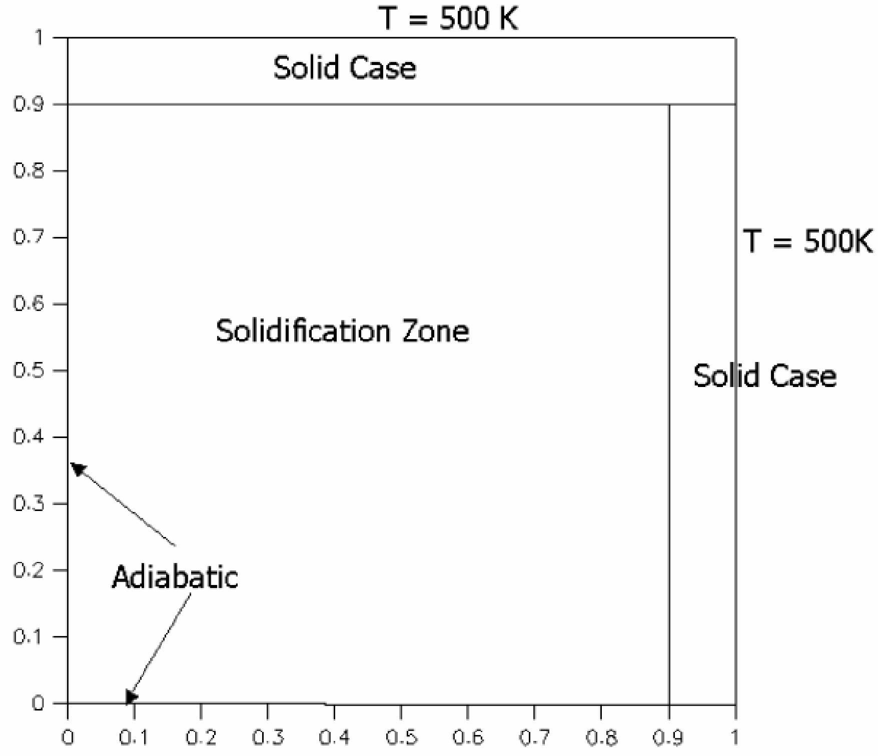
Case 1

Pure Heat Conduction, Multizone and Isothermal Case

Problem Description

Details on the computation model:



Material:	Aluminium
Time step:	10 s
Density:	2702 kg/m^3
Cp:	900 J/kg*K
Latent heat:	399903 J/kg
Solidification temperature:	933.25 K
Initial temperature:	1000 K

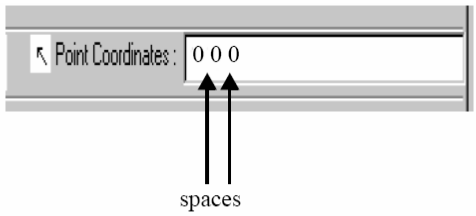
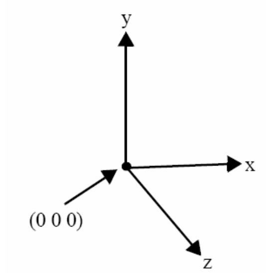
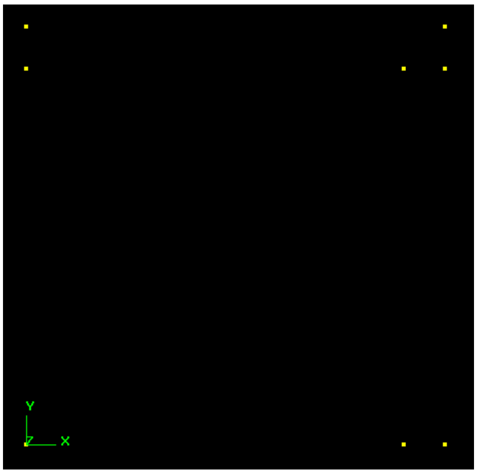
Geometry Creation

1. Create the points for the model.

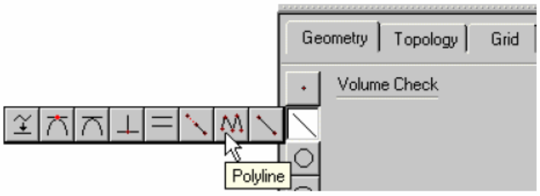
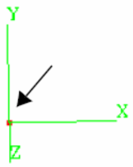
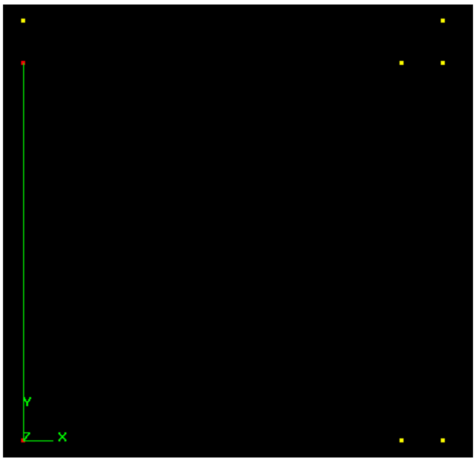
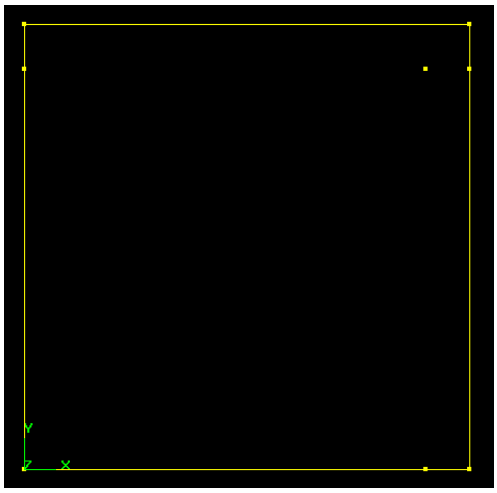
We will use the CFD- GEOM program to draw the computation model and create the structured grid, in order to import the file to the CFD-ACE+ and work with the file.

<table>
<tr><td>Select the Point / Create Point option from the GEOM toolbar.

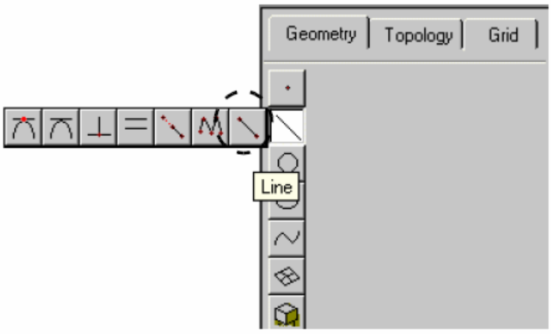
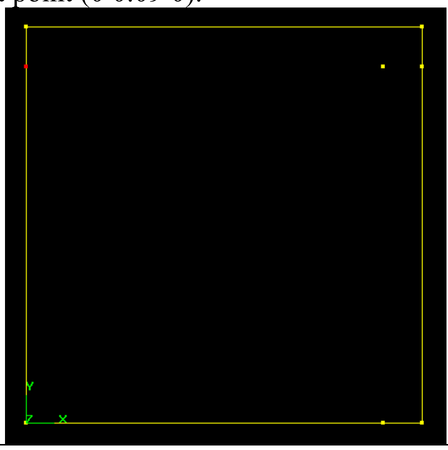
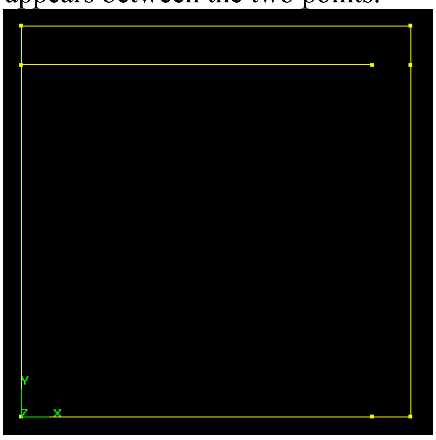
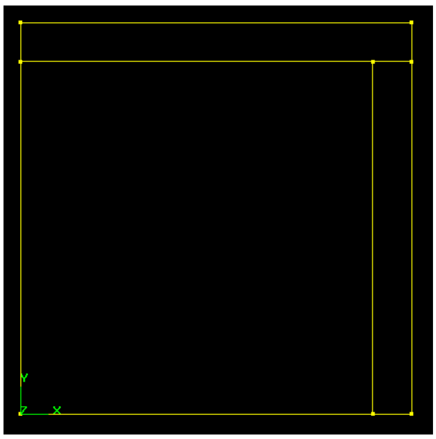
</td><td></td></tr>
<tr><td>At the bottom center of the main Graphics Window toolbar, select the Point Coordinate option icon.</td><td>A type-in box appears to the right of the icon.

