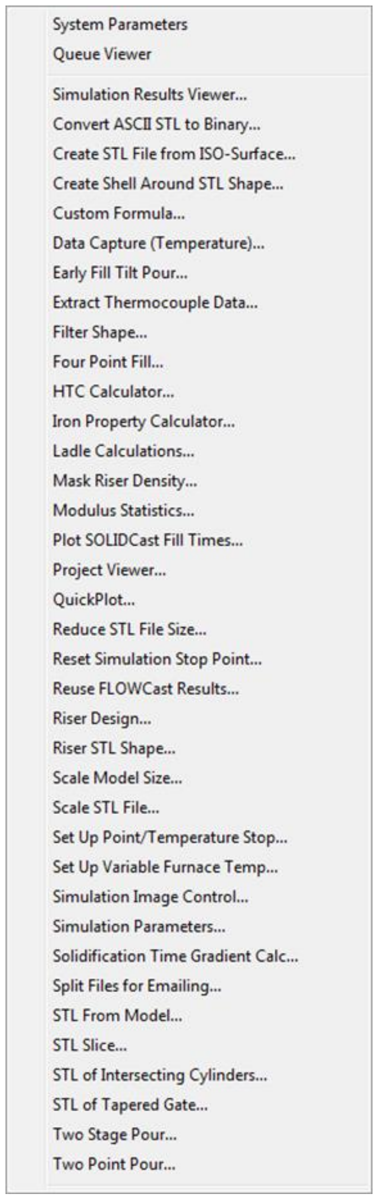
UNIT 46: SOLIDCast Utilities

Over time, a number of special purpose utilities have been created for use with SOLIDCast. These programs cover many topics; some are used in model building, some in file handling/conversion, and others for specific special purposes.

These utilities have been bundled with the SOLIDCast package. When you install SOLIDCast or update the program, these utilities are also installed and are accessible while running SOLIDCast from the Tools Menu. You need to have a model loaded first. Then, when you click on Tools, you should see something like this:



Descriptions of how each utility functions are given on the following pages.

UTILITY: Convert ASCII STL to Binary

This utility converts an ASCII format STL file to a BINARY format STL file.

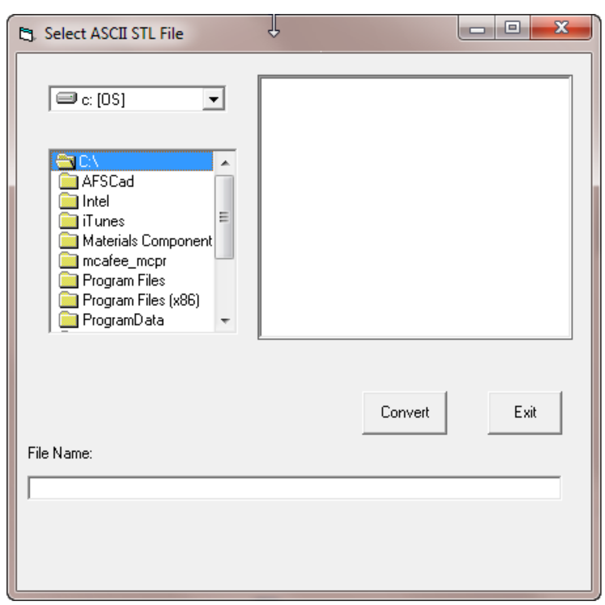
STL file formats are normally either binary or ASCII. A binary file contains data which is encoded. An ASCII file is a text file. If you open an ASCII STL file with a text editor (like Windows WordPad) you will see readable data like the following:

```
solid 139_6106_CAM
 facet normal 1.000000e+00 0.000000e+00 0.000000e+00
  outer loop
   vertex 6.000000e+01 1.450745e+02 -2.920325e+01
   vertex 6.000000e+01 1.450745e+02 -9.229675e+01
   vertex 6.000000e+01 1.680000e+02 -9.975000e+01
  endloop
 endfacet
 facet normal 0.000000e+00 -3.091782e-01 -9.510041e-01
  outer loop
   vertex -2.090000e+01 1.680000e+02 -9.975000e+01
   vertex 6.000000e+01 1.680000e+02 -9.975000e+01
   vertex 6.000000e+01 1.450745e+02 -9.229675e+01
  endloop
 endfacet
```

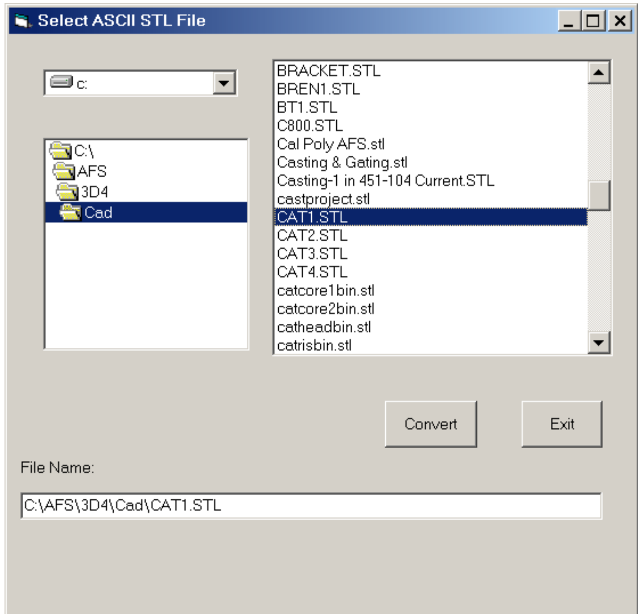
A binary file will be unreadable if opened with a text editor.

SOLIDCast requires that STL files be in binary format in order to be loaded into the model builder. Therefore, if you have an ASCII STL file, you need to convert it to binary format before using it. This can be done with the Convert ASCII STL to Binary utility.

