin/precast prefix
DataCAST	\$ProCAST500/bin/datacast prefix
ProCAST	\$ProCAST500/bin/procast prefix
ViewCAST	\$ProCAST500/bin/viewcast prefix
Status	\$ProCAST500/bin/status prefix


Software configuration

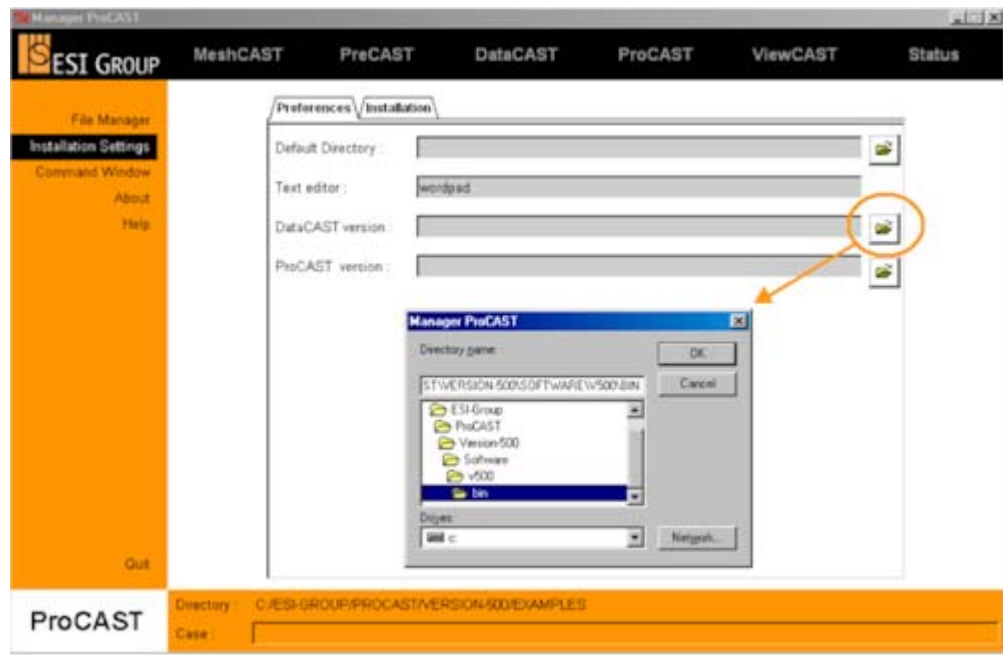
The software can be configured in the "Installation Settings" window :



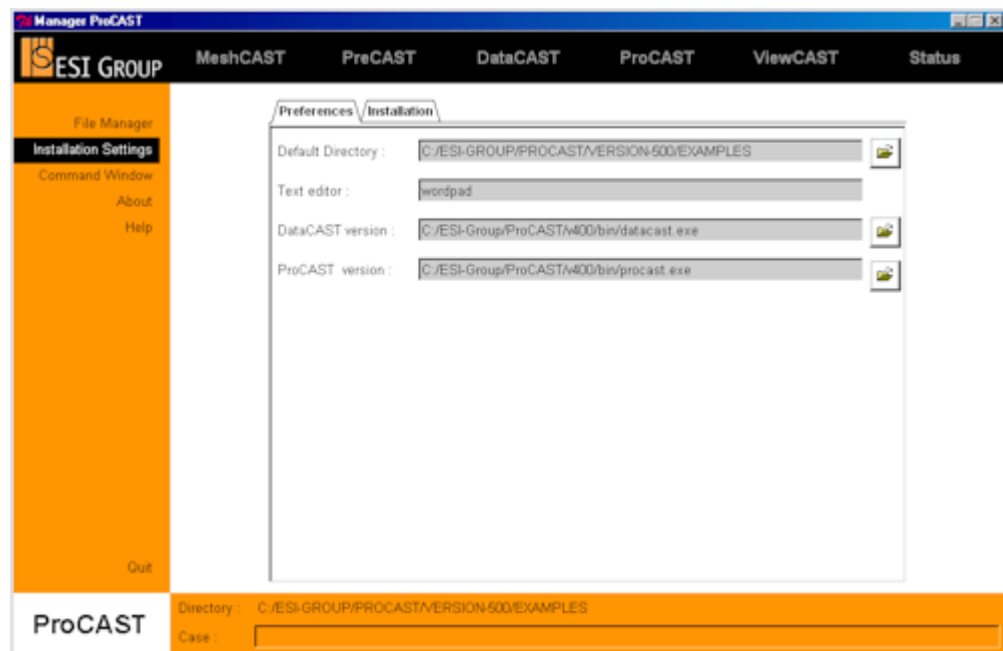
Firstly, the Preferences can be defined, such as the Default directory which will appear every time the Manager is launched, the preferred Text editor and the DataCAST and ProCAST versions that are called by the Manager.




The Browse button  should be used in order to select the desired directory, as shown below.




As the User interface (Manager, MesCAST, PreCAST and ViewCAST) of version 500 will be released before the solver (DataCAST and ProCAST), it will be used in conjunction with the version 40x of the solver. As DataCAST and ProCAST may be installed at different places on your computer, this is why the user has to browse the right executables of DataCAST and ProCAST, as shown hereafter (for version 400).

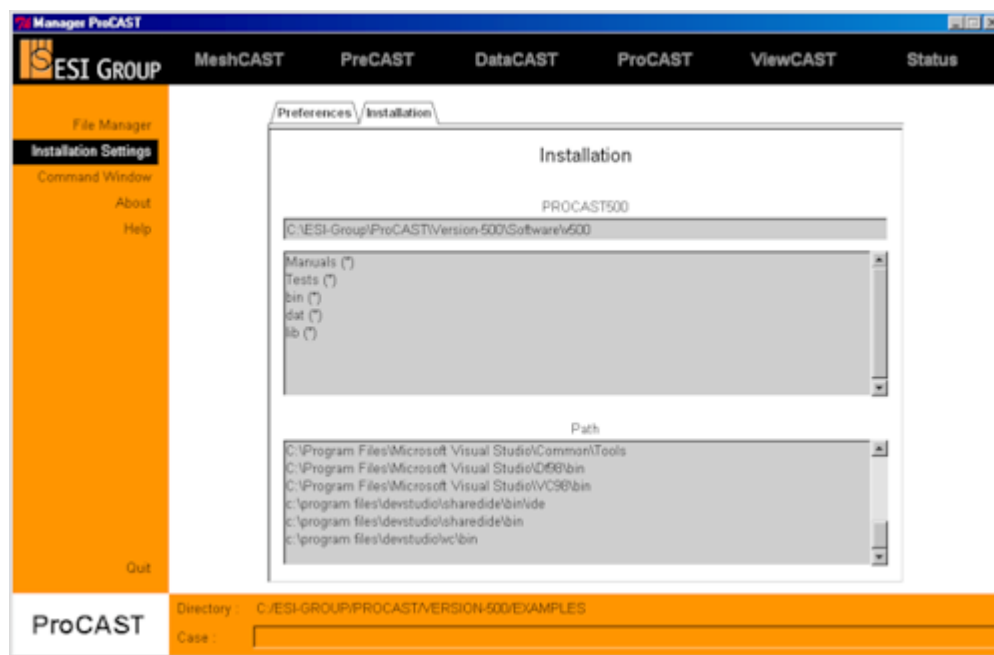


Then, automatically, when the DataCAST and ProCAST buttons will be used, the corresponding 40x executables will be called.

DataCAST version : 

ProCAST version : 

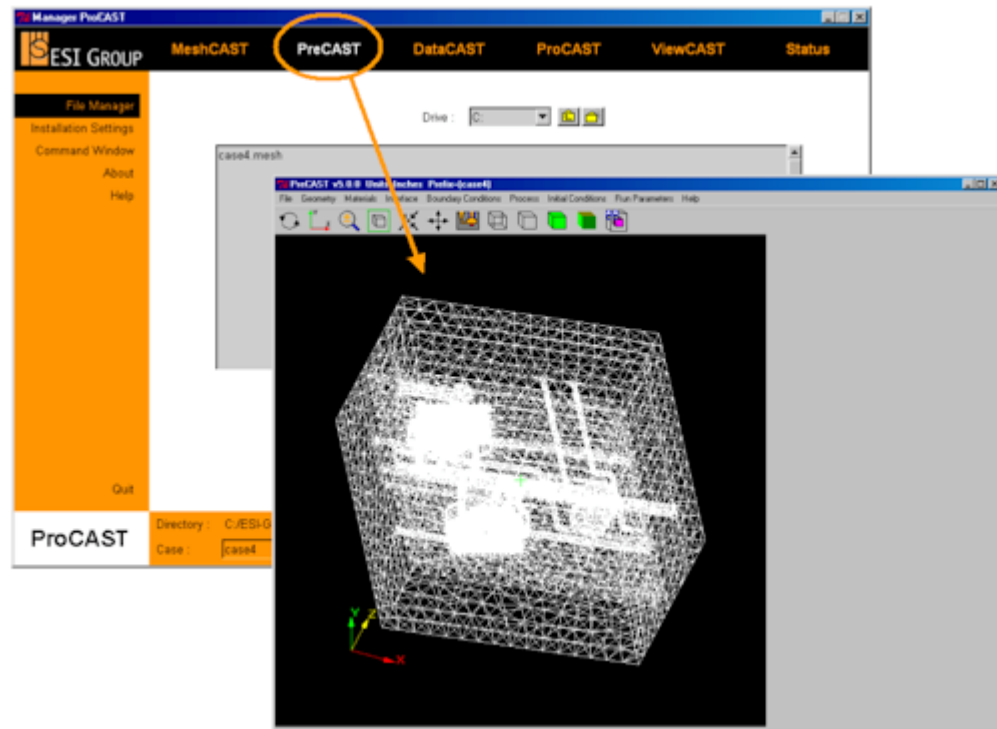
In the "Installation" tab, one can check that the environment variable ProCAST500 is well configured. It should point towards the v500 directory, above the "bin", "dat" and "lib" directories. Please refer to the "Software installation" section for more details.