</td></tr>
<tr><td>Position the cursor in the type-in box and click the left mouse button.
Enter the coordinates of point (0 0 0) into the type-in box, separating the zeroes with single spaces. Press the Return key on the keyboard after the last coordinate.

</td><td>Pressing the Return key positions a point on the screen.
(A black point appears on the screen at the origin.)

</td></tr>
<tr><td>Create the remaining points in the same manner, by entering the coordinates shown below in the type-in box. (Be sure to insert a space between the elements.)
0.09 0 0
0.1 0 0
0.1 0.1 0
0.1 0.09 0
0.09 0.09 0
0 0.09 0
0 0.1 0</td><td>Each point appears on the screen as a yellow marker.
</td></tr>
</table>

2. Connect the points using the polyline geometry tool.

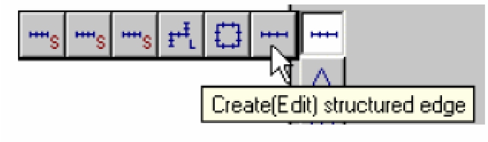
Select the *Line / Polyline* option from the GEOM toolbar. 	The Status Line at the bottom of the Graphics Window prompts you to select points to create a polyline.
Select the point at the origin in the Graphics Window by pointing at it and clicking the left mouse button. 	The point changes color from yellow to red. The Status Line at the bottom of the window prompts you to select the additional points.
Select the point (0 0.09 0) 	The point turns red and a green line appears between the points (0 0 0) and (0 0.09 0)
Continue around the perimeter of the base selecting each point until you return to the origin. After you have selected each point, press the middle mouse button.	Yellow lines appear between each of the points selected. 

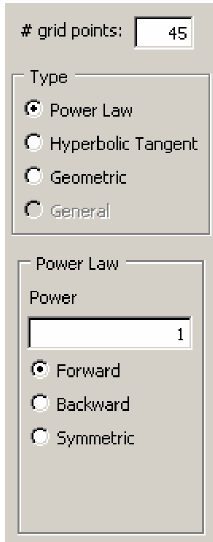
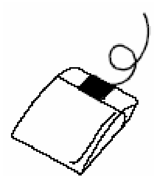
3. Create the lines across the square using the line drawing tool.

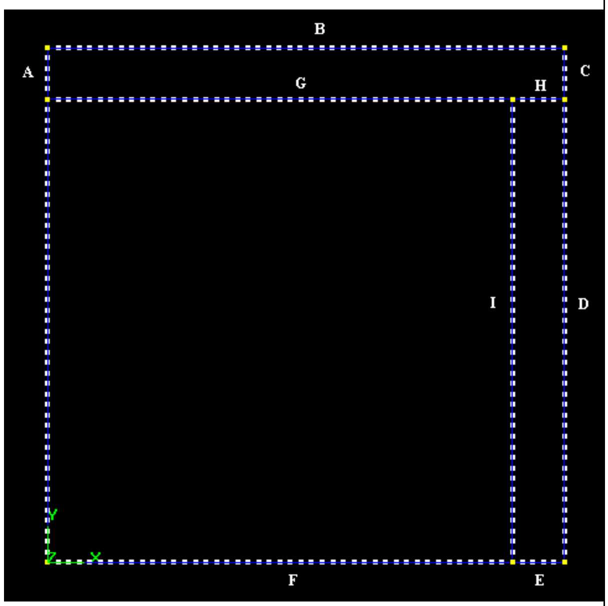
Select the Line / Line option from the GEOM toolbar. 	The Status Line prompts you to select the first point using the appropriate point mode.
Select point (0 0.09 0). 	The point turns red and the Status Line prompts you to select the second point.
Select point (0.09 0.09 0) and press the middle mouse button.	A line appears between the two points. 
Create two more lines between points (0.09 0.09 0) and (0.1 0.09 0), (0.09 0.09 0) and (0.09 0 0).	

Grid Creation

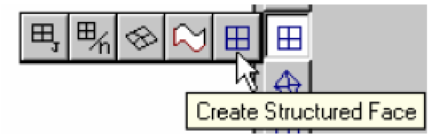
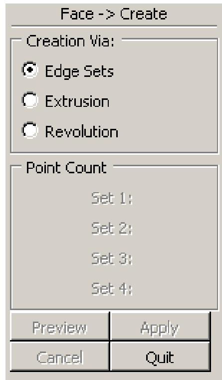
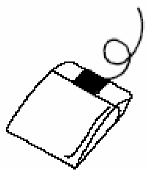
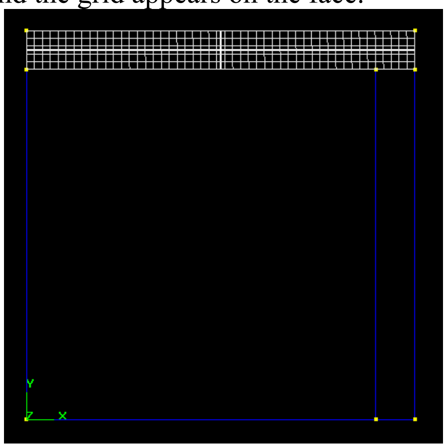
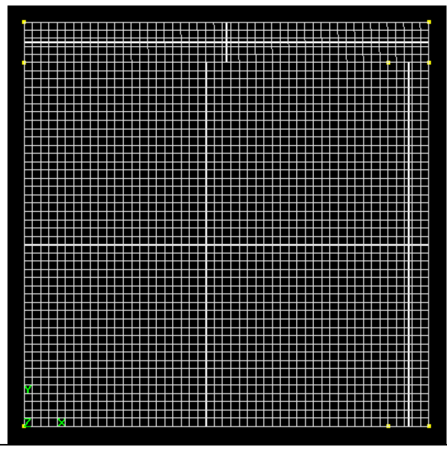
1. Create structured edges in the model.

<table>
<tr><td>Select the Edge / Create Edge option under the GRID toolbox.
</td><td>The menu shown here appears at the bottom of the toolbox.

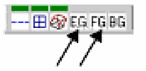
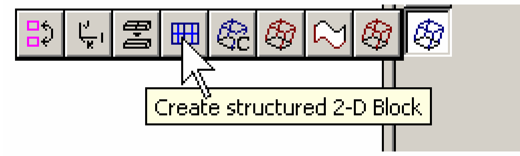
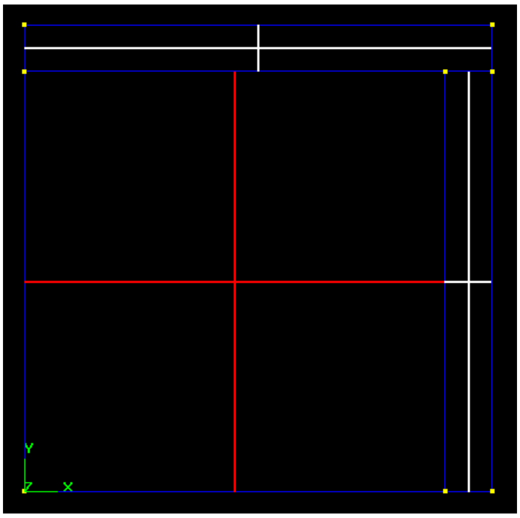
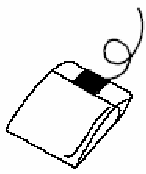
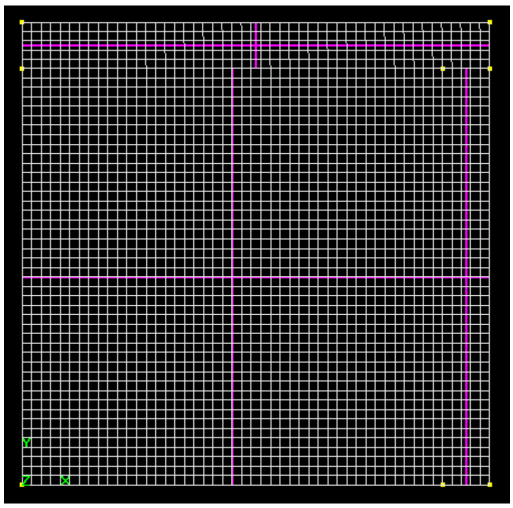
This menu allows you to specify the number of grid points and the spacing distribution on each edge you create.
</td></tr>
<tr><td>From the Graphics Window, select the line between points (0 0 0) and (0 0.09 0).</td><td>The line turns red.</td></tr>
<tr><td>Ensure that the Grid Points type-in field is set to 45 so that forty-five points are placed on the edge.</td><td>The line turns blue indicating the edge has been created.</td></tr>
<tr><td>Click the middle mouse button to create the edge.
</td><td><u>Note:</u> You do not have to re-select the Create Edge option. Once in this toolbox, you can continue to create edges until a new toolbox is selected.</td></tr>
<tr><td>Do the same for the rest of lines. Set the number of grid points as follows:

<table>
<tr><th>From</th><th>To</th><th>Line</th><th>Grid point</th></tr>
<tr><td>(0 0.09 0)</td><td>(0 0.1 0)</td><td>A</td><td>6</td></tr>
<tr><td>(0 0.1 0)</td><td>(0.1 0.1 0)</td><td>B</td><td>50</td></tr>
<tr><td>(0.1 0.1 0)</td><td>(0.1 0.09 0)</td><td>C</td><td>6</td></tr>
<tr><td>(0.1 0.09 0)</td><td>(0.1 0 0)</td><td>D</td><td>45</td></tr>
<tr><td>(0.1 0 0)</td><td>(0.09 0 0)</td><td>E</td><td>6</td></tr>
<tr><td>(0.09 0 0)</td><td>(0 0 0)</td><td>F</td><td>45</td></tr>
<tr><td>(0 0.09 0)</td><td>(0.09 0.09 0)</td><td>G</td><td>45</td></tr>
<tr><td>(0.09 0.09 0)</td><td>(0.1 0.09 0)</td><td>H</td><td>6</td></tr>
<tr><td>(0.09 0.09 0)</td><td>(0.09 0 0)</td><td>I</td><td>45</td></tr>
</table></td><td>The lines of the model are blue with the points created:
</td></tr>
</table>

2. Create a face for each square of the 2-D model.

Turn edge entities off using the entity button in the Entity Bar. 	All edge entities disappear from the Graphic Window, leaving only the blue lines and the yellow points.
Select the *Create Structured Face* option under the GRID toolbox. 	A new window opens in the Control Panel and the Status Line prompts you to pick the first of four edge sets. 
We will first create the face bounded by lines A, B, C, G and H.	
Select the edges G and H using the left mouse	The edges turn red.
Press the middle mouse button after picking all three edges to enter the edge set. **Note:** If you make a mistake, press 'q' from the keyboard and start over. 	This edge set is comprised of more than one edge.
Continue around the square selecting and entering edge sets A, B and C.	After the last edge set is entered, a white cross hair and the grid appears on the face. 
Create the remaining two faces of the block, similarly to the way you created the first. **Note:** remember that the left mouse button <u>selects</u> edges, and the middle mouse button <u>enters</u> edge sets.	The final result shown here. 

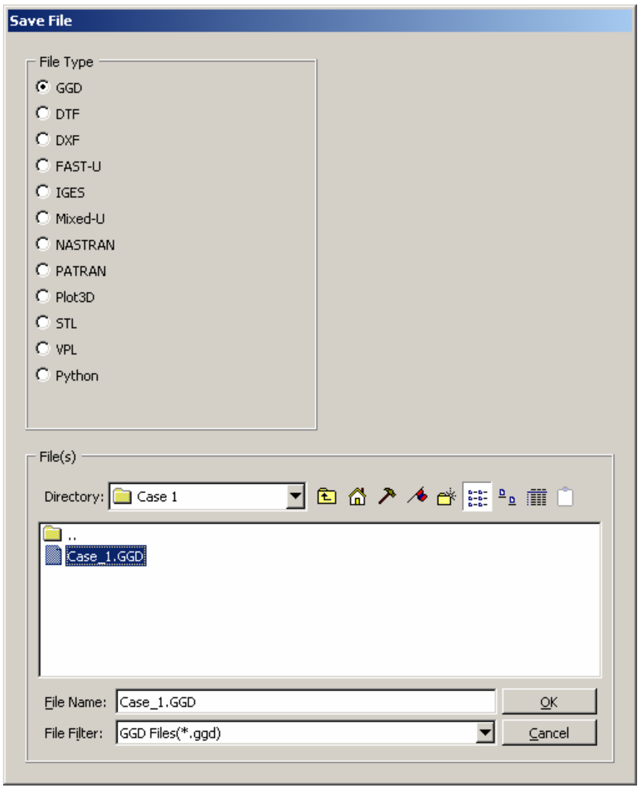
3. Create a structured 2-D block grid.

Turn edge entities and the face entities off using the entity button in the Entity Bar. 	All edge and face entities disappear from the Graphic Window, leaving only the blue lines, the face handlers and the yellow points.
Select the *Create structured 2D Block* from the *Structured Block* options located in the GRID toolbox. 	The Status Line prompts you to select faces to create a 2D block.
Select the face of the main square by placing the cursor over the white face handler and clicking the left mouse button. 	The face handler turns red as it is selected.
Click the middle mouse button to enter the face set. 	The face turns magenta. The Status Line prompts you to select the next face set. Entering face sets is similar to entering edge sets when creating faces. Also, a face set can be composed of either one face or multiple faces.
In the same manner, select and enter the two other faces.	After the two last faces are entered the 2D grid block for the model has been built. 

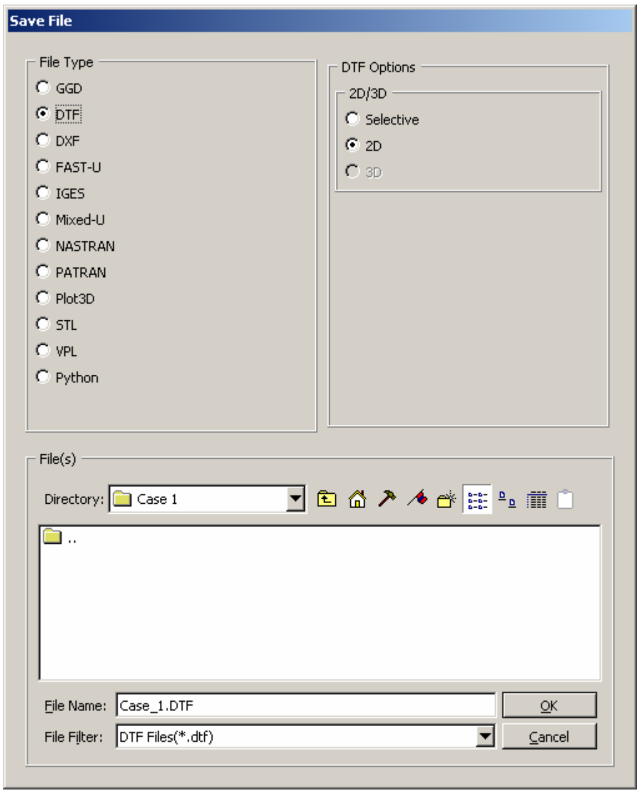
4. Save the *.GGD file and save a structured grid for use by CFD-ACE.

From the Menu Bar, select *File / Save As*.	A *Save File* window should appear with a type-in *File Name* field. The default file type is *.GGD*.

- Under File Type, ensure that the GGD options is active.
- Enter the file name *Case_1.GGD* and click the *Accept* button.



- Select *File / Save As* to re-open the Save File panel.
- Select DTF from the File Type options.
- From the DTF Options that appear on the right side of the panel, ensure that the 2D button is active.
- Change the file name to *Case_1.DTF* and click the Accept button to save.



We have finished creating the model. Now we will use the **CFD-ACE** program in order to assign the necessary conditions so the program will be able to calculate the solution.