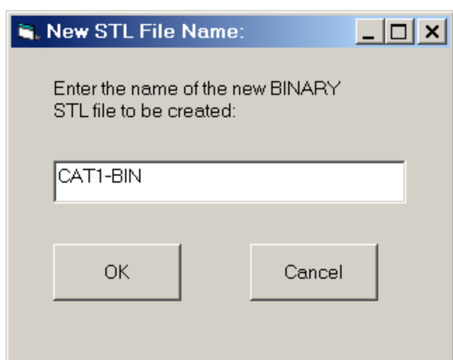
To use this utility, first load a model. Then select Tools…Convert ASCII STL to Binary…. You will see a screen like the following:



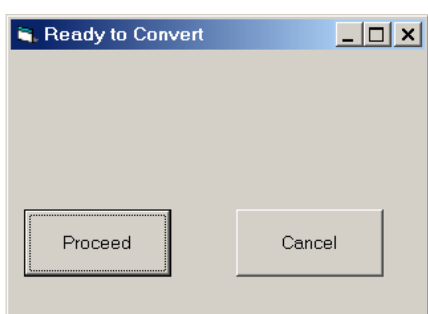
Navigate to the STL file you want to convert, and select it in the file list window as shown below:



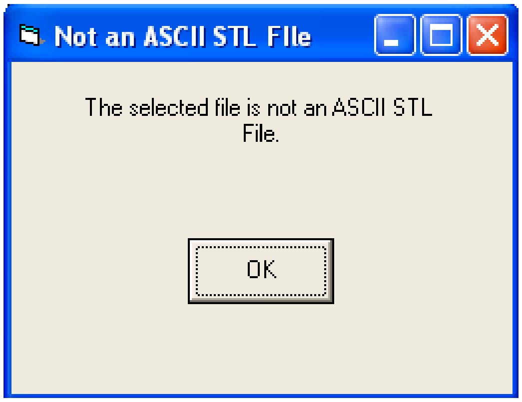
Now click the Convert button. You will be asked to enter a name for the new binary STL file.



Enter a name for the new file. (The original file will be maintained.) Then click the OK button. The system will verify that you want to do this by displaying the following:



Click on Proceed. A new file will be created in binary format from the ASCII file. The Percent Complete and number of triangles processed will be displayed during the conversion. The new binary file will exist in the same folder as the original, and can now be loaded into SOLIDCast as an object. If the selected file is already in binary format, the following message will be displayed:



Converting from an ASCII format to binary format DOES NOT AFFECT THE SHAPE; the only difference between the file types is in the way that the surface data is stored in the file. A binary STL file is approximately one fifth the size of an ASCII STL file.

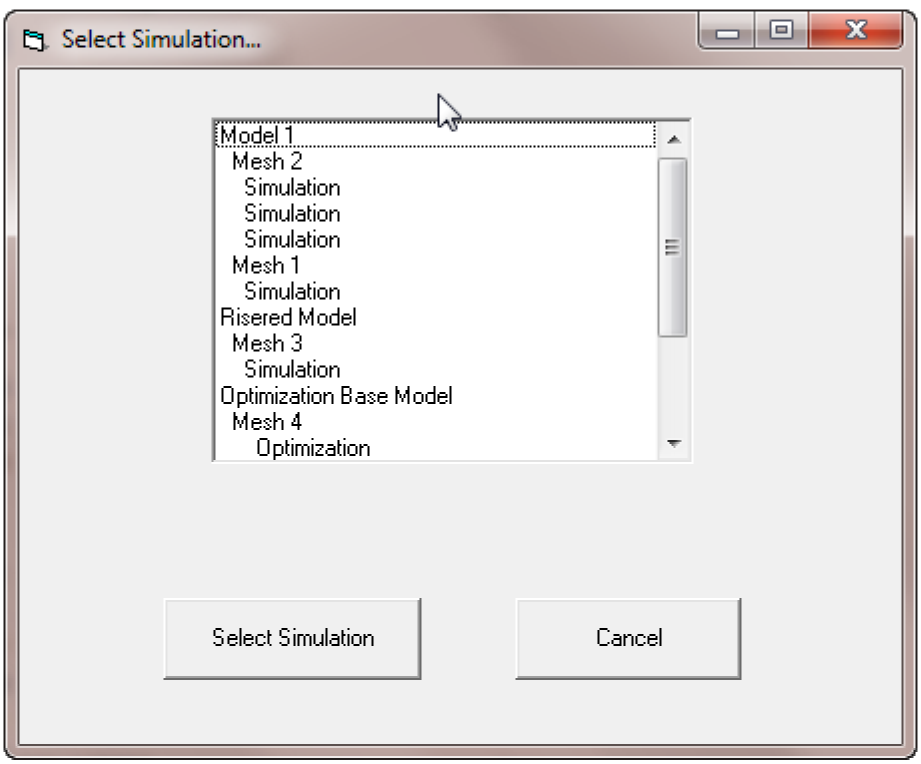
UTILITY: Create STL File from ISO-Surface

This utility allows you to take ISO-Surface plot data and export it as an STL file. This was so that it would be possible to take SOLIDCast information and place it into a CAD model in the original CAD system, to facilitate placement of risers, etc.

To use this feature, you must first have completed a simulation.

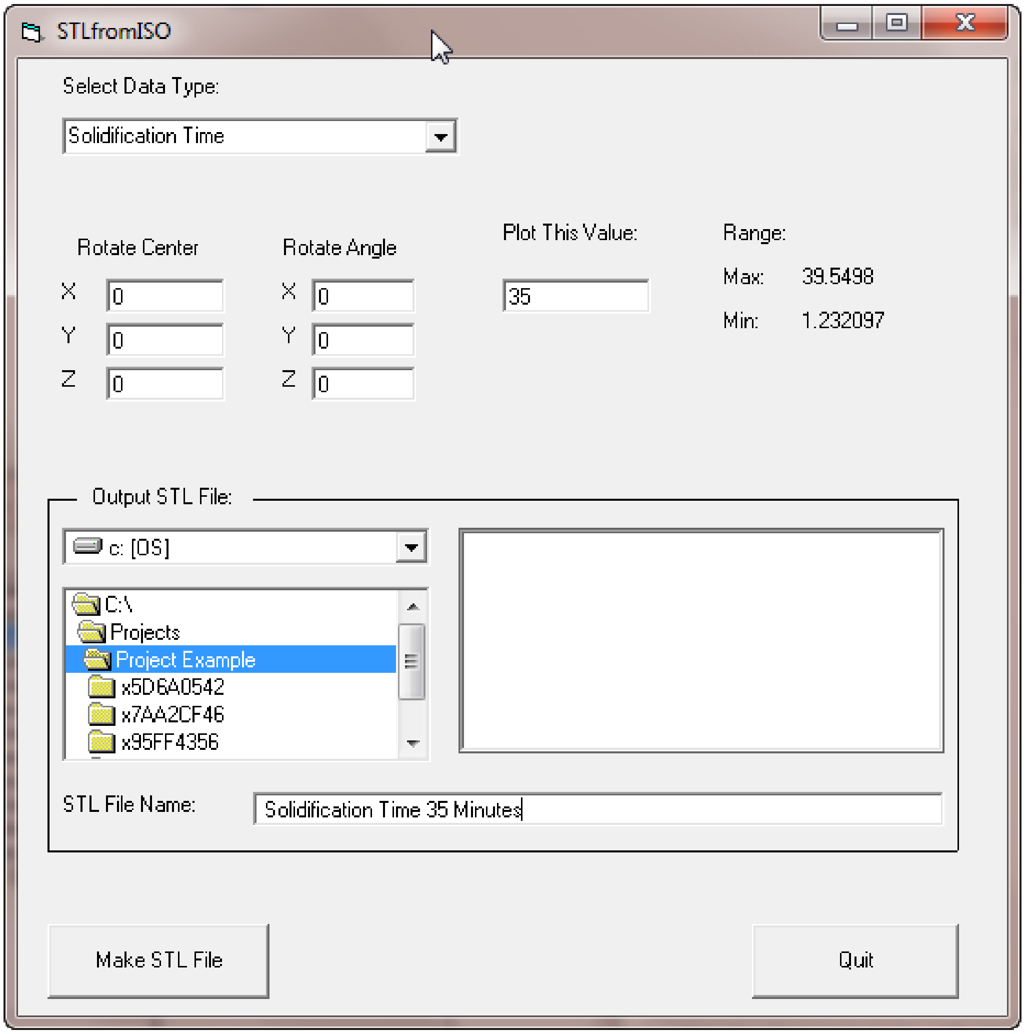
Note that this function allows the ISO-Surfaces to be rotated in space before being written out as STL files. This is because other CAD systems may be using a different coordinate system orientation than SOLIDCast. You may need to experiment a bit with the angles of rotation to find which entries will match the orientation of your CAD system.