Pre-processing

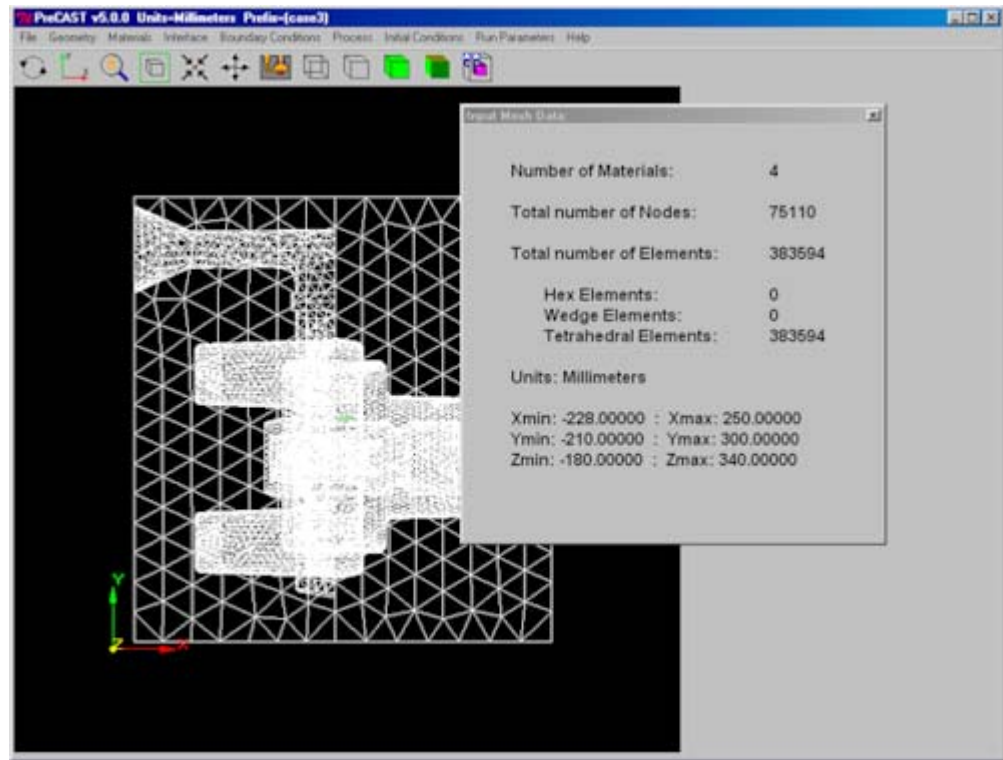
Introduction

To start the Pre-processor, the "PreCAST" button should be used. If a mesh file (case.mesh) or a "d.dat" file (cased.dat) is present, the case will be automatically opened.



If there is no case present in the working directory, then the browse window will open so that the user can select the desired input files (see the Geometry import section).

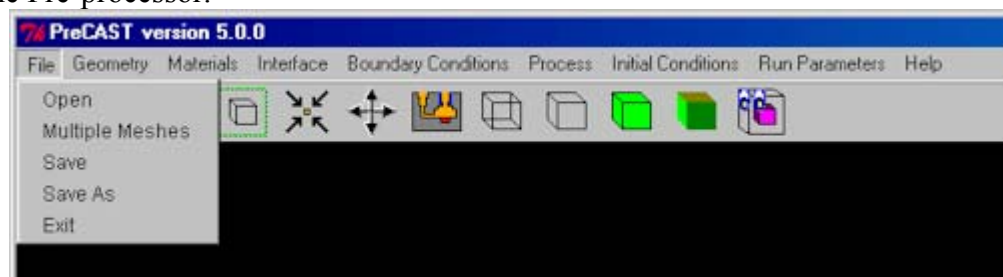
When the case is loaded, a window appears with some information about the model, such as the number of materials, the number of nodes and elements, as well as the model size. Then, the pre-processor is ready to set-up a case.



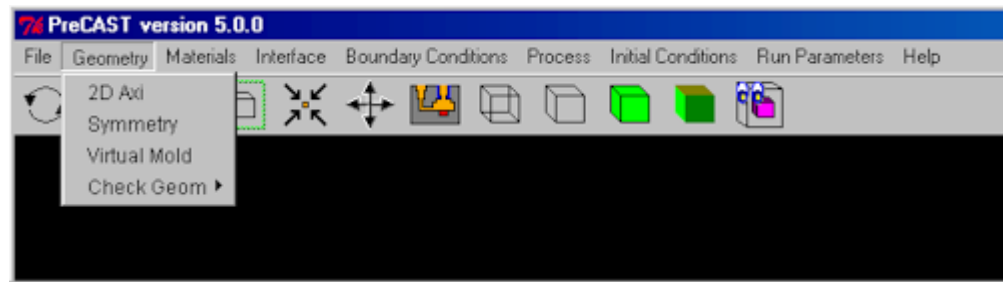
The top bar menu is divided in 8 menus which allow to perform all the operations to set-up a case :

- File
- Geometry
- Materials
- Interface
- Boundary Conditions
- Process
- Initial Conditions
- Run Parameters
- Help

First the model should be opened or saved in the **File** menu. It allows also to quit the Pre-processor.



Then, in the **Geometry** Menu, symmetries can be defined, as well as the virtual mold characteristics. Moreover, some features of the FEM mesh can be checked, such as negative Jacobians, or the volume of each domain.

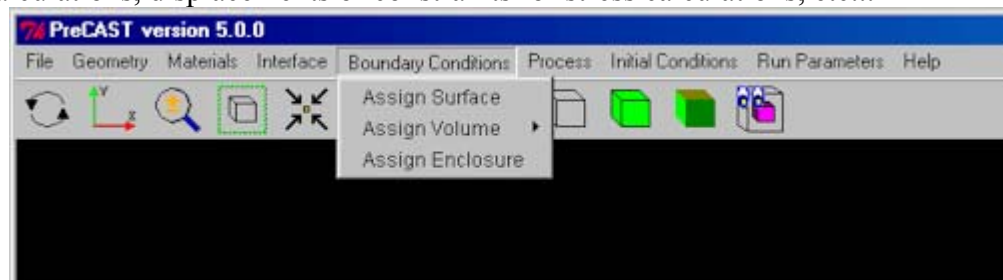


In the **Material** menu, the characteristics of each domain (or each material) can be defined. In addition to the material properties, one can specify the type of the domain (casting, mold, filter, foam, ...), as well as if it will be empty or not at the beginning of the calculation (for mold filling).

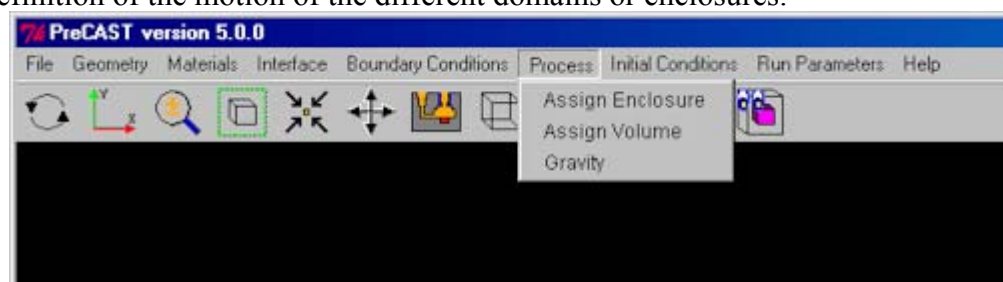


The **Interface** menu has no sub-menus. It opens the window which allows to define the interactions between the different materials, such as heat transfers.

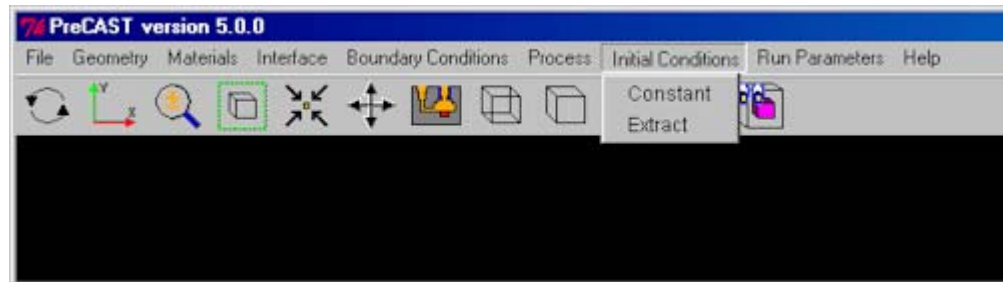
The **Boundary condition** menu allows to define all the interactions between the different materials and the outside world (i.e. on the outside surfaces of the model), such as external cooling, velocities at the surface of the model for flow calculations, displacements or constraints for stress calculations, etc...



The **Process** menu gives access to the definition of the gravity, as well as the definition of the motion of the different domains or enclosures.

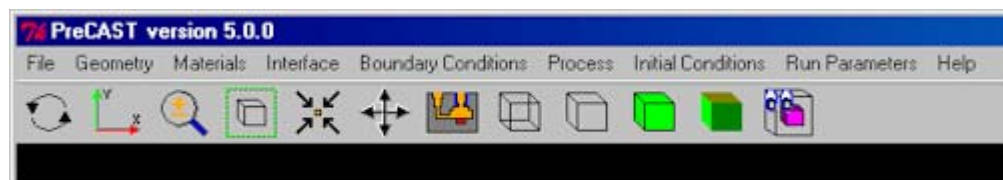


The initial temperatures of each materials are defined in the **Initial Conditions** menu.



The **Run Parameters** menu, as well as the **Help** menu have no sub-menus. All the calculation parameters are defined in the Run Parameters window. The on-line Help can be access from the Help menu.