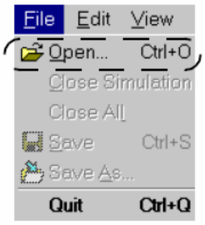
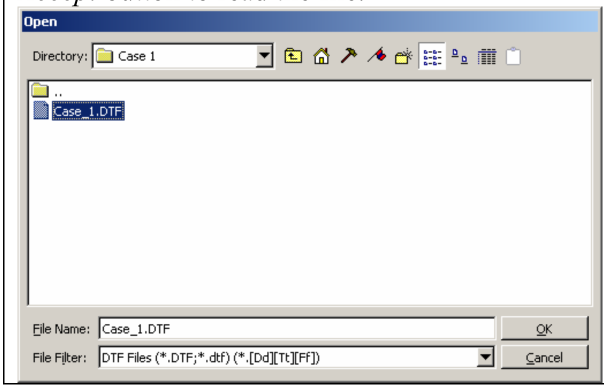
Close the **CFD-GEOM** program and open the **CFD-ACE** program.

Modules and Features Used

Modules		Major Features		Other Features	
	Flow		Gas Phase Reaction		3D
•	Heat Transfer (Heat)		Surface Reaction	•	2D Planar
	Turbulence (Turb)				2D Axisym
	Chemistry		Arbitrary Interface BC		
	User Scalar (Scalar)		Thin Wall BC		Transient
	Radiation (Rad)		Cyclic BC		
	Spray				
	Free Surface (VOF)		Fan Model		
	Two-Phase (Fluid2)		Momentum Resistance		
	Cavitation (Cav)		Rotating System		
	Grid Deformation (Deform)				
	Stress		Parallel Processing		
	Plasma		User Subroutines		
	Electric (Electr)				
	Magnetic (Magnet)				
	Kinetics				
	Semi Device				

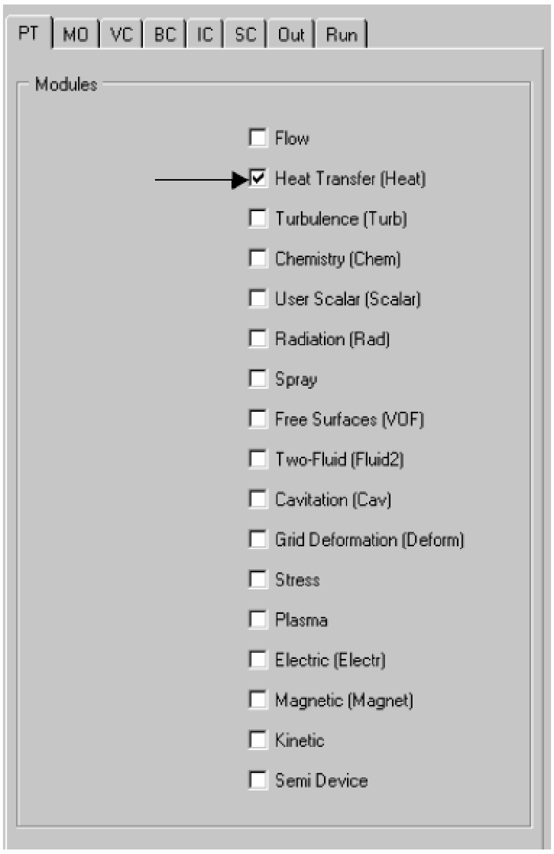
Procedure

1. Load the DTF file.

<table>
<tr>
<td>From the File Menu, select Open.

</td>
<td>A File Open dialog panel appears.</td>
</tr>
<tr>
<td>Select the file named case_1.DTF and press the Accept button to read the file.

</td>
<td>The DTF file is read into CFD-GUI and a wire-frame outline of the model appears in the viewing window.

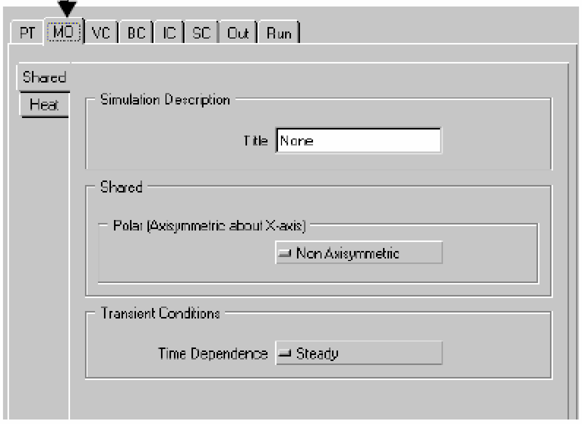
The control panel opens in the Problem Type setting mode.</td>
</tr>
</table>

2. Specify the Problem Type Settings.

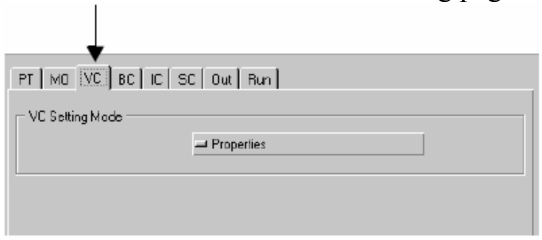
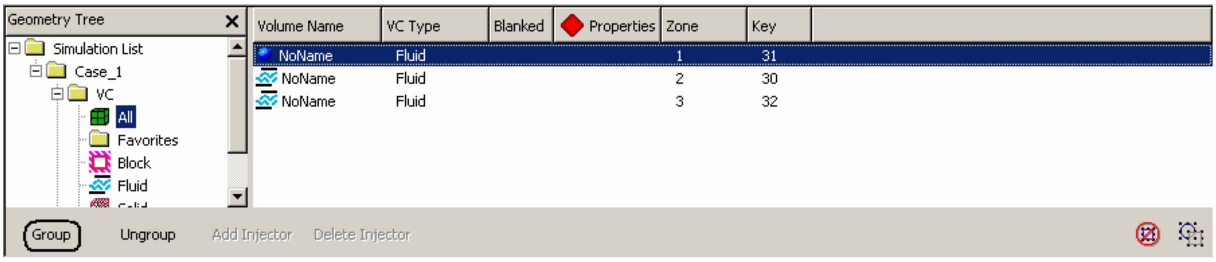
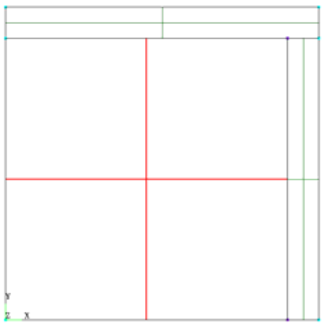
<table>
<tr>
<td>Select Heat Transfer from the Available Modules section of the Problem Type [PT].

</td>
<td>The Heat Transfer Module is the only module needed for this simulation.

Activation of the Heat Transfer Module implies solution of the total Enthalpy equation for this 2D simulation.</td>
</tr>
</table>

3. Specify the Model Options.

Press the *Model Options* [MO] tab to activate the Model Options setting page. 	The Model Options page has a Shared tab and an Adv tab which contains parameters that are available globally; and a tab for each of the Modules that were activated earlier. In this case, only the Heat tab will be visible.
Select the *Shared* tab. Ensure the following: • *Polar* is set to *Non Axisymmetric* • *Transient Conditions* is set to *Time Dependence, Transient*: *Transient Time Step* is set to *Standard.* *No. of steps* is 100. *Time step* is 2s. *Time Accuracy* is set to *Euler (1st Order).*	These should be the default settings. Not thus for the Transient Conditions and Solidification (Heat tab), they have to be changed. The Ice Melting option can be used to simulate the heat transfer needed for phase change, the Moving Solid option is used to allow convection effects to be present in a translating or rotation solid.
Under the *Heat* tab ensure the following: • *Ice Melting* is off. • *Solidification* is on.	
Under the *Adv* tab ensure the following: • The *Moving Solid* is off.	

4. Specify the Volume Condition Properties.

<table>
<tr><td>Press the Volume Conditions [VC] tab to activate the Volume Conditions setting page.

</td>
<td>The Volume Conditions Page opens and the Model Explorer changes to the VC mode to list all of the volume conditions in the currently loaded simulation.</td></tr>
<tr><td>Ensure that the Setting Mode is set to Properties.</td>
<td>The Volume Condition Page has several setting modes. The Properties mode allows us to assign the properties of each volume condition.</td></tr>
<tr><td colspan="2">A marker is activated in the headings bar of the Model Explorer next to the Property heading to let you know that you are working in Property setting mode.