To start the utility, load a model, then go to Tools…Create STL File from ISO-Surface. You should see the following:

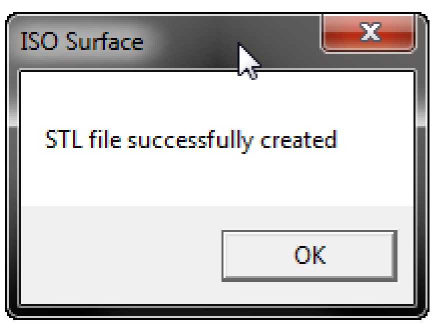


Use the down arrow to scroll through the project tree to find the simulation you want, click the name to highlight it, then click Select Simulation. You will see a form that allows you to choose what data you want, what value you want it plotted at, whether or not you want to rotate the data, and the name and location of the STL file you want to save it in.

A typical filled-in form would look something like this:



When you have the form filled out the way you want, click Make STL File. If all goes well, you should get a response like this:

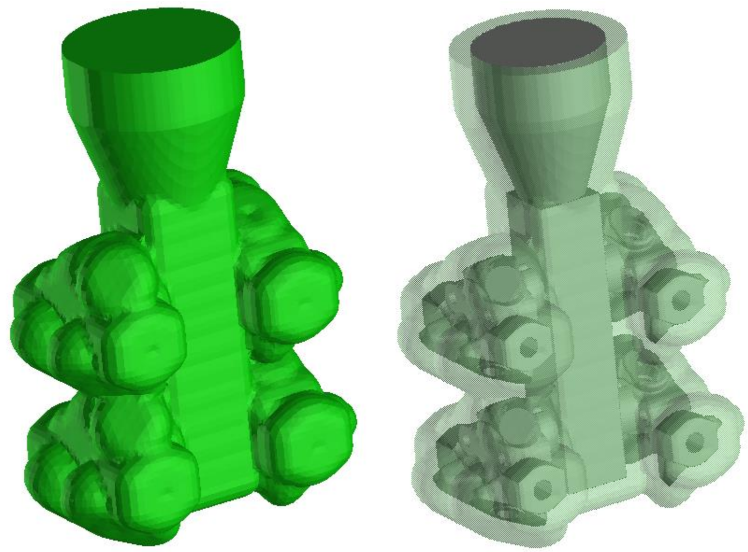


Click OK to return to the Model Building screen.

UTILITY: Create Shell Around STL Shape

Create Shell Around STL Shape is a utility used to create a constant-thickness shell around a shape. The shape *around which the shell is to be created* is contained in an STL file. The output of Create Shell Around STL Shape is a second STL file which contains the geometry of the shell. This STL file can then be loaded as a shape into a SOLIDCast model, as with any other STL file. In this way, the shell then becomes part of the model and is NOT created using the “Shell” option for mold creation when meshing.

A typical shape created by Create Shell Around STL Shape might appear as follows:



External Surface of Shell **Shell with Casting Tree Visible**

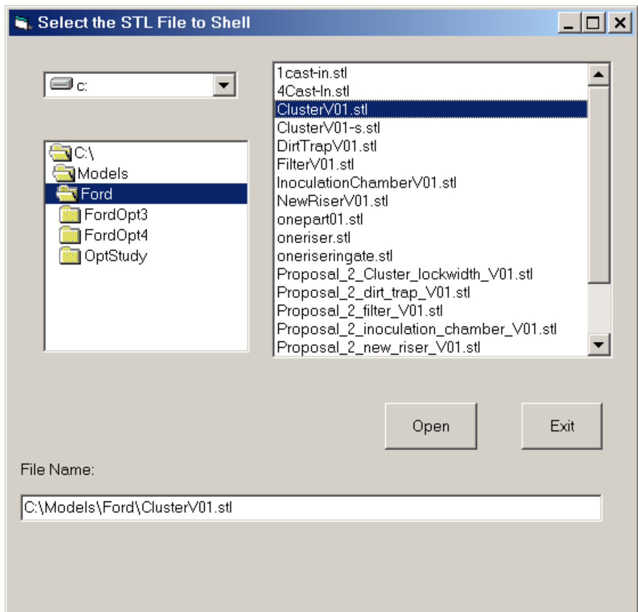
Why would we want to have the shell represented as a shape within the model? The primary reason is that this allows us to add shapes external to the shell. For example, in investment casting, we often have insulating material such as Kaowool or Fiberfrax which is placed outside the shell. Previously, this was somewhat difficult to do using the “Shell Mold” option in meshing; this required placing the insulation at an offset from the casting surface, and then the shell mesh operation filled in the annular space. However, the insulating material was also shelled over in the process. Using Create Shell Around STL Shape, we can add the shell to the model and then add any external shapes much more easily. In some casting processes, a ceramic shell is placed inside another material such as a bed of vermiculite, sand or plaster; this arrangement can be simulated easily by using Create Shell Around STL Shape.