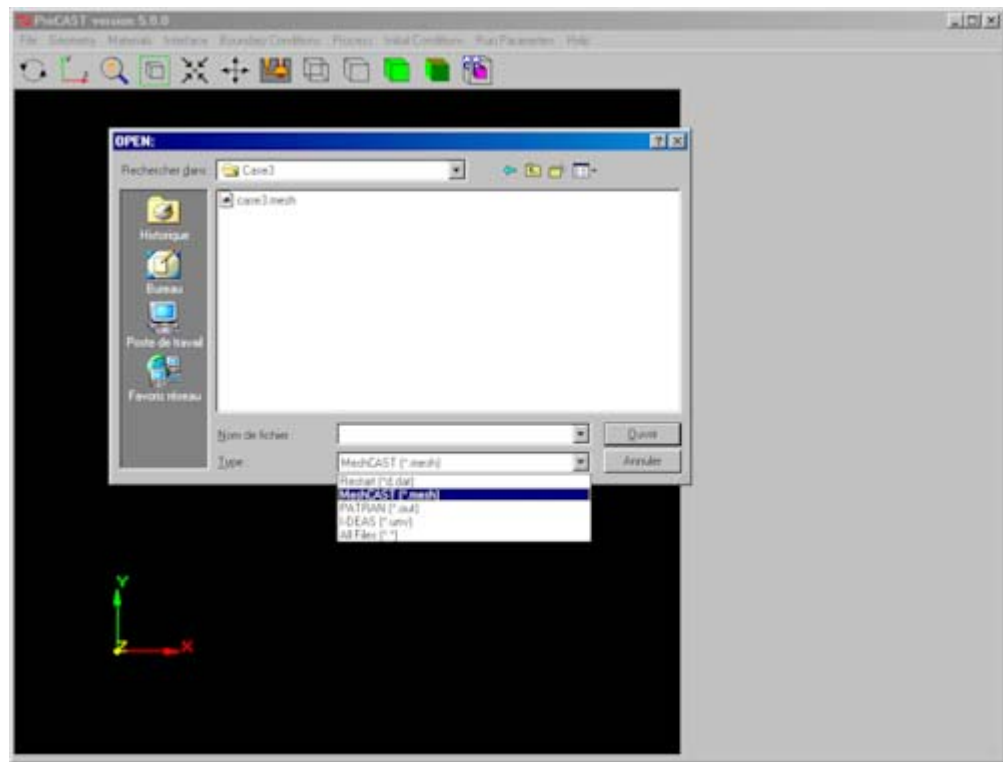
Below the menus, icons allows to perform a number of operations linked to the display of the model on the screen. These icons are described in the Geometry manipulation section.



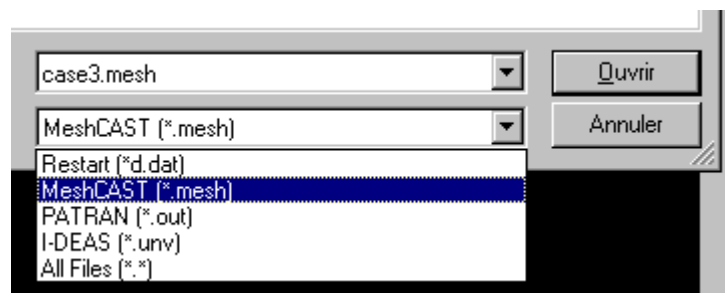
Geometry import

When PreCAST is started from the Manager, it is automatically reading a mesh or a d.dat file (if they are present in the working directory, with the selected prefix). The priority is first a "d.dat" and then a ".mesh" file.

If there is no mesh file or d.dat file present in the working directory with the corresponding prefix, the pre-processor opens with the browser window.



Then the user has the choice of the input format, through the following filter :



The Pre-processor is able to read PreCAST input files (*.d.dat), also called "Restart" files, or meshes coming from MeshCAST (*.mesh), from PATRAN (*.out) or I-DEAS (*.unv).

The reading of multiple meshes (for non-coincident meshes or for radiation calculation) is described in the "Advanced features" section of the Pre-processor.

Thermal

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).

Geometry assignments

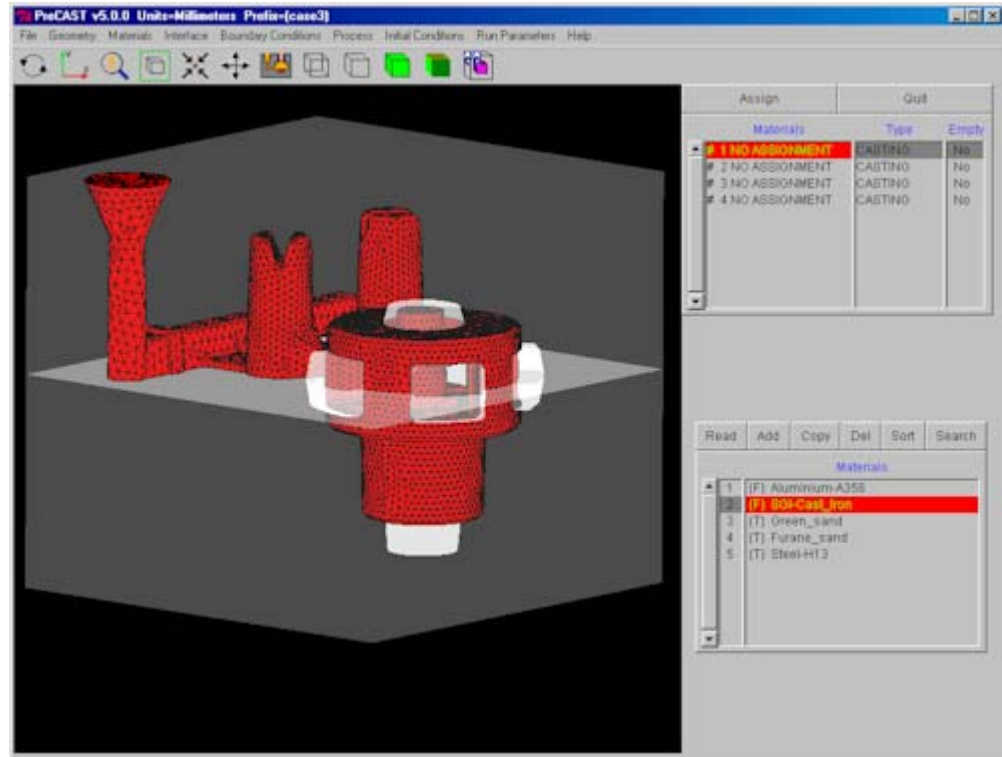
Enter topic text here.


Materials assignment

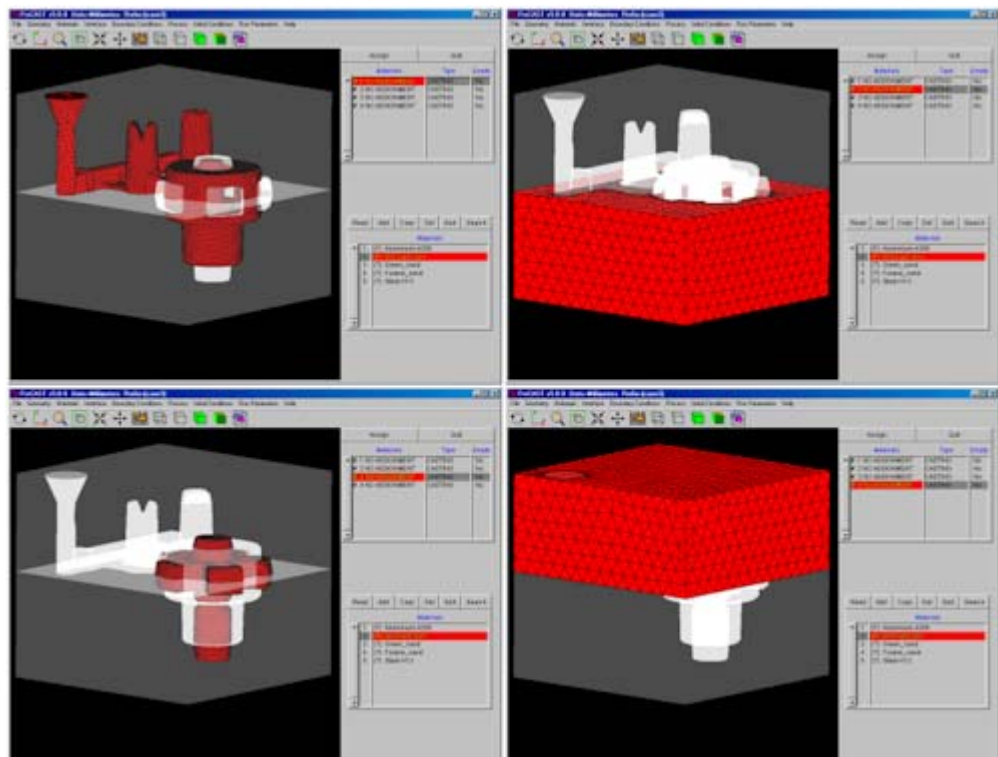
Once the model is loaded (see the "Geometry import" section), the first operation is to define the different materials with their properties and attributes. This is performed in the Material/Assign menu.



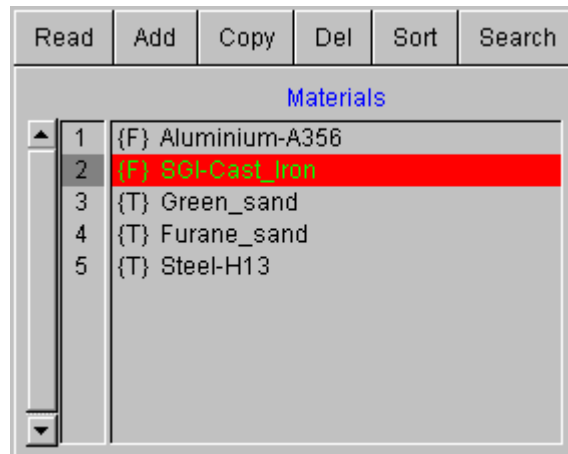
On the right of the window, two frames are shown. The top one contains the material list (or domain list), whereas the bottom one corresponds to the material database.



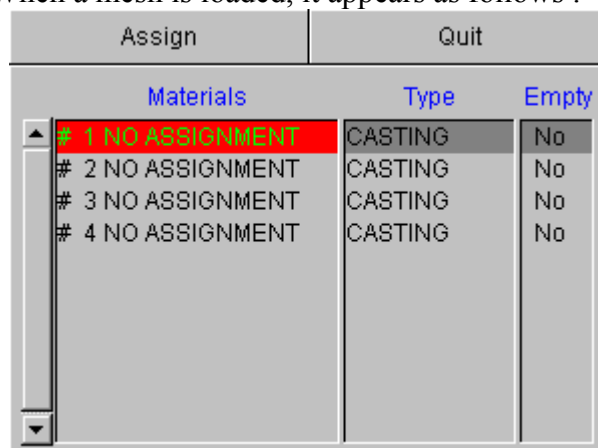
When one clicks on the different materials in the material list, the corresponding domains are highlighted (in the picture below, the hidden mode with mesh  was selected - see the "Geometry manipulation" section for the other display modes).