</td></tr>
<tr><td>Pick the cross of the main square in the Graphics Window.
</td>
<td>The cross turns red.

The volume conditions picked is highlighted in reverse video in the Model Explorer list. It's also highlighted by thick lines in the viewing window.</td></tr>
<tr><td>Volume Conditions Setting Mode:

Properties are set to Fluid.
Fluid Subtype is set to Liquid.
Liquid Material Name is Aluminium.

Under the Phys tab ensure the following:

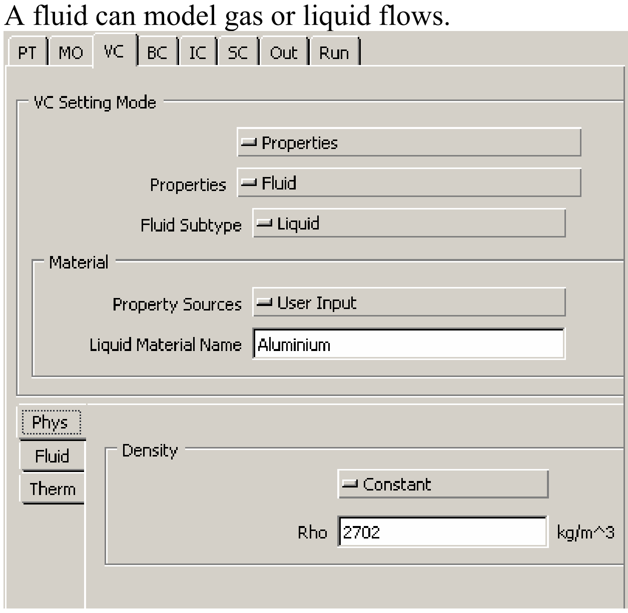
Set Density to a const value of 2702 kg/m^3.

Under the Fluid tab ensure the following:

Set the Viscosity to a constant kinematic value of 28.6 m^2/s.

Under the Therm tab ensure the following:

Specific Heat is set to 900 J/kg*K.
Thermal Conductivity is set to 23,7 W/mK.
Solidification is set to Isothermal. Values:
Latent Heat = 399903 J/kg
Temperature = 933.25 K

Press the Apply button to accept the values.</td>
<td>A fluid can model gas or liquid flows.

</td></tr>
</table>

<table>
<tr><td>Select the two other crosses.</td><td>Select a volume condition handle in the viewing window and then Ctrl-Select the other volume condition handles in the viewing window.</td></tr>
<tr><td>To permanently group the selected items together, press the Group button located in the lower left corner of the viewing window.

This is an optional step and is recommended as it ensures that all of the volume conditions in the group will always receive the same property settings.</td><td>The items are now part of a permanent property group. A group name is given in the property column of the model explorer.

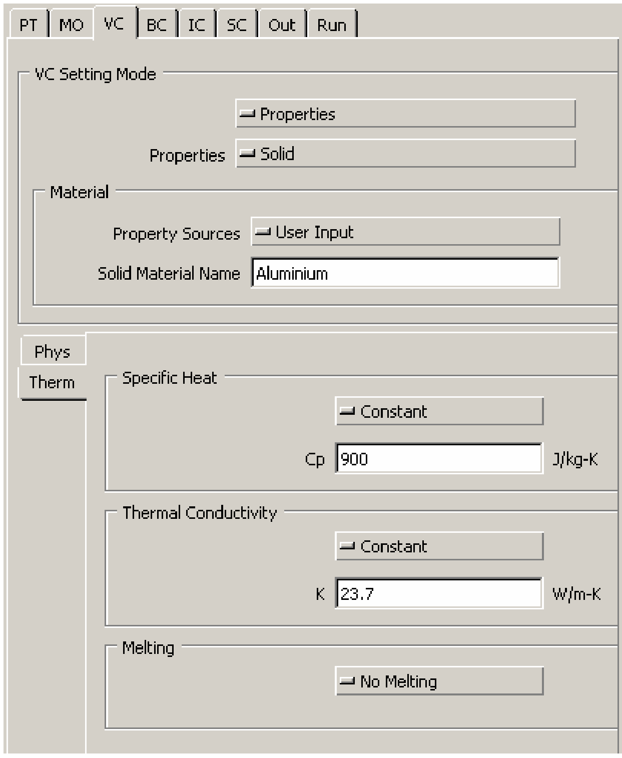
A permanent property group implies that whenever you are in properties setting mode, if you select any volume that is part of the group, then all members of the group will be automatically selected.

To ungroup them you can select the Ungroup button located in the lower left corner (next to the Group button).</td></tr>
<tr><td>Volume Conditions Setting Mode:
• Properties are set to Solid.
• Solid Material Name is Aluminium.

Under the Phys tab ensure the following:
• Density is set to Constant.
• Rho is 2702 kg/m^3.

Under the Therm tab ensure the following:
• Specific Heat is set to 900 J/kg*K.
• Thermal Conductivity is set to 23,7 W/mK.
• Melting is set No Melting.

Press the Apply button to accept the values.</td><td>A solid type allows heat transfer but no flow.

Volume Conditions:

</td></tr>
</table>

5. Specify the Boundary Condition Values.

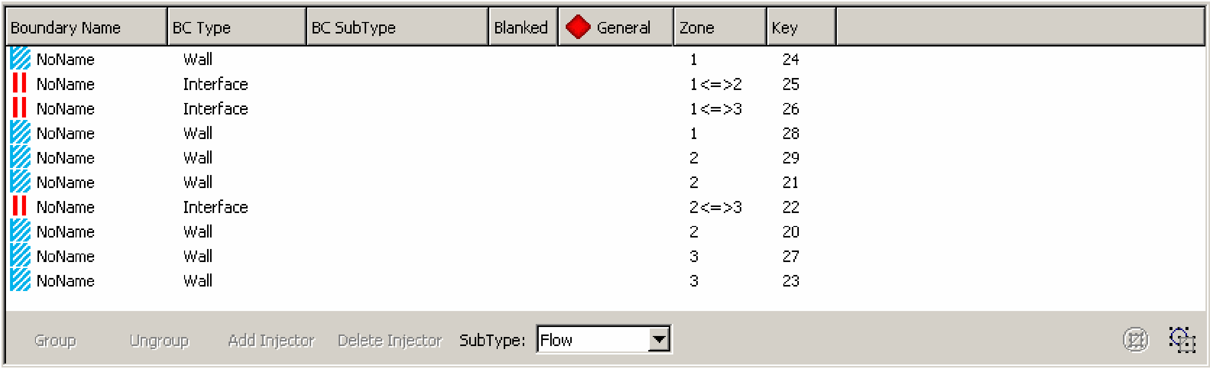
Press the *Boundary Conditions* [BC] tab to activate the Boundary Conditions setting page.

The Boundary Condition Page opens and the Model Explorer changes to the BC mode to list all of the boundary conditions in the currently loaded simulation.

Ensure that the *Setting Mode* is set to *General*.

The Boundary Condition Page has several setting modes. The General mode allows us to assign the values of each boundary condition.

A marker is activated in the headings bar of the Model Explorer next to the General heading to let you know that you are working in General setting mode.



Boundary Name	BC Type	BC SubType	Blanked	General	Zone	Key
NoName	Wall				1	24
NoName	Interface				1<=>2	25
NoName	Interface				1<=>3	26
NoName	Wall				1	28
NoName	Wall				2	29
NoName	Wall				2	21
NoName	Interface				2<=>3	22
NoName	Wall				2	20
NoName	Wall				3	27
NoName	Wall				3	23

Group Ungroup Add Injector Delete Injector SubType: Flow

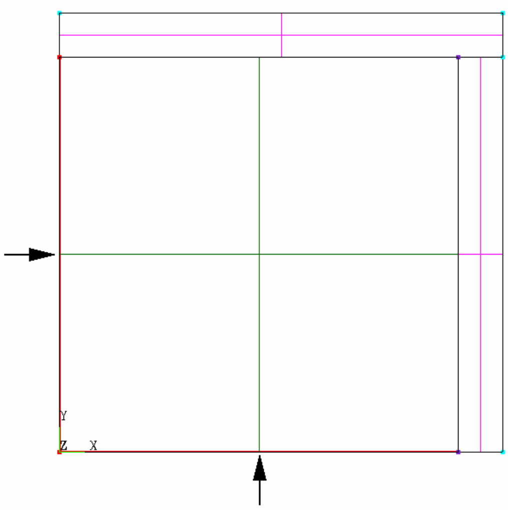
The Model Explorer list can be sorted based on the contents of any of the columns of data. Click on any column heading with the left mouse button and the list will be sorted in ascending order based on that column's contents. Click on the same column again and the sort order will be reversed.