Another advantage of Create Shell Around STL Shape is that it allows us to create multiple shells of different materials, if that is how we are creating the mold for our process. Create Shell Around STL Shape may also be used in Permanent Mold processes to create a mold which is truly a constant thickness around a casting shape.

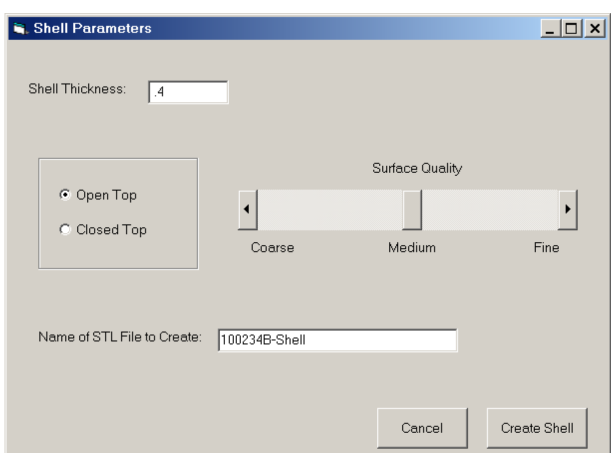
Using Create Shell Around STL Shape... typically would require the following steps:

1. An STL file of the casting around which the shell is to be created must exist. If you have created the entire geometry of the casting, gating, feeders and risers as a single STL file, then you can just use this as the starting file. However, in many cases the casting and rigging are an assembly of shapes in a SOLIDCast model. In this case, you can use the STL From Model utility to create a single STL file of the rigged casting geometry, and then use this file as the input for the Create Shell Around STL Shape utility.

2. Load a model, then select Tools…Create Shell Around STL Shape. The initial screen appears as follows, and allows you to select an existing STL file around which to create the shell:



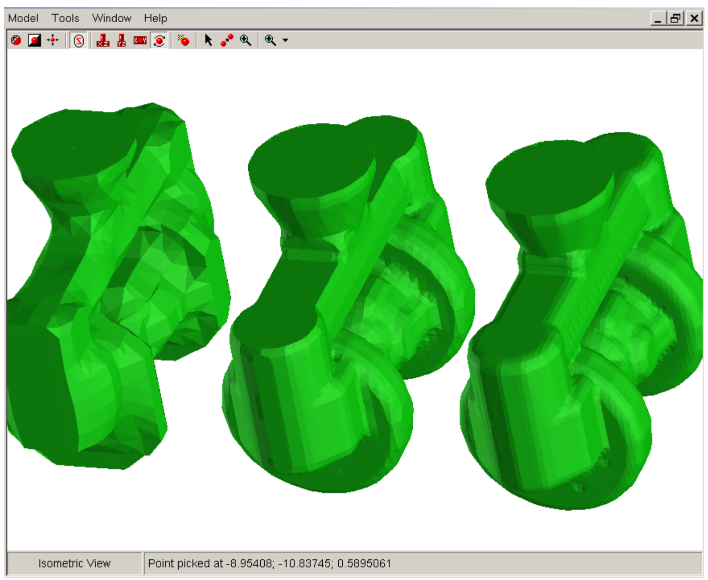
3. Click the **Open** button after you have selected the STL file that you want to use. You will see the following window appear:



4. Enter the shell thickness, whether the shell is open-top or closed-top (this has the same meaning as in the Mesh operation in SOLIDCast), the “Surface Quality” of the shell, and the name to use for the output STL file to contain the shell.

   Note A:

   Surface Quality refers to the detail and smoothness of the shell surface to be created. The higher the quality, the longer the shell creation operation will take, and the larger the resulting STL file will be. For example, in the illustration below, the same shell was created using Coarse, Medium and Fine settings:



Coarse **Medium** **Fine**

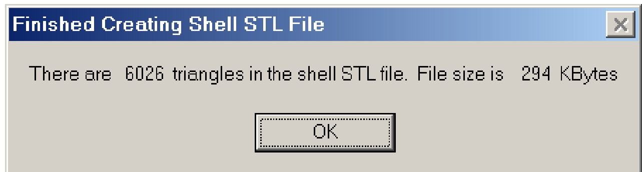
   The coarse setting produced an STL file that was 228 Kbytes in size. The Medium setting produced an STL file that was 1549 Kbytes in size. The Fine setting produced an STL file that was 4178 Kbytes in size, which is relatively large. For most applications, the Medium setting should be sufficient.

   Note B:

   The **shell thickness** should be entered in units which correspond to the units used in the STL file. If the STL file is in millimeters, then shell thickness should be entered as millimeters (for example, you might enter 10 for an investment shell thickness as expressed in millimeters). If the STL file is in inches, then you should enter the shell thickness as inches (an example investment shell thickness in inches might be 0.41). Also, note that if you use the STL From Model utility to create an STL file from a SOLIDCast model, this STL file will be created in inches.

5. Press the **Create Shell** button. The shell creation process takes two passes and may take a few minutes to complete.

6. When finished, Create Shell Around STL Shape will display an informational window that shows the size of the STL file just created. This may appear as follows:



7. Click the OK button to close Create Shell Around STL Shape and return to the Model Builder.

8. Now you can load the shell into your model as an STL file. Be sure to make the Priority Number of the shell *larger* than the Priority number of the casting or any other shape inside the shell. For example, if the Priority Number of the casting is 5, then the Priority Number of the shell should be larger (6, 7, 8, 9 or 10). That way, if there is any overlap between casting and shell, the casting will take precedence.

9. When meshing the completed model, you would normally select “None” under Mold Type in the meshing window (unless you wanted to add either a rectangular mold or another shell external to this shell).

10. Note: If simulating an investment process, you would use the ViewFactor Calculation just as you would with a shell created during the meshing process.

UTILITY: Custom Formula

This utility allows you to create a customized formula for plotting a calculated output from the results of a simulation. The formula is evaluated, and the data is placed in the Custom (either Custom-High or Custom-Low) criterion plotting function in SOLIDCast. You can then plot the data with any of the SOLIDCast plotting tools.

The formula which can be created with this function is of the following form:

$$Result = K * T^{e1} * ST^{e2} * CFS^{e3} * G^{e4} * r^{e5} * LT^{e6} * LST^{e7} * V^{e8}$$

Where

K	=	Constant
T	=	Temperature
ST	=	Solidification Time (min.)
CFS	=	Critical Fraction Time (min.)
G	=	Temperature Gradient (C/cm)
R	=	Cooling Rate (C/min)
LT	=	Liquidus Time (min.)
LST	=	Local Solidification Time (ST – LT) (min.)
V	=	Solidus Velocity (cm/min)
e1 - e8	=	exponents applied to each of the above factors

By entering the constant and exponents for this equation, this utility will evaluate the formula to give the result data for plotting throughout the casting.