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.

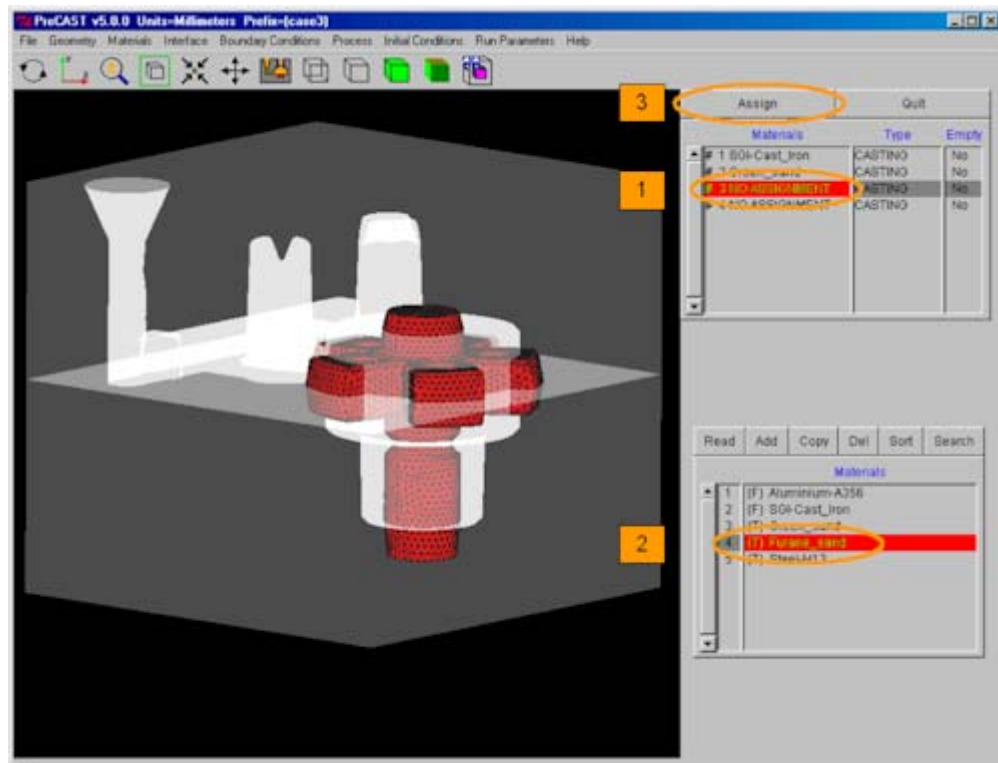


In the top frame ("Domain list"), all the domains (or materials) present in the mesh are listed. When a mesh is loaded, it appears as follows :

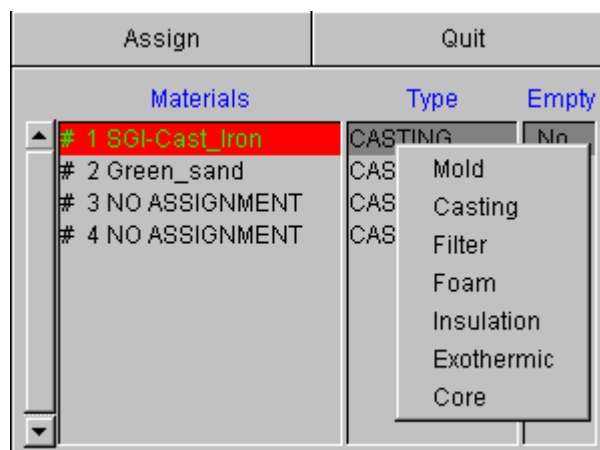


Then, one should assign a Material to each domain, to define the type of each domain and to specify whether the domain is empty or not at the beginning of the calculation.

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.



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 :



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.

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 will be active only with the version 500 of the ProCAST solver)*
xxxxxx

Exothermic : *(this will be active only with the version 500 of the ProCAST solver)* xxxxxx

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.

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).

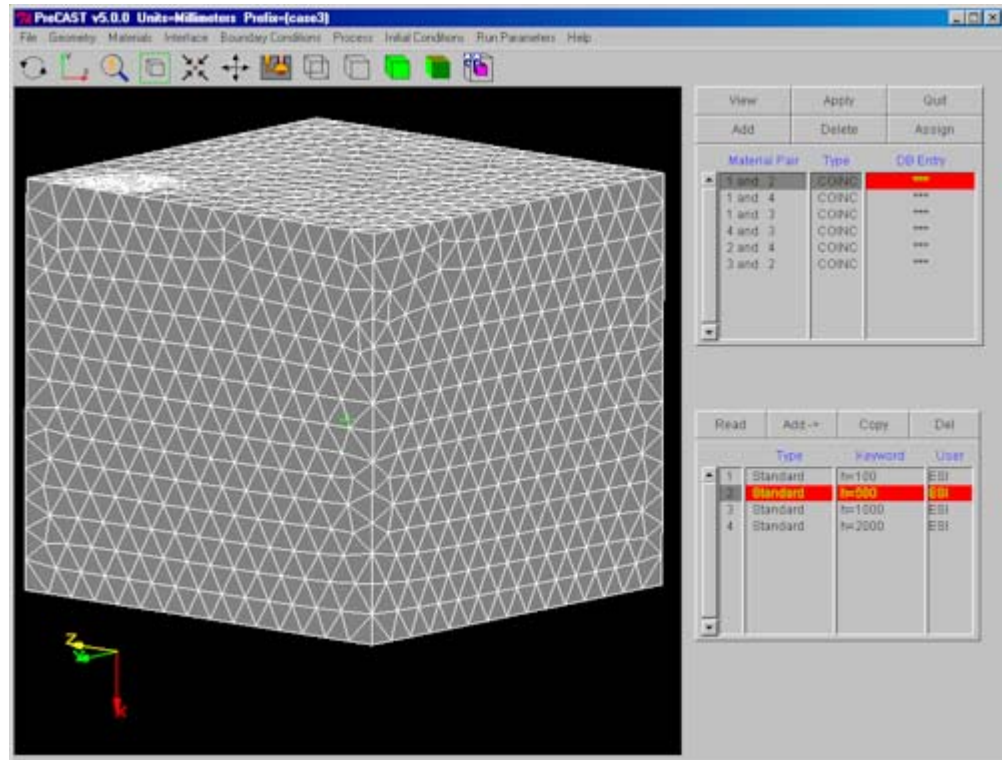
Assign		Quit
Materials	Type	Empty
# 1 SGI-Cast_Iron	CASTING	Yes
# 2 Green_sand	MOLD	No
# 3 Furane_sand	MOLD	No
# 4 Green_sand	MOLD	No

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.



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

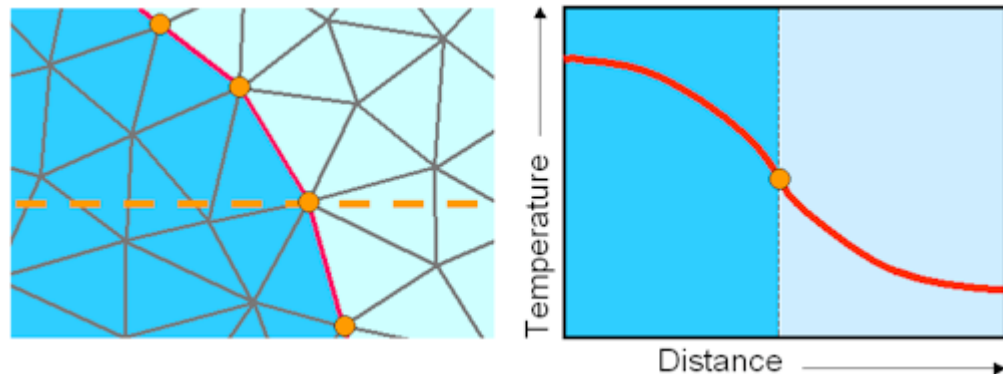
View	Apply	Quit
Add	Delete	Assign
Material Pair	Type	DB Entry
1 and 2	COINC	***
1 and 4	COINC	***
1 and 3	COINC	***
4 and 3	COINC	***
2 and 4	COINC	***
3 and 2	COINC	***

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".

When a mesh is generated with MeshCAST (or with most common mesh generator), the elements which are on either side of an interface (i.e. adjacent elements which belongs to two different domains) are sharing the same nodes. This is called a coincident mesh.

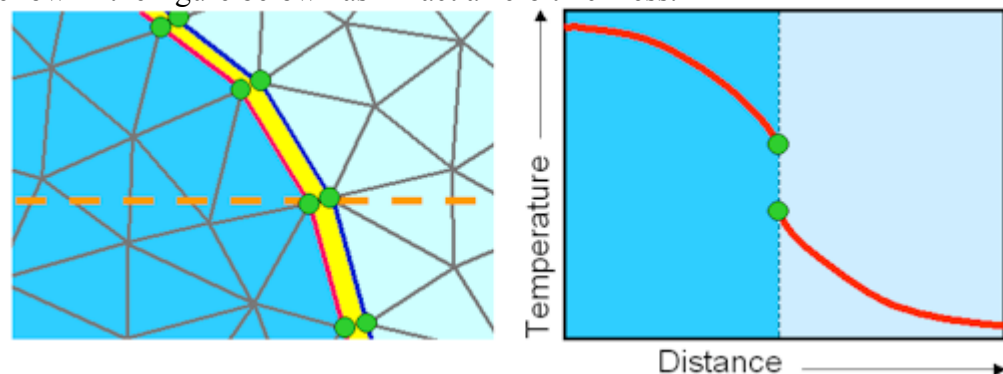
EQUIV option

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).