The Model Explorer is an efficient way to find and select boundary conditions (or volume conditions) of interest.

5.a. Select the adiabatic boundary and set the values.

There are two ways to select the adiabatic boundary condition:

1. From the Model Explorer, select the entries of the Zone 1.
2. Use the left-mouse button to select the edges from the bottom and from the left in the viewing screen.

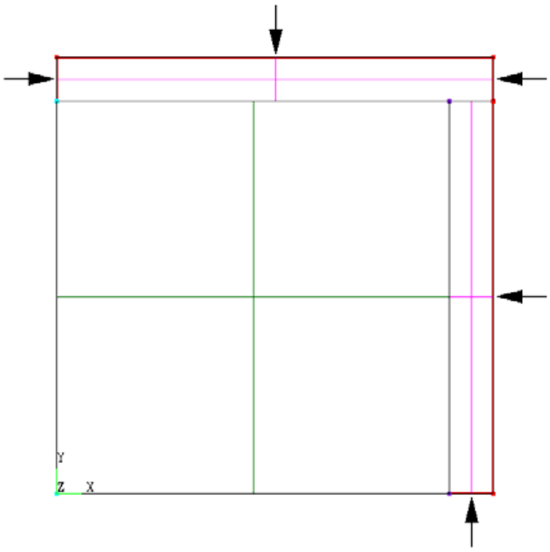


The boundary condition picked is highlighted in reverse video in the Model Explorer list. Also, in the viewing window thick lines highlight the adiabatic walls of the geometry.

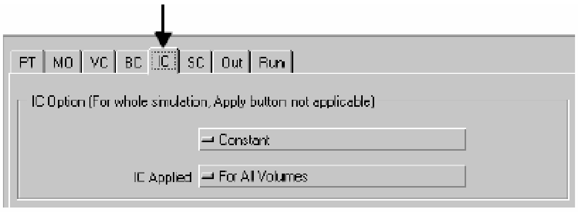
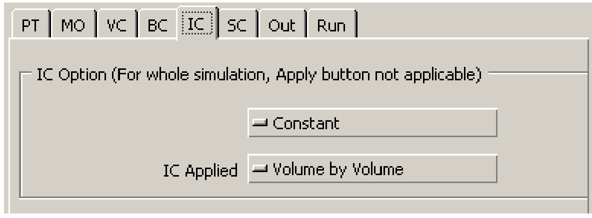
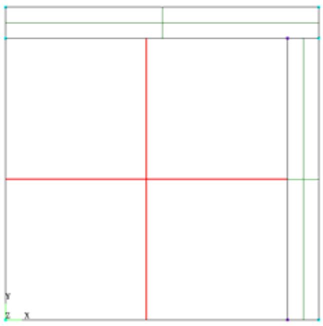
The Control Panel displays the boundary condition type. The boundary condition value settings are shown in a tabbed list. There is one tab for every module is activated. In this particular case only the Heat tab is shown.

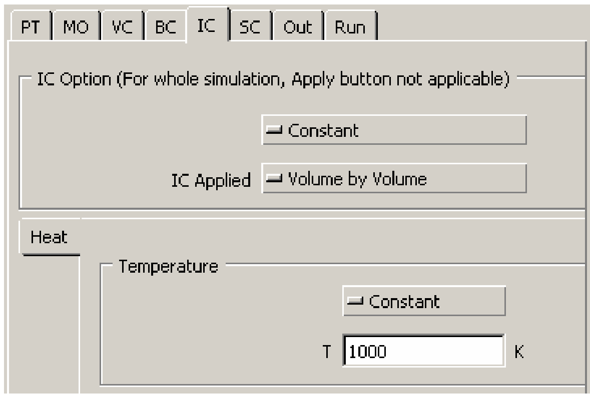
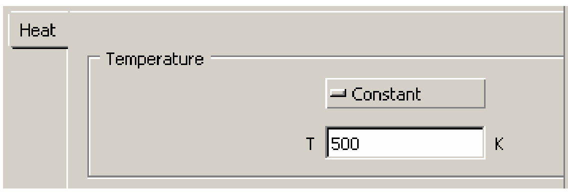
To permanently group the selected items together, press the Group button located in the lower left corner of the viewing window.	The items are now part of a permanent property group. A group name is given in the property column of the model explorer.
Ensure that the *Heat* tab is set to *Adiabatic*.	The default SubType for a Wall boundary condition is Adiabatic.
Press the *Apply* button if it's necessary and deselect the boundary condition.	

5.b. Select the isothermal boundary and set the values.

There are two ways to select the isothermal boundary condition: 1. From the Model Explorer, select the entries of the zone 2 and 3. 2. Use the left-mouse button to select the edges from the top and from the right in the viewing screen. 	The boundary condition picked is highlighted in reverse video in the Model Explorer list. Also, in the viewing window thick lines highlight the adiabatic walls of the geometry. The Control Panel displays the boundary condition type. The boundary condition value settings are shown in a tabbed list. There is one tab for every module is activated. In this particular case only the Heat tab is shown.
To permanently group the selected items together, press the Group button located in the lower left corner of the viewing window.	The items are now part of a permanent property group. A group name is given in the property column of the model explorer.
• In the *Heat* tab, select *Isothermal* from the menu. • Set the wall temperature to a constant value of 500 K.	For this simulation we want to specify the temperature at the wall so we changed the SubType to Isothermal. Once Isothermal is selected then we have the opportunity to specify the temperature of this wall. The solver will maintain this wall temperature at 500 K.
Press the Apply button to accept the values and deselect the boundary condition.	

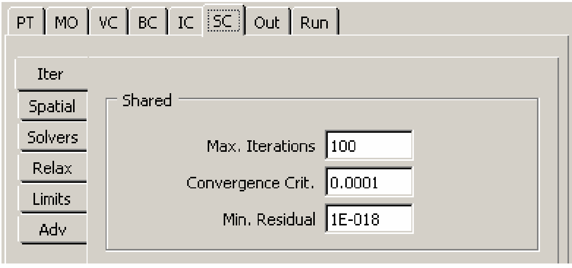
6. Specify the Initial Condition Settings.

<table>
<tr><td>Press the Initial Conditions [IC] tab to activate the Initial Condition setting page.

</td><td>The Initial Condition Page is presented.</td></tr>
<tr><td>
Ensure that the Initial Source is set to Constant.
Change the IC Global Setting to Volume by Volume.</td><td>
</td></tr>
<tr><td>Pick the cross of the main square in the Graphics Window.
</td><td>The cross turns red.

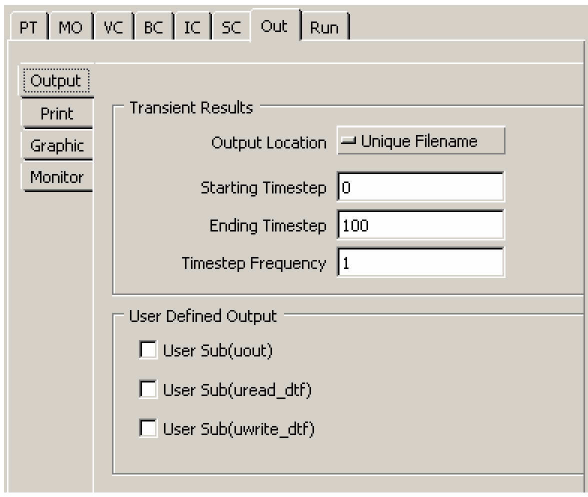
The volume conditions picked is highlighted in reverse video in the Model Explorer list. It’s also highlighted by thick lines in the viewing window.</td></tr>
<tr><td>
Set the initial condition value for Temperature equal to 1000 K.
Click Apply.</td><td>
</td></tr>
<tr><td>Select the two other crosses and press the Group button located in the lower left corner, in order to permanently group the selected items together.</td><td>Select a volume condition handle in the viewing window and then Ctrl-Select the other volume condition handles in the viewing window.</td></tr>
<tr><td>
Set the initial condition value for Temperature equal to 500 K.
Click Apply.</td><td>
</td></tr>
<tr><td colspan="2">Constant implies that the every cell in the computational domain will use the specified values as an initial condition.