Entering 0 for an exponent essentially removes that term from the equation, since any variable raised to the 0^{th} power equal 1. If you do not enter an exponent for a variable, then it will default to 0.

For example, a formula which has been developed for DAS (Dendrite Arm Spacing) in steel and reported in the technical literature is as follows:

$$DAS = 100 * LST^{0.41}$$

This can be evaluated for a simulation by entering the constant and exponents as follows:

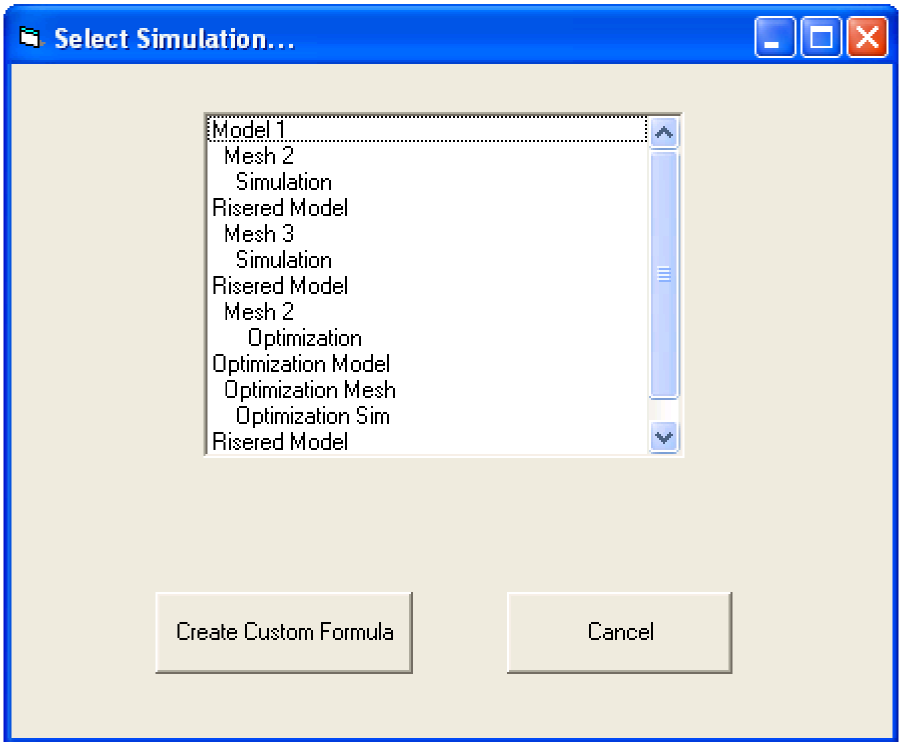
$$Result = 100 * T^{0} * ST^{0} * CFS^{0} * G^{0} * r^{0} * LT^{0} * LST^{0.41} * V^{0}$$

After evaluating the equation, you could then plot the results in SOLIDCast using Custom-High to see DAS in the casting.

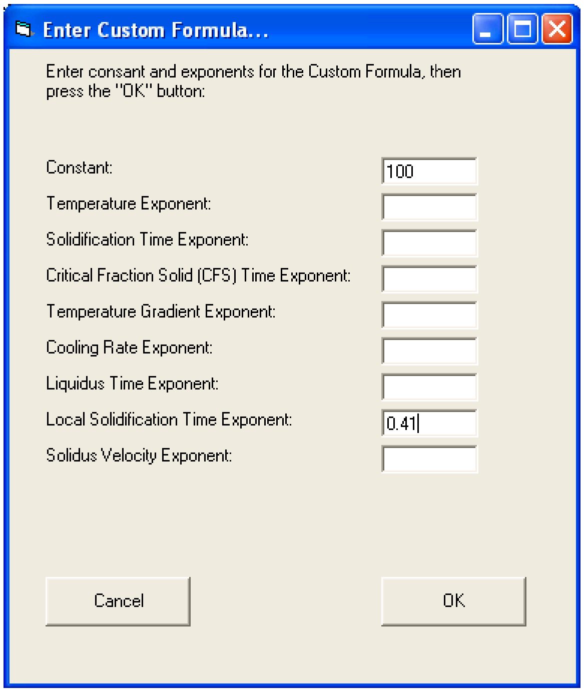
A similar formula for DAS in aluminum that has been proposed is:

$$DAS = 35 * LST^{0.333}$$

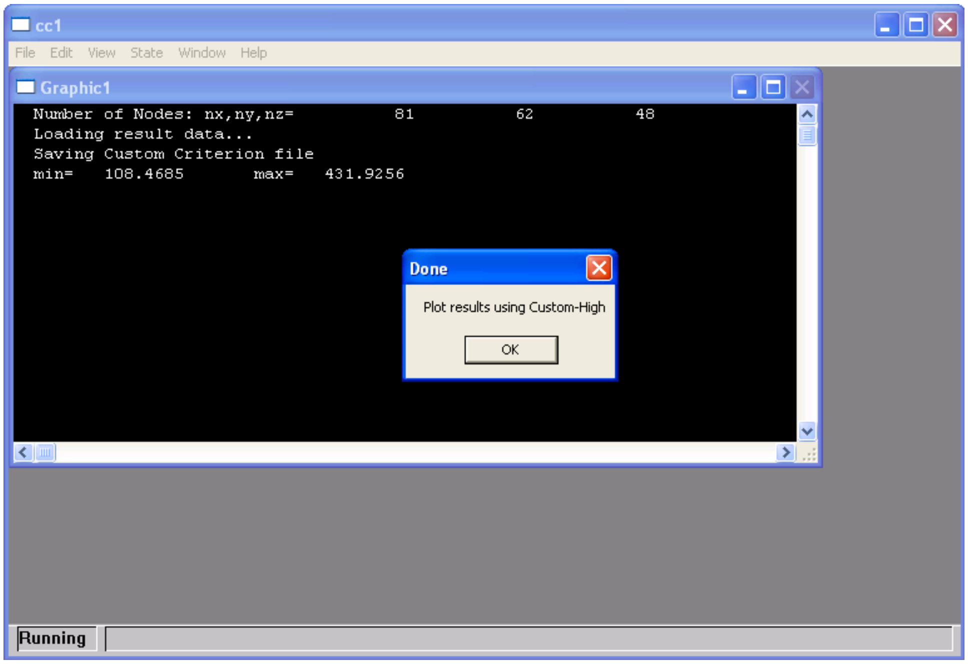
To use this utility, load a model then select Tools…Custom Formula. You will see a reproduction of the project tree, similar to the following screen:



This utility requires that you select a simulation within that project. Highlight the desired simulation on the Project Tree and click the Create Custom Formula button. You will see screen similar to that shown on the next page.