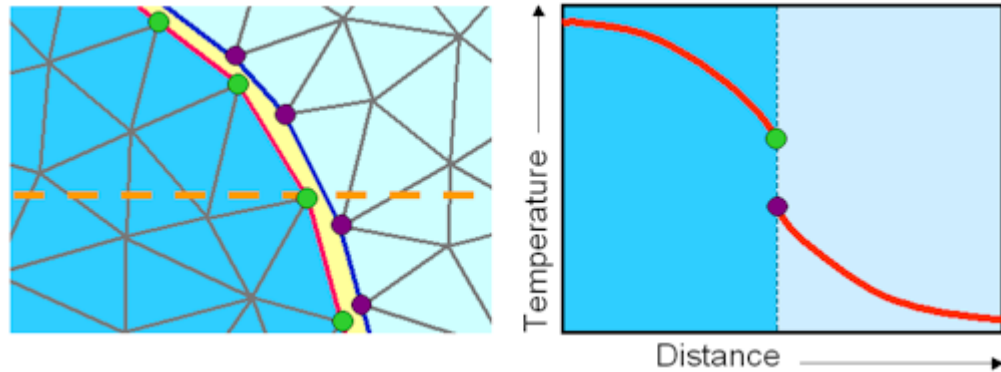
COINC option


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.

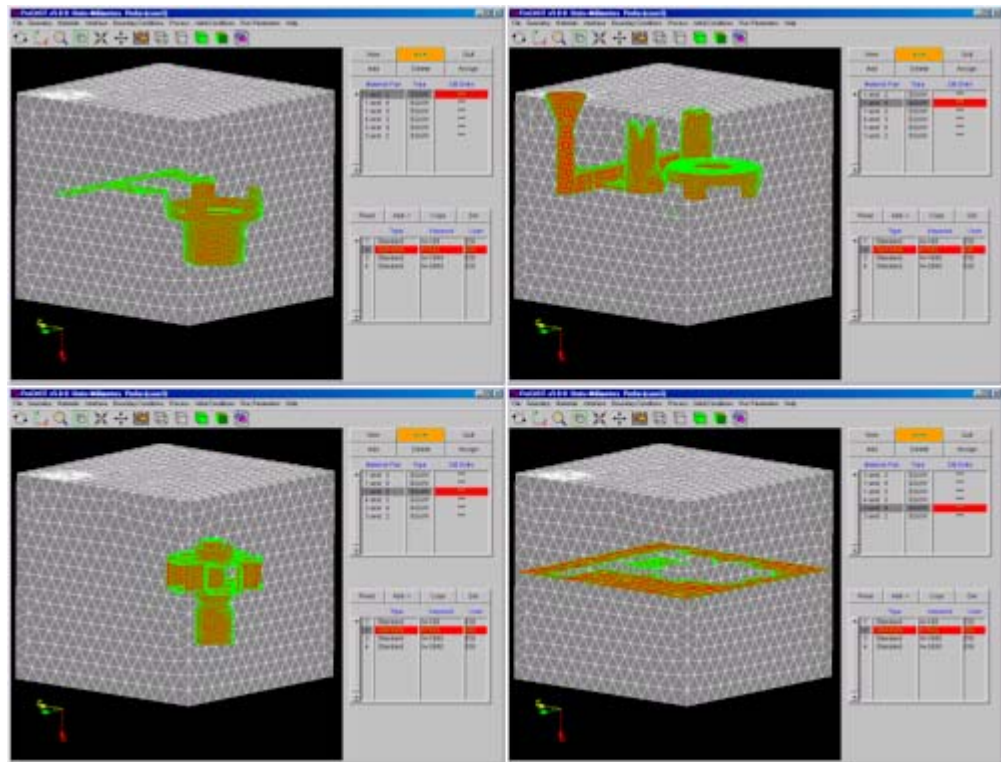


NCOINC option


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.

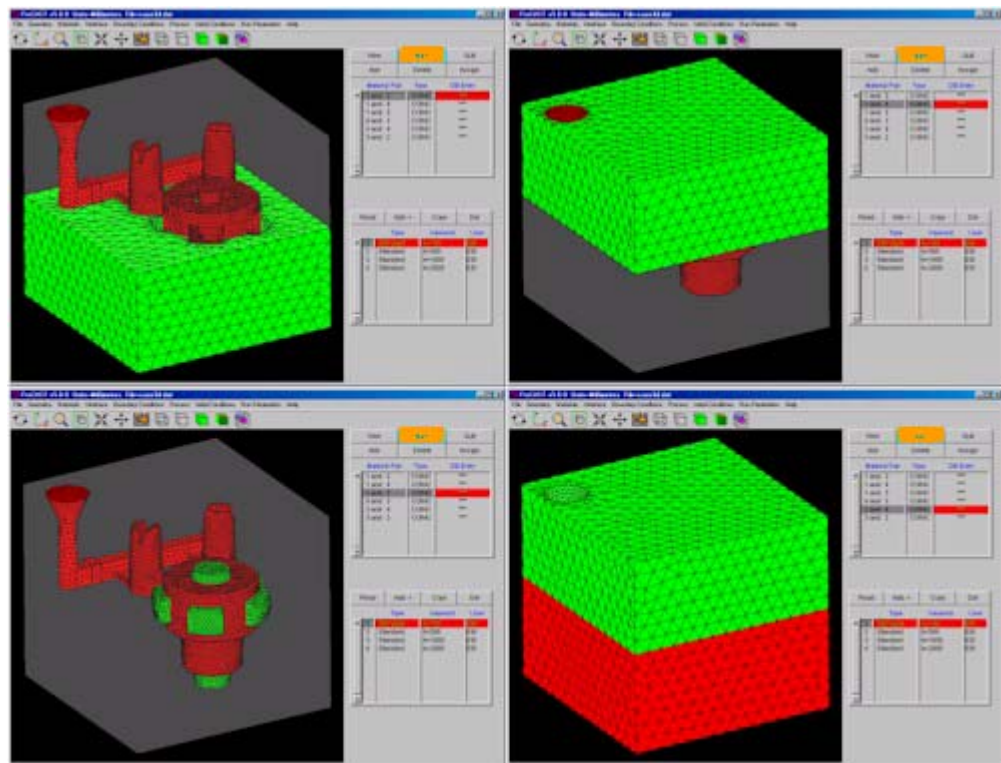


When one toggles between the different options, the interface appears in red and green. It is thus possible to well identify whether it corresponds well to the desired interface (see figure below, which was obtained in hidden mesh mode )

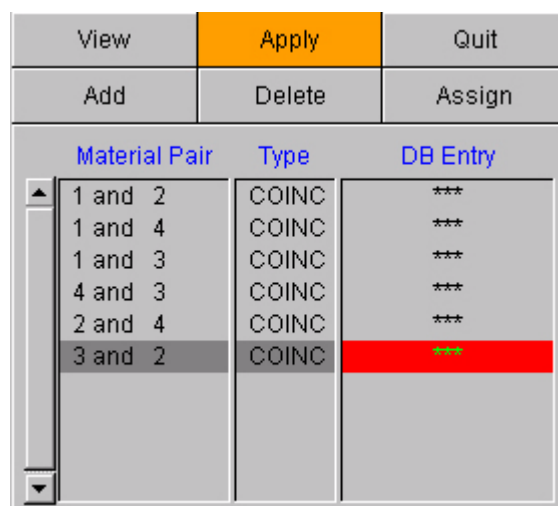


An other way to view the desired interfaces is to click on the "Material Pair" and the material on both sides will be highlighted in red and green respectively - the

first material in the list is in red and the second one in green (see figure below, which was obtained in hidden mesh mode )



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.



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 500 to go from EQUIV to COINC and vice-versa, 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.

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.

Boundary conditions assignment

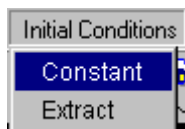
Enter topic text here.

Process conditions assignment

Enter topic text here.

Initial conditions assignment

The initial temperature of each material should be defined in the initial condition menu.



Run parameters assignment

The following Run Parameters should be specified :

THERMAL

...

Please refer to the "Run Parameters" section for the Advanced Run parameters

Fluid flow & Filling

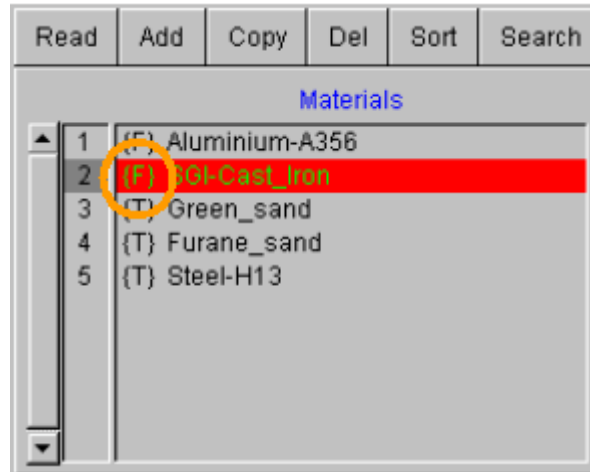
This section describes the additional set-up necessary for fluid flow and mold filling. For the set-up of a thermal case, please refer to the "Thermal" section of the Pre-processor.

Materials menu

Concerning the material assignments, the only requirement is that the domains in which fluid flow calculation will be performed are defined as "Casting" domains (or "Filter" or "Foam" - see the "Advanced features" section for more details). Then, one should define whether the corresponding domain is Empty or not (in the case of mold filling). One can change from No to Yes and vice-versa by clicking on the text directly.

Assign		Quit	
Materials	Type	Empty	
# 1 SGI-Cast_Iron	CASTING	Yes	
# 2 Green_sand	MOLD	No	
# 3 Furane_sand	MOLD	No	
# 4 Green_sand	MOLD	No	

Moreover, the material properties of the fluid material should have the flow properties (e.g. viscosity). This can be checked in the Material database list. The Material name should be preceded by an {F} (for Fluid properties).



Interface menu

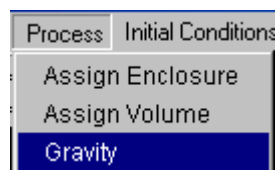
Concerning the interfaces, nothing special should be done concerning the fluid flow. One can however notice that if the casting (i.e. the flow domain) is non-coincident with the mold domain(s), one will need to set a zero velocity boundary condition all around the casting domain (in order to prevent "leaks"). See the "Advanced features" section for more details on non-coincident meshes.

Boundary conditions menu

Concerning the boundary condition assignments, one has to specify **xxxxx**

Process menu

For fluid flow problems, the gravity has to be defined.



Initial conditions menu

Nothing should be specified in this menu for fluid flow problems.

Run parameters menu

The following Run Parameters should be specified :

FLOW

...

Please refer to the "Run Parameters" section for the Advanced Run parameters

Radiation

In addition to heat conduction and thermal convection heat transfer, ProCAST allows to calculate radiative heat transfer.