Reasonable initial conditions will help prevent divergence in the first few iterations.</td></tr>
</table>

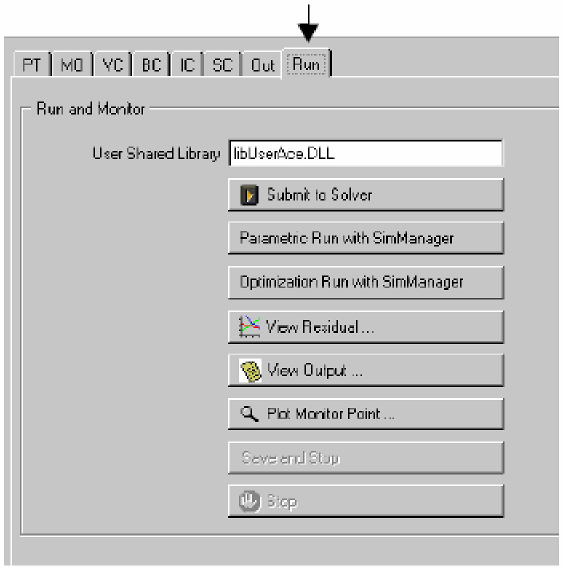
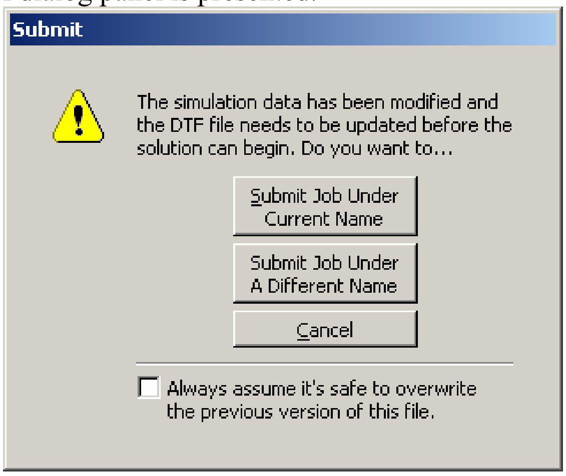
7. Specify the Solver Control Settings.

Press the *Solver Control* [SC] tab to activate the Solver Control setting page.	The Solver Control Page is presented. There is a tab for each major type of solver control setting. There are also tabs for the solver output options available from this page.
Under the *Iterations* tab, set the *Max. Iterations* to 100.  	This instructs the solver to run through the main iteration loop 100 times. The full set of equations will be solved each iteration and we will look for a 3-5 order of magnitude drop in the residuals for each equation solved.
Under the relax tab, set the inertial relaxation values for the enthalpy equation to 0.05. Also verify that the linear relaxation values for the auxiliary variable temperature are set to 1.0.	The default inertial relaxation parameter is 0.05 for the enthalpy equation. Decreasing this value can make the solution unstable so caution should be exercised.

8. Specify the Output Options.

Press the *Output* [Out] tab to activate the Output setting page.	The Output Page is presented. There is a tab for each major type of output setting.
Under the *Output* tab, select: • *Starting Timestep* is 0 • *Ending Timestep* is 100 • *Timestep Frequency* is 1	
Under the *Print* tab, select any desired text based output to be printed to the model.out file. For this case the *Heat Transfer* summary is often beneficial to determine a level of convergence.	See the CFD-ACE(U) User's Manual for more details on the types of printed output that are available.
Under the Graphics tab, select any desired variables to be output to the DTF file for postprocessing in CFD-VIEW. For this case the Static Temperature, the Thermal Conductivity and Wall Heat Flux are of interest.	These variables will be written to the DTF file at the frequency specified under the Output tab. The variables will be interpolated to the computational nodes from the cell-centered values calculated by the solver.

9. Run the Simulation.

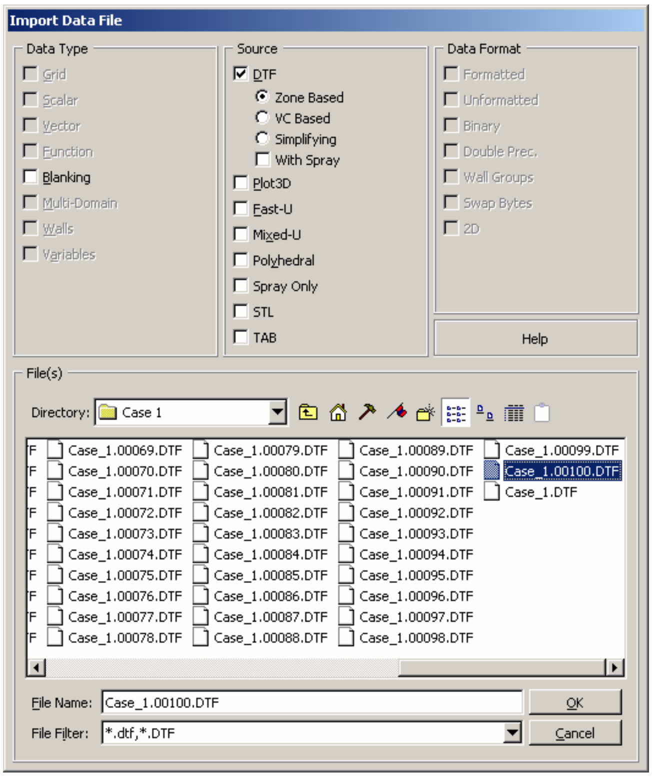
Press the *Run* [Run] tab to activate the Run Control page. 	The Run Control Page is presented.
Press the *Submit to Solver* button to start the solution process.	A dialog panel is presented. 
Because we have modified the simulation data during the solution setup process, the data must first be saved to a DTF file before the solver can start. Press the *Submit Job Under Current Name* button to save the information back to *Case_1.DTF* and to launch CFD-ACE(U) using that same name.	The data is saved to the *Case_1.DTF* file and the solver is started.
You can press the *View Residual*... and *View Output*... buttons to see real-time displays of the residual history and output file contents.	We are looking for a three to five order of magnitude drop in the solution residuals.

We have finished processing the model. Now we will use the **CFD-VIEW** program in order to post process the solution given by the **CFD-ACE** program.

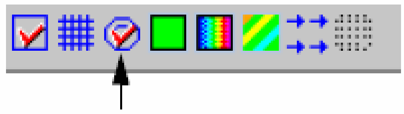
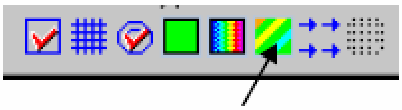
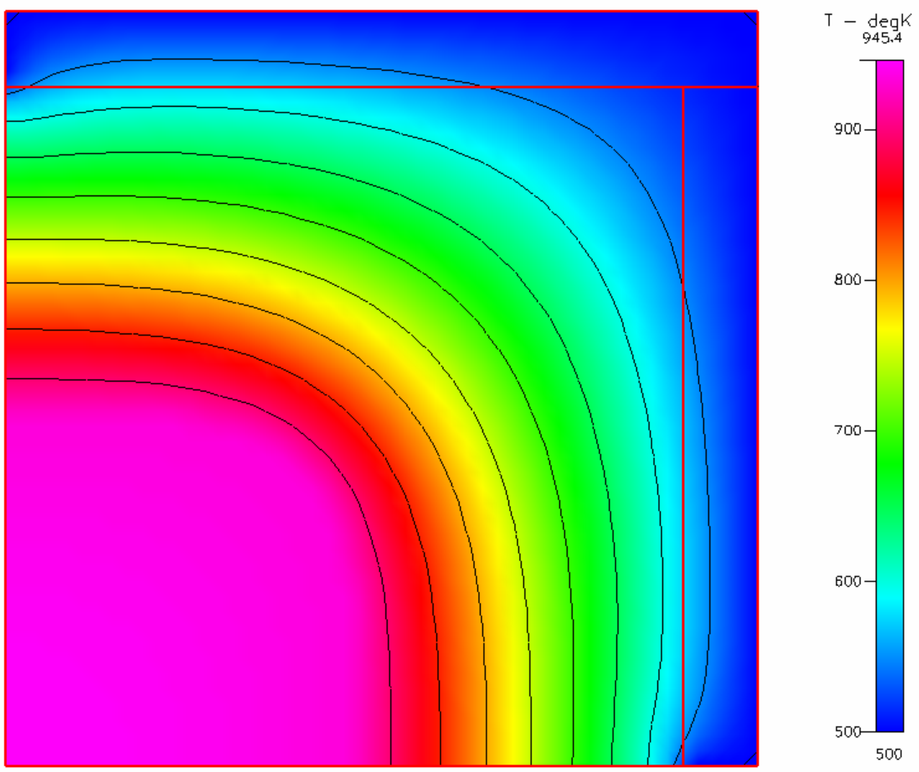
Close the **CFD-ACE** program and open the **CFD-VIEW** program.