This screen allows you to enter the constant and the exponents for each of the output data items to create the formula. In this example, the proposed formula for DAS for steel has been entered above. Now just click the OK button to proceed. The formula will be evaluated, and the data placed in the Custom function. You should see a summary screen similar to that shown here:



Then in SOLIDCast you can plot Custom-High to view this data using any of the SOLIDCast plotting techniques.

UTILITY: Data Capture (Temperature)

The Data Capture (Temperature) utility allows you to create a text file of (x,y,z) point locations within a SOLIDCast mesh, and then record temperature data at each of these locations while the SOLIDCast simulation is running. The temperature data is output in a text file in the same folder as the original text file. There is no limit on the number of point locations for data capture.

Following are the steps required to use this capability:

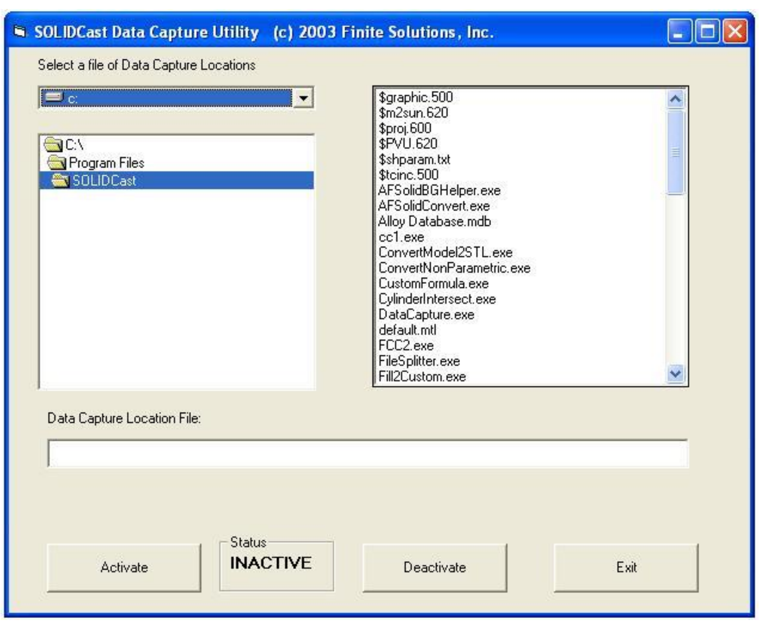
1. Create a text file which contains the (x,y,z) coordinates of the data capture points. The easiest way to do this manually is to use the Windows NotePad accessory program or some other text editor. This text file should appear in the following example format:

   101.6 101.6 101.6
   190.5 101.6 190.5
   215.9 215.9 215.9

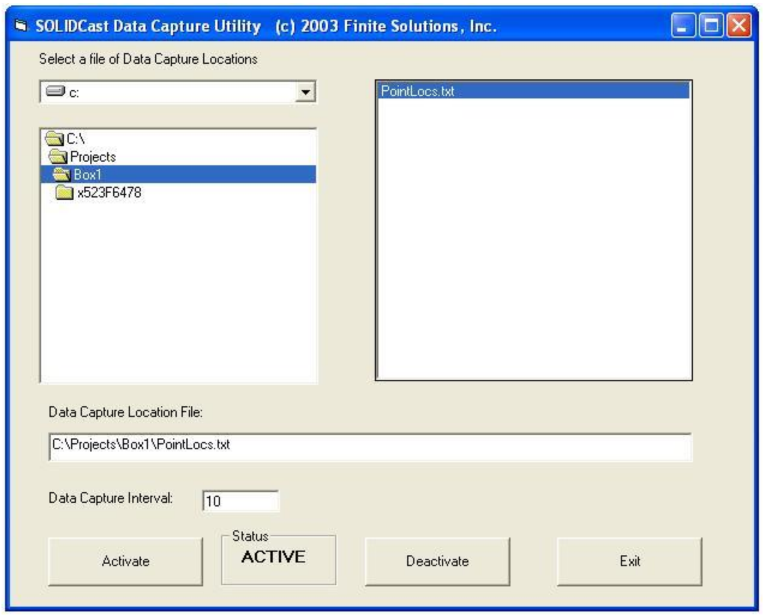
   On each line should appear the x, y and z coordinate of a point at which data is to be collected, separated by spaces or by commas. If your system is set to Metric Units, these coordinates should be in mm. If your system is set to English units, these should be in inches. Use the period for the decimal point, not a comma.

   Save this file using whatever name you want, in whatever folder you want.

2. Next, run the SOLIDCast program. Open the model. Select Tools…Data Capture (Temperature). You will see the following screen appear:



Navigate to the folder where the file of data capture points exists (this is the file created in Step 1). Click on the file name to highlight it and click the Activate button. You should see a window similar to that shown here:



Note that the Status Box has changed from INACTIVE to ACTIVE. This tells SOLIDCast to use the selected file to determine the (x,y,z) points for data collection. Note also that there is now an entry labeled “Data Capture Interval”. This tells the system how many time steps to skip before recording the temperature data. This is important, as this can be used to control the size of the output data file. Since a simulation may take many time steps to complete, if you record data at each time step the size of the output data file may be very large. By increasing this number you can reduce the amount of output data. For example, if a simulation takes 8000 time steps to complete and you specify a Data Capture Interval of 1000, you will have only 8 sets of temperature data at the end of the simulation.

Now that this function is active, press the Exit button. After this, you can run the simulation. The system will create a file called **DataCapture.txt** when the simulation runs and will place it in the simulation folder. This is a text file which contains the output data, as shown in the following example:

```
101.600 101.600 101.600 1.623 1454.444
190.500 101.600 190.500 1.623 1404.990
215.900 215.900 215.900 1.623 53.886
101.600 101.600 101.600 2.165 1454.055
190.500 101.600 190.500 2.165 1400.335
215.900 215.900 215.900 2.165 58.923
101.600 101.600 101.600 2.706 1452.909
190.500 101.600 190.500 2.706 1396.710
215.900 215.900 215.900 2.706 64.045
101.600 101.600 101.600 3.247 1450.797
```

The first three columns are the (x,y,z) data capture locations. Column 4 is the time in minutes. Column 5 is the temperature at this point. Since this is a standard text file, you will be able to import it into programs such as Excel for sorting or plotting.