Radiation can be either treated as a simple radiative flux (described in the "Thermal/Boundary conditions" section), or with a complex radiation algorithm which takes into account reflexions, obstructions and shadowing effects. The set-up of a case with such complex radiation (called hereafter "Radiation with view factors") is explained in this section.

As Radiation with view factors involves the calculation of the interaction of the components (casting and mold) with the environment (furnace, castshop, ...), it is necessary to include the environment into the model. This is done with an "Enclosure".

If the casting is put into a furnace, the Enclosure is the furnace itself (or the inner skin of the furnace). However, if the casting (and mold) is sitting on the floor of the castshop, one should set an "artificial" enclosure which will surround the casting and which will have the same effect as the environment.

An enclosure can be either a solid (represented by a solid 3-D FEM mesh, as the casting, mold, etc...), or by a closed surface (represented by a closed FEM surface mesh).

The figure below shows a casting within a solid enclosure (for symmetry reasons, only a sector is modeled).

A yellow rectangular box with the text "Figure under construction" in blue.

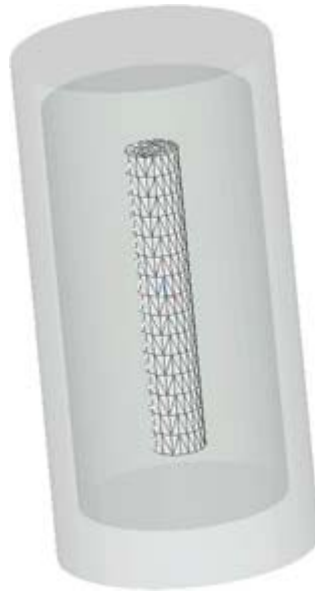
The figure below is showing a casting within a surface enclosure (for symmetry reasons, only a sector is modeled)

A yellow rectangular box with the text "Figure under construction" in blue.

Symmetry menu

Firstly the symmetries (if any) should be defined. ProCAST is able to deal with symmetry implying one mirror, two orthogonal mirrors, a single rotation of n sectors and a combination of them.

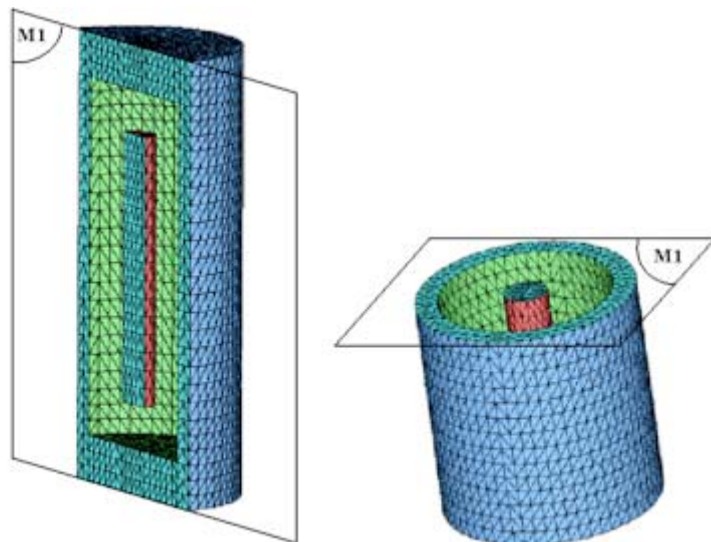
In order to illustrate the different possibility of symmetry, consider the simple but explicit example of two concentric cylinders (the inside cylinder is the casting and the outside cylinder is a solid enclosure). The full geometry is shown in the figure hereafter.



The full geometry of the two concentric cylinders.

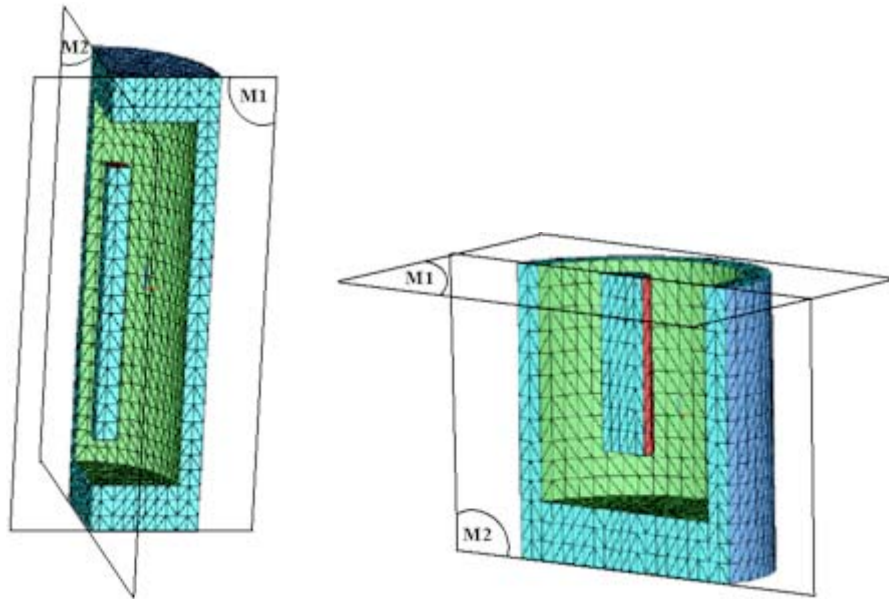
The following figures illustrate the different possibility of simplification by symmetry.

- One mirror (M1)



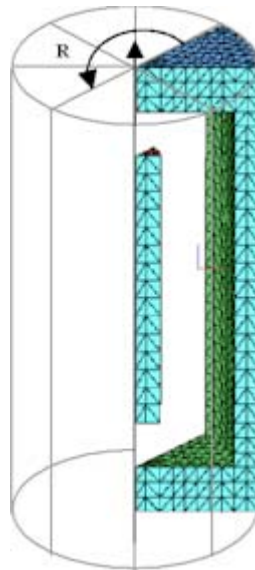
Two examples of simplified geometry by one mirror.

- Two orthogonal mirrors (M1 and M2)



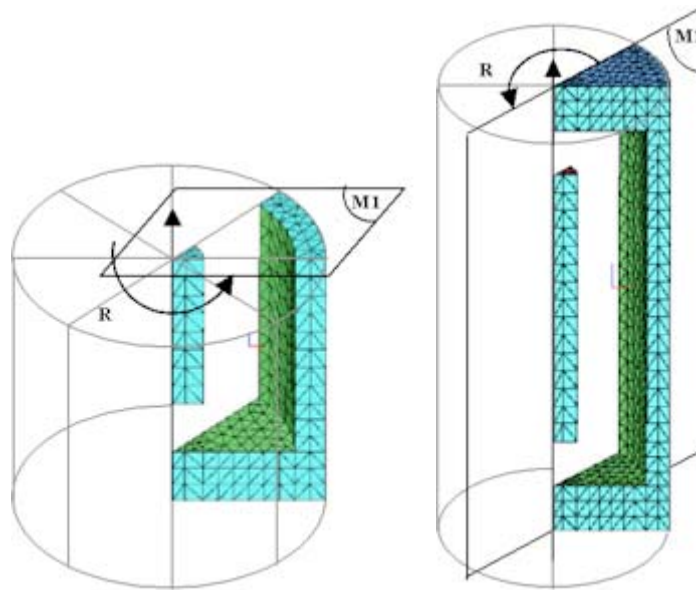
Two examples of simplified geometry by two orthogonal mirrors.

- A simple rotation of n sectors (R)



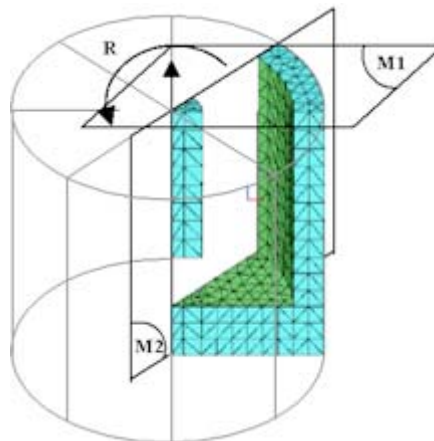
A simplified geometry by a rotation of 8 sectors.

- A rotation of n sectors (R) associated with a mirror (M1)



Two examples of simplified geometry by a rotation (left: 8 sectors, right: 4 sectors) and one mirror.

- A rotation of n sectors (R) associated with two orthogonal mirrors (M1 and M2)



A simplified geometry by a rotation with 4 sectors and two orthogonal mirrors.

The symmetries are defined in the Geometry/Symmetry menu :

The image shows a 'Symmetry Specs' dialog box with the following structure:

- Rotational:** An unchecked checkbox. To its right are six input fields for coordinates: X1, Y1, Z1, X2, Y2, Z2, all containing '0.000000e+000'. Below these is a '#Sectors:' field containing '1'.
- Mirror-1:** A checked checkbox. To its right are six input fields for coordinates: X1, Y1, Z1, X2, Y2, Z2, all containing '0.000000e+000'. Below these are three more input fields: X3, Y3, Z3, also containing '0.000000e+000'.
- Mirror-2:** An unchecked checkbox. To its right are six input fields for coordinates: X1, Y1, Z1, X2, Y2, Z2, all containing '0.000000e+000'. Below these are three more input fields: X3, Y3, Z3, also containing '0.000000e+000'.
- Buttons:** At the bottom are three buttons: 'Apply', 'Get Co-ord', and 'Cancel'.

In case of a rotation, the axis of rotation should be defined by the coordinates of two points and the number of sectors. Then, the "Rotational" check box should be checked. One should be careful that the mesh should be defined so that the selected number of sectors will not create overlaps of the mesh (i.e. to specify 7 sectors for a mesh which corresponds to a sector of 60°).

For mirror symmetries, each plane of symmetry should be defined by the coordinates of three points (which should not be co-linear). Then the "Mirror-1" and "Mirror-2" (if applicable) check box should be checked. If two mirror symmetry are used, one should be careful to make sure that the two planes are orthogonal. In the case of mirror symmetry with a rotation, the axis of rotation should be either perpendicular or parallel to the mirror plane(s). In case of parallel plane(s), the axis of rotation should be within the mirror plane(s).

Then, the "Apply" button should be used to validate the symmetry definition. To disable a symmetry, just uncheck the corresponding check box.