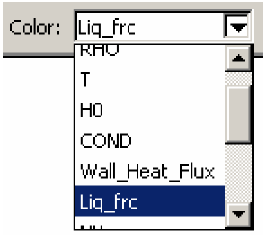
Post Processing with CFD-VIEW

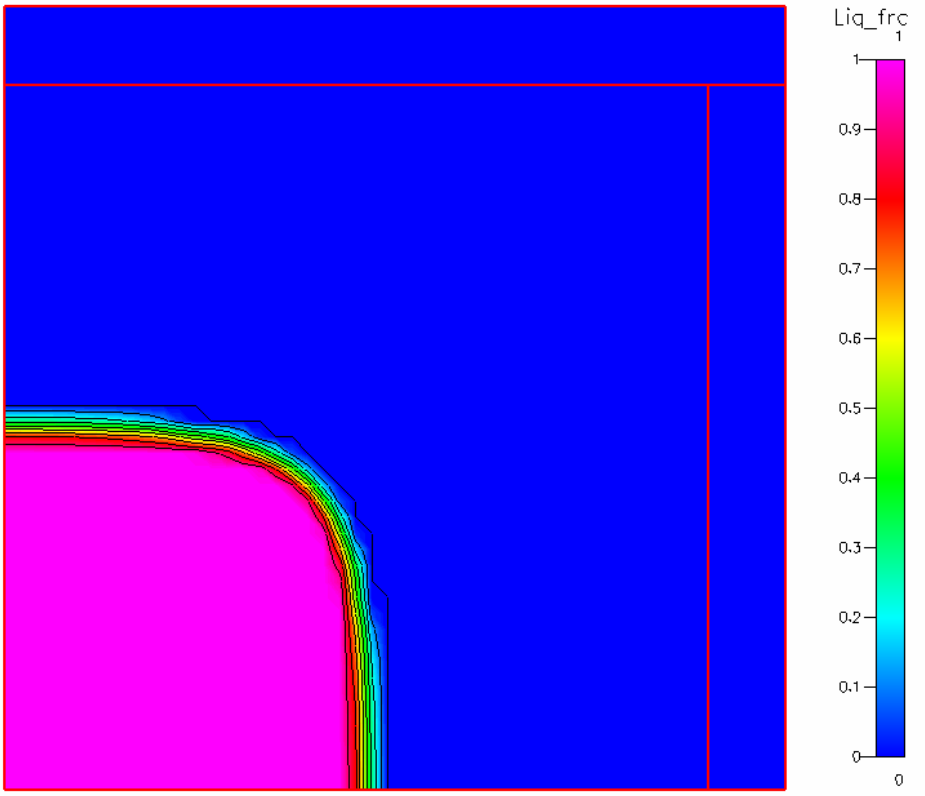
1. Read the DTF file.

Click *File / New Model.* Click *File / Import Additional Data File* from the *Menu Bar*.	A new 2-D Viewer window opens. An Open Data File panel opens (see figure below).
	
Select *DTF / Zone Based* from *Source* and Select the file *Case_1.00100.DTF* from file selection box, click *OK*.	The DTF file is read. An outline of the Case_1 geometry is displayed in the Graphics Viewer Window.

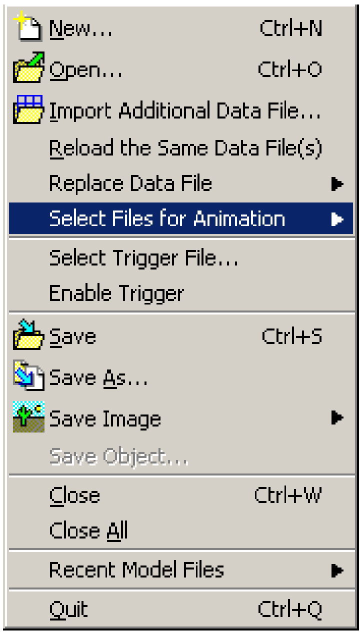
2. Plot contours of primary variable 'T'.

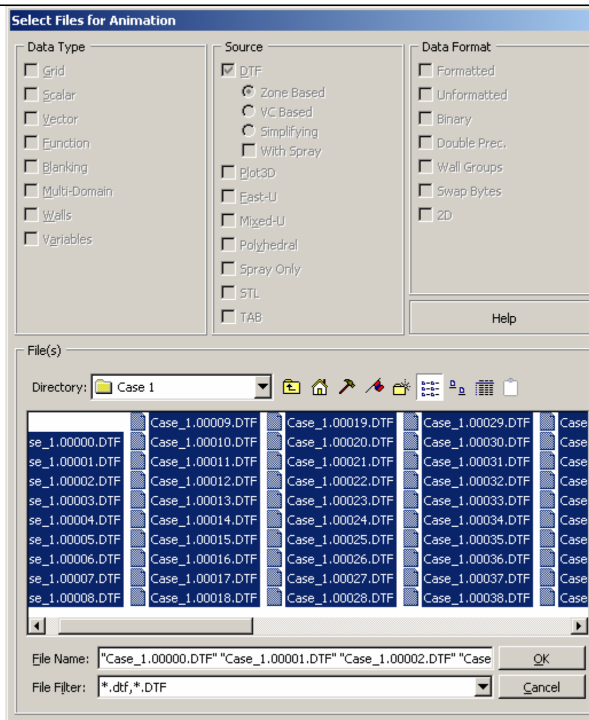
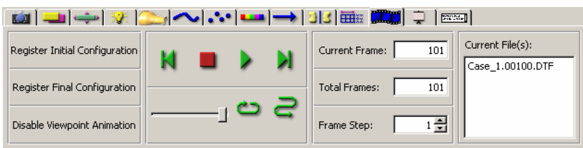
Click *Select All Surfaces* and *Group* the three selected surfaces *Zone#1*, *Zone#2* and *Zone#3*.	
Select the *Contour* button.	
From the *Visualization Panel*, select 'T' from the *Color* pull-down menu.	The contour lines for the active object appear and are now colored by the temperature variation.
Select the *Flooded Contour* button from the *Visualization Panel*.  **Note:** you can also click the *Smooth Contour* button or the *Colored Surface* button.	The area between the contour lines is now flood-shaded with the primary variable color. Notice that the contours lines changed to a solid color. When contour lines are displayed in combination with a solid surface, the lines use a solid color.
Note: Since we have shaded the model with a colormap associated with a primary variable, it would be be appropiate to display a color legend showing the variation of the values of the primary variables. You can do this with the Legend object.	
From the *Objects Palette*, click the *Legend* button.	A new entry for the Legend is placed in the Object box.
To move the legend to a new location, positioning the cursor over the legend and hold down the left mouse button. Resize the legend by selecting and dragging one of the green corner points.	The legend follows the mouse movement until the mouse button is released. If the menu becomes wider than it is long, it will switch to a horizontal format.
Temperature distribution at t=200s. 	

From the Primary Variable pull-down menu, select other variables and notice how the contour lines change.	
Liquid fraction distribution at t=200s.	



3. Animation of the Solidification Process.

<table>
<tr><td>Click File / Select Files for Animation from the Menu Bar.</td><td>
</td></tr>
<tr><td>Select all the files from Case_1.00000.DTF to Case_1.00100.DTF from the file selection box. Click OK.

A new box appears and prompts you to click yes, in order to confirm the selection we had done.</td><td>
</td></tr>
<tr><td>Select the Animation tab in the Attributes Panel.</td><td>The Animation Page is presented.

</td></tr>
<tr><td>Click the Play button.</td><td>The Graphical Window displays the sequence of the process from t=0s to t=200s.</td></tr>
</table>

This example is now finished.