When you are finished collecting data, you **MUST** run the utility again and select the “Deactivate” button. If this is not done, then SOLIDCast will continue to try and record temperature data for all subsequent simulations.

NOTES:

1. If you are running a permanent mold multi-cycle simulation, Data Capture (Temperature)… will capture data from only the final cycle.

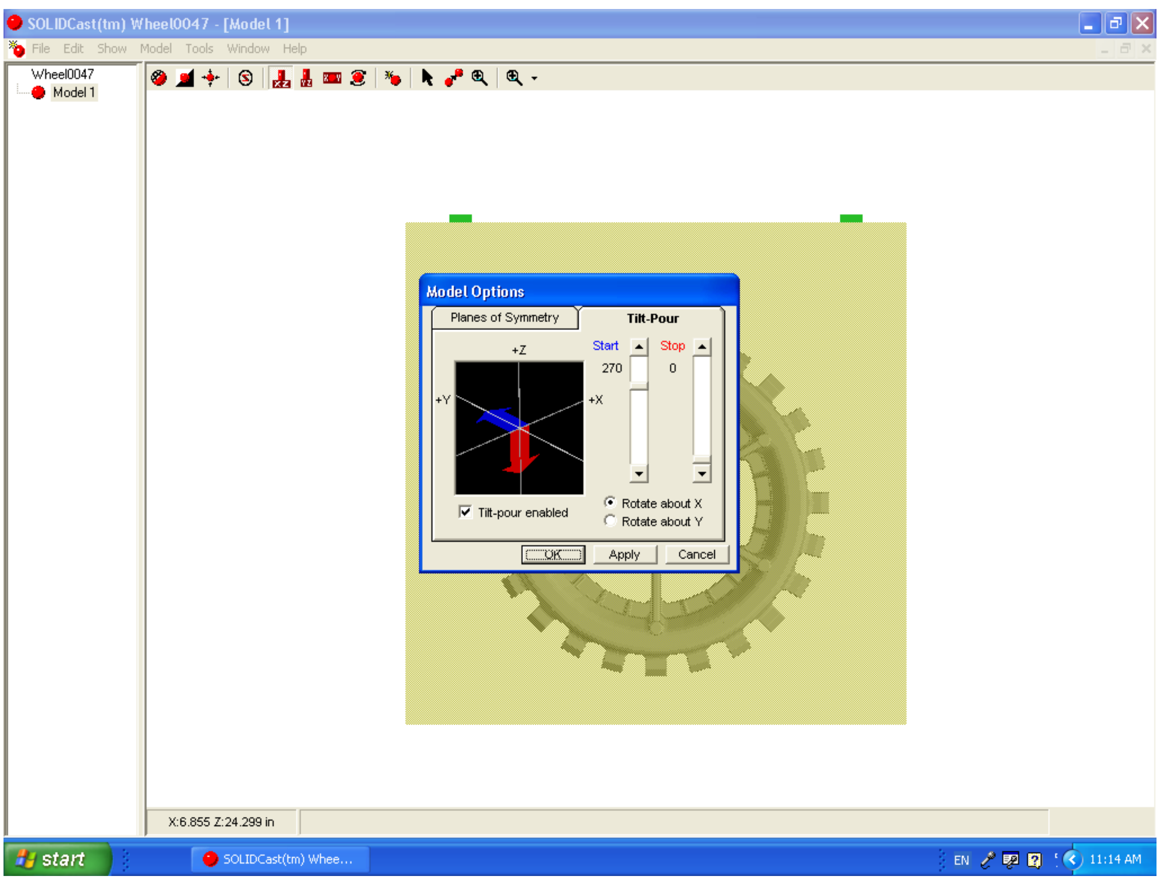
2. Do not interrupt and restart the simulation while Data Capture (Temperature)… is active. If this is done, only the data from the restart point will be recorded.

UTILITY: Early Fill Tilt Pour

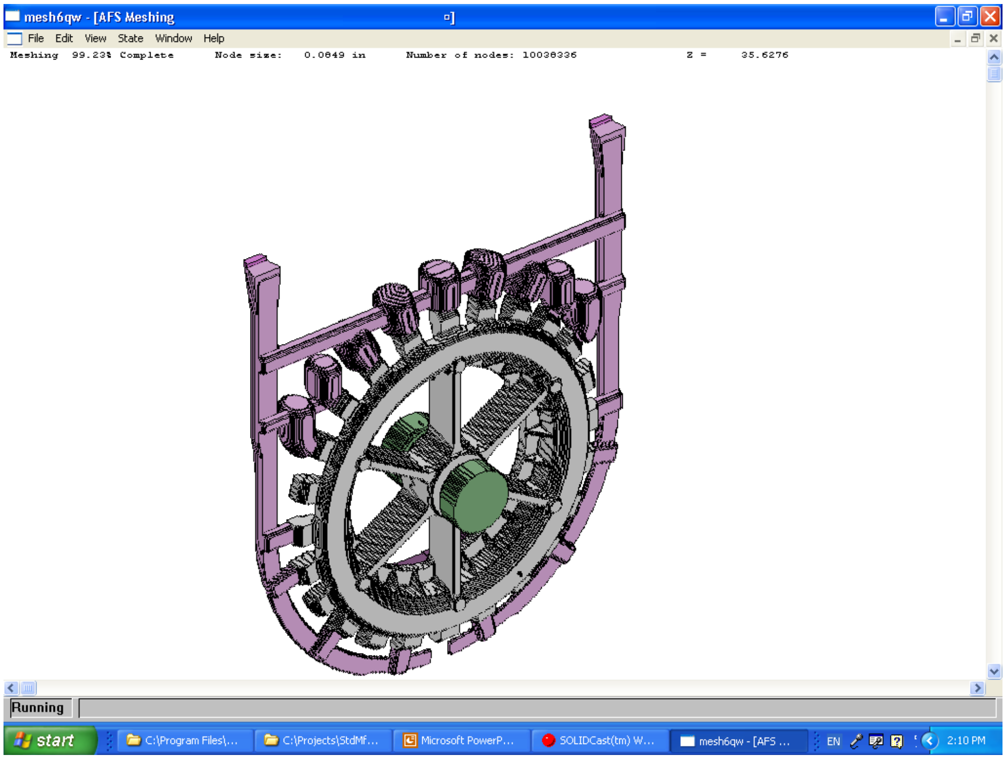
This utility allows most of the metal in a tilt pour FLOWCast simulation to enter the mold at a given percent of tilt. For example, if the mold tilts through 90 degrees during filling, and the user specifies 50% as the completion point, then most of the liquid metal will have entered the mold during the first 45 degrees of tilting.