The "Get Co-ord" button allows to pick nodes of the FEM mesh for an interactive definition of the mirror planes or rotational axis. To use it, first click in the X coordinate box of the point which should be defined interactively, then click on "Get Co-ord" and finally, click on the desired node on the geometry. The corresponding coordinates will fill automatically the corresponding fields. Repeat it for the other points.

Materials and Interface menu

Nothing specific should be defined at the level of the Materials and Interface menus. The complex radiation method should be set in the "Boundary conditions" and "Process" menus.

Boundary conditions menu

Process menu

The process menu allows to define the motion (if any) of the enclosure with respect to the casting.

Run parameters menu

The following Run Parameters should be specified :

RADIATION

...

Please refer to the "Run Parameters" section for the Advanced Run parameters

Stress

This section describes the additional set-up necessary for stress calculations. For the set-up of a thermal case, please refer to the "Thermal" section of the Pre-processor.

Materials menu

Concerning the material assignments,

Interface menu

Concerning the interfaces,

Boundary conditions menu

Concerning the boundary condition assignments, one has to specify **xxxxx**

Process Initial conditions menu

Nothing should be specified in these menus for stress problems.

Run parameters menu

The following Run Parameters should be specified :

STRESS

...

Please refer to the "Run Parameters" section for the Advanced Run parameters

Databases

Enter topic text here.

Thermotec database : Include the Alloying elements available in each database.

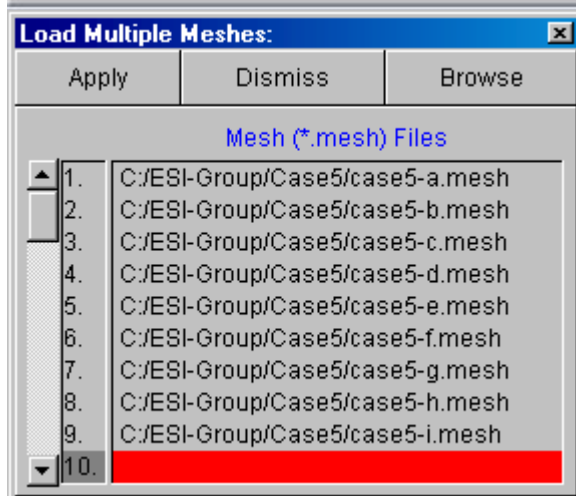
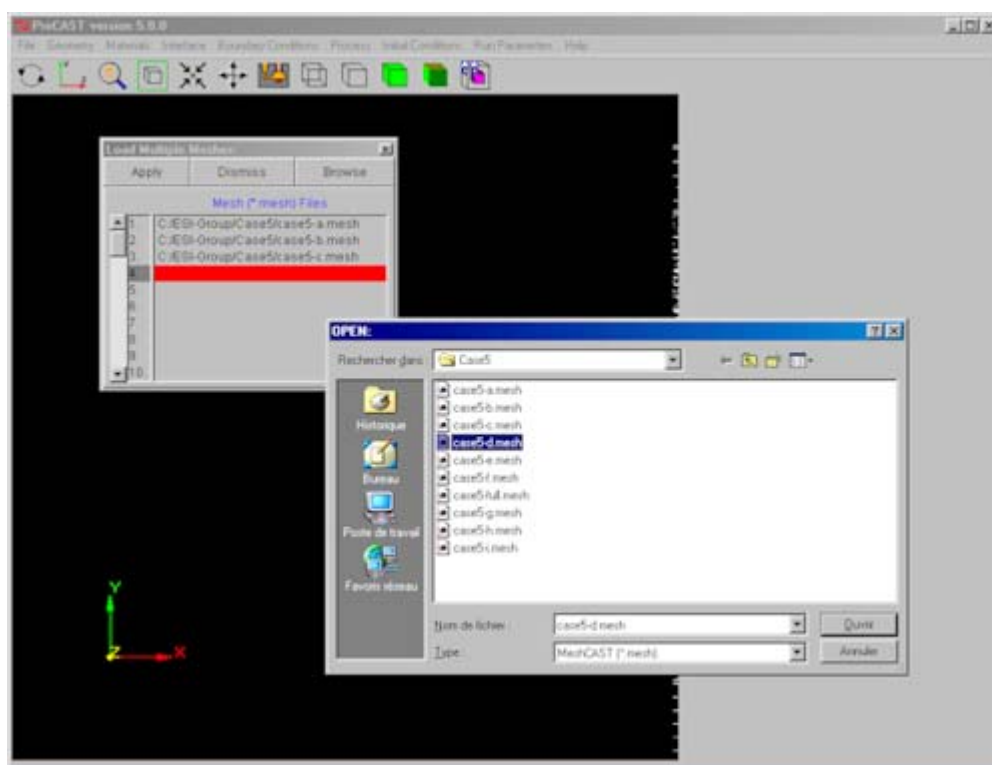
Run Parameters

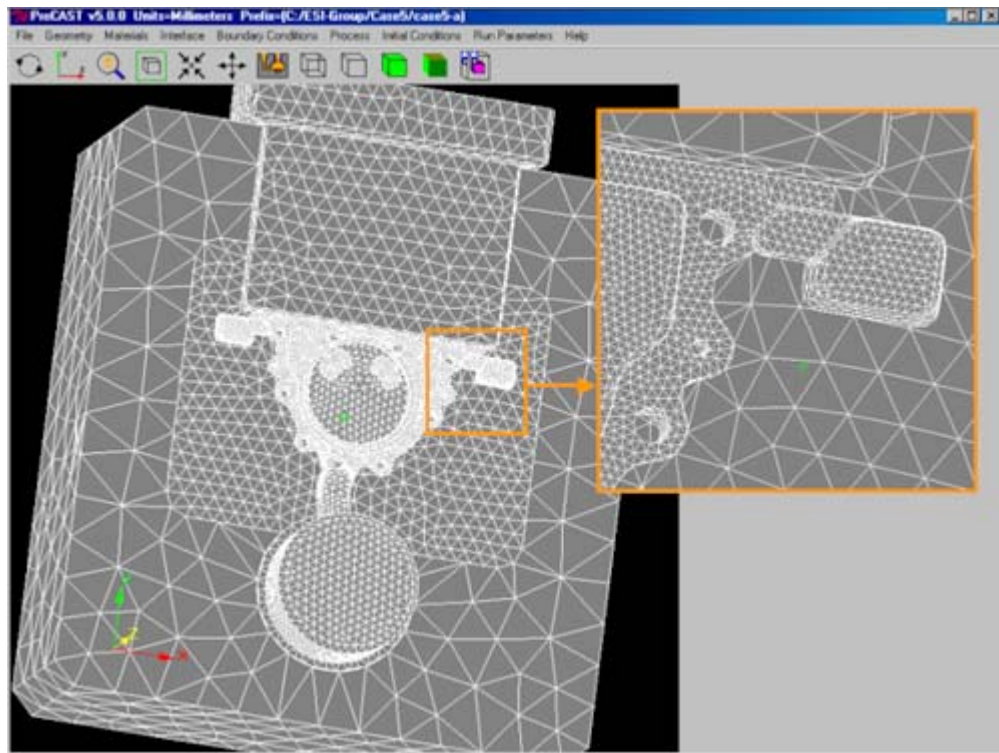
Enter topic text here.

Advanced features

Enter topic text here.

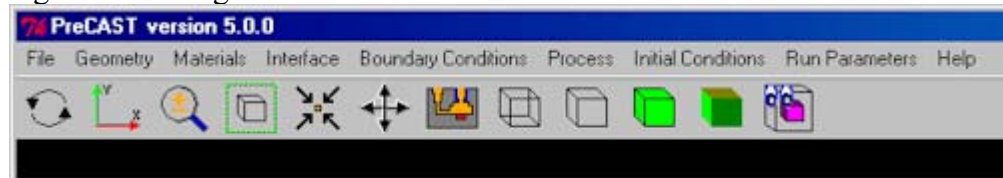
- multiple meshes
- cycling
- filter
- lost foam



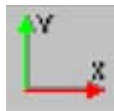


Geometry manipulation

The geometry can be manipulated and can be displayed with different modes, using the following icons.



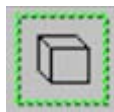
Manual rotation of the model (it opens the Rotate window). Otherwise, the model can be rotated interactively with the mouse at any time (as long as the Center or Drag icon is not activated). If the model is rotated interactively with the mouse, while the "Shift" key is pressed in the same time, the model is rotating along the axis perpendicular to the screen (the mouse should be rotated horizontally).



Restore the X-Y orientation of the model (Z-axis perpendicular to the screen)



Interactive zoom (the model is enlarged when the cursor is moved towards the bottom of the screen and it is reduced when the cursor is moved towards the top of the screen)



Auto-scale (automatic scaling of the model so that it fits into the graphics window)



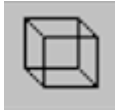
Center of the model (the location of the model which is selected will move to the center of the screen)



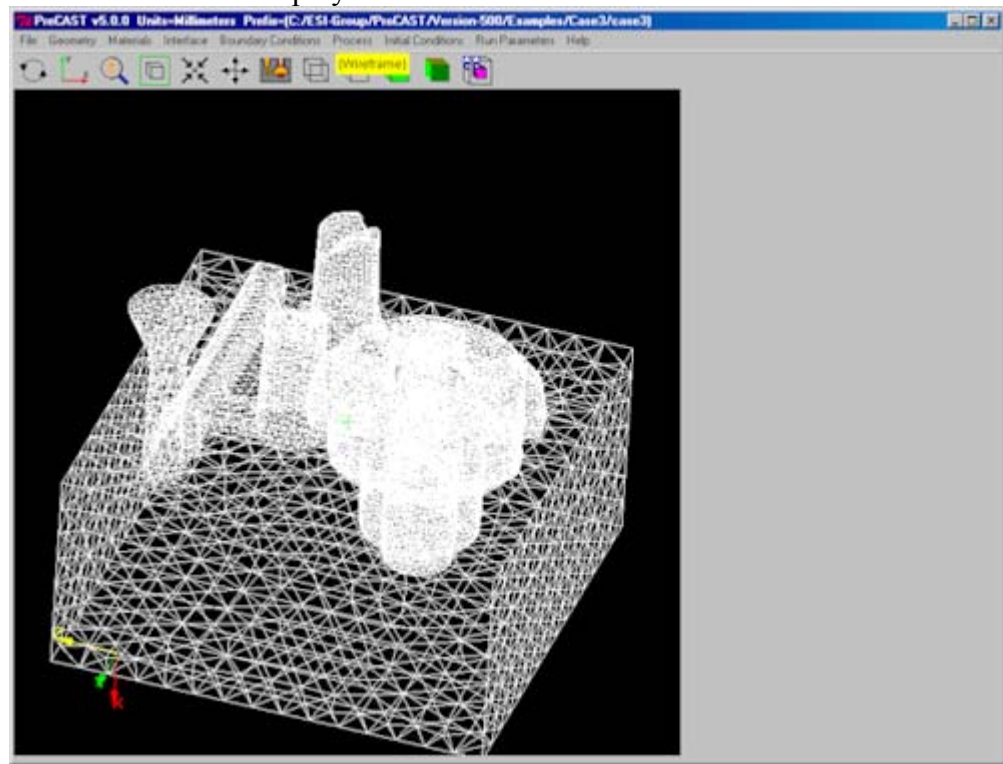
Drag of the model (interactive move of the model on the screen)



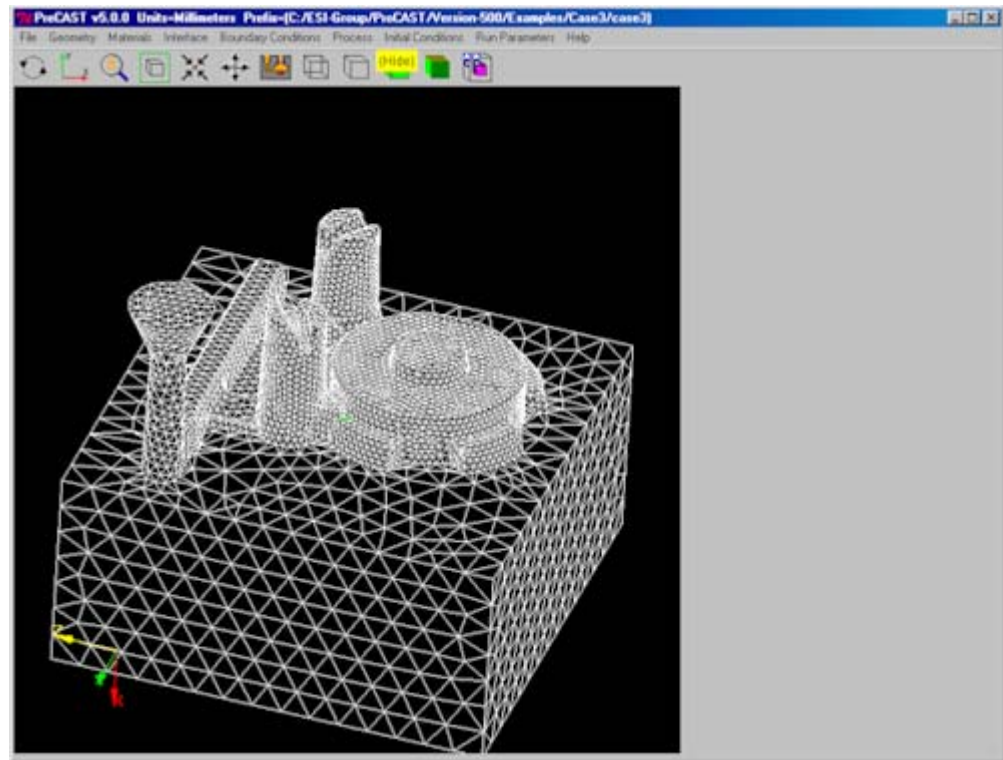
Material (or domain) selection



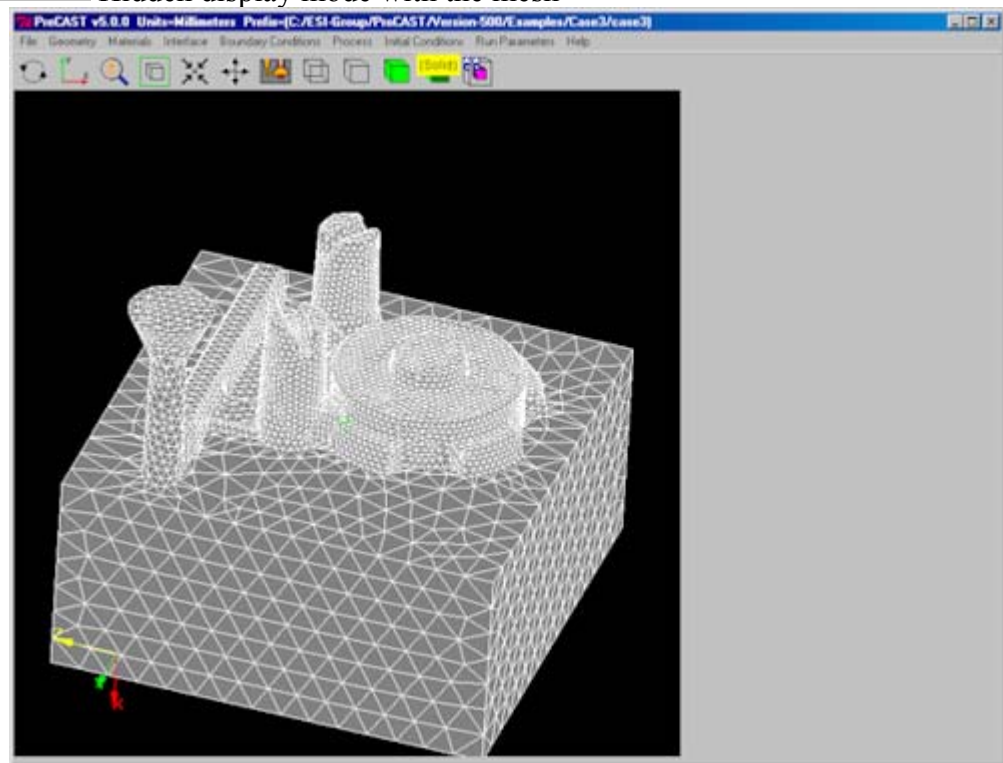
Wire frame display mode



Hidden wire frame display mode

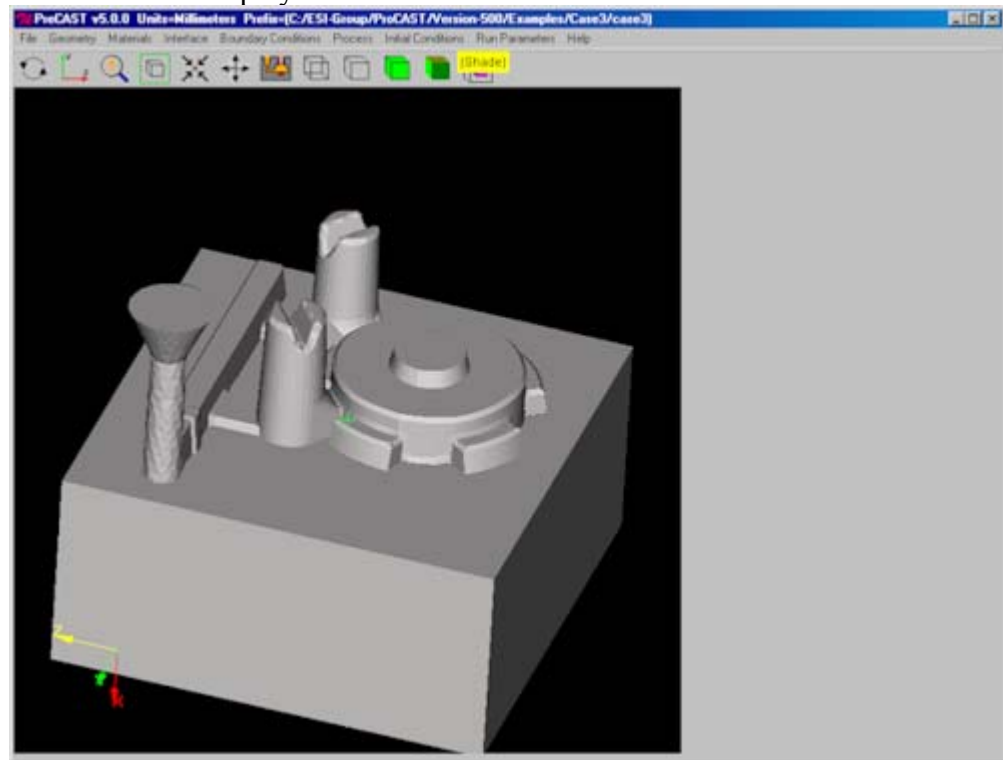


Hidden display mode with the mesh





Shaded display mode



Activation of the enclosure viewing (for radiation models)

Run of the calculation

Solver

Enter topic text here.

Troubleshooting

Enter topic text here.

Results viewing

Introduction

Enter topic text here.

Field selection

Enter topic text here.

Display parameters

Enter topic text here.

Geometry manipulation

Enter topic text here.

Curves

Enter topic text here.

Results analysis

Defects analysis

Enter topic text here.

Porosity

Enter topic text here.

Criterion functions

Enter topic text here.

Results Exports

Introduction

Enter topic text here.

Formats

Enter topic text here.

Tips & Traps

Tutorials

HPDC

Enter topic text here.

LPDC

Enter topic text here.

Gravity-Sand

Enter topic text here.

Gravity-Die

Enter topic text here.

Investment

Enter topic text here.

Tilt

Enter topic text here.

Filter

Enter topic text here.

Stress

Enter topic text here.

Lost Foam

Enter topic text here.

Thixo

Enter topic text here.

Appendices

Software installation

Enter topic text here.

Windows

Enter topic text here.

When PreCAST is launched manually, several options are available to the user :

`precast -nolock -m prefix`

Unix - Linux

Enter topic text here.

Files structure

Enter topic text here.

File names

Enter topic text here.

Input file structure (d.dat file)

Enter topic text here.

Input file structure (p.dat file)

Enter topic text here.

Models & Algorithms

Enter topic text here.

Physical models

Enter topic text here.

Numerical models

Enter topic text here.

Your index page goes here...

In MS-Word, select INDEX AND CONTENTS from the INSERT menu.
Select INDEX and click OK.