FLOWCast does not currently have the ability to simulate a "zero flow" condition during tilt pouring, therefore a small amount of metal will still enter the mold during the last part of the tilt. The amount of metal which enters during the last portion of the tilt is dependent on the size of the contact area between "Fill Material" and either casting or riser material. If the contact area decreases, the amount of metal entering during the last portion of the tilt also decreases. This is due to a lower limit on the velocity of the metal in the CFD flow calculations. Early Fill Tilt Pour… will let you know what percent of the metal will enter the mold during the first part of the tilt, and whether this could be increased by reducing the size of the Fill Material contact area.

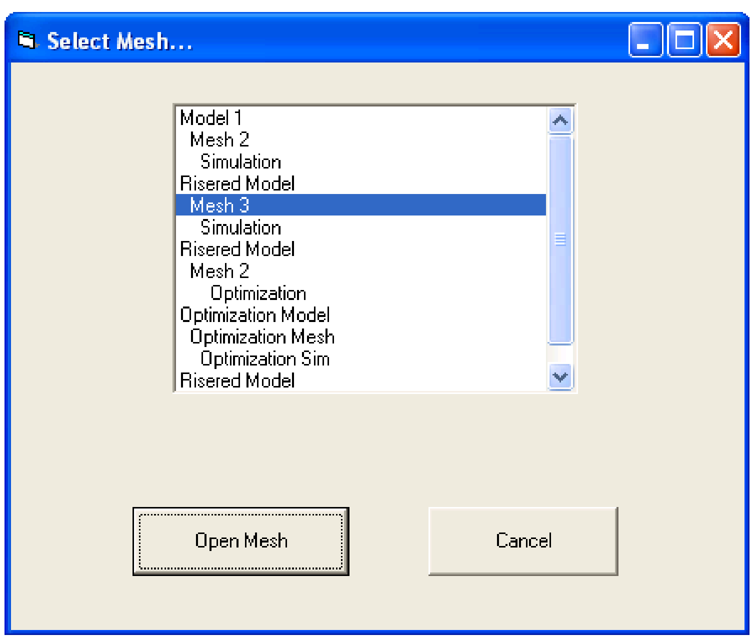
The first step in using this utility is to make sure that Tilt Pour is enabled in the model, by selecting Model…Options…Tilt Pour. In the example shown below we have specified a Start Angle of 270 degrees and a Stop Angle of 0 degrees, which causes the mold to rotate through 90 degrees. Click the Apply button after setting the angles.



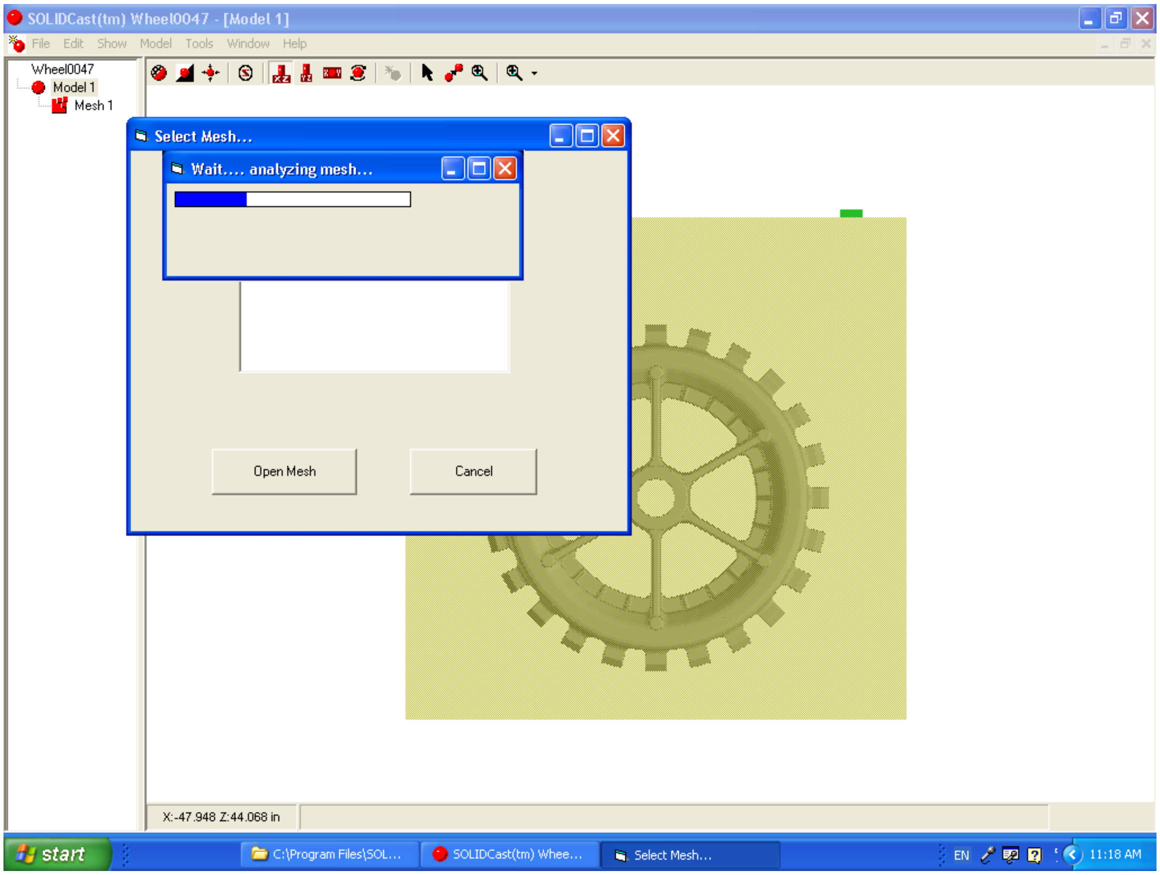
Next, mesh the model by going to Model…Create Mesh. This figure shows an example part in the process of meshing:



Once the model is meshed, you will be returned to the model building screen. Select File…Save Project, to make sure you have all your data saved. Then select Tools…Early Fill Tilt Pour…. The system will display the Project Tree, as shown in the following window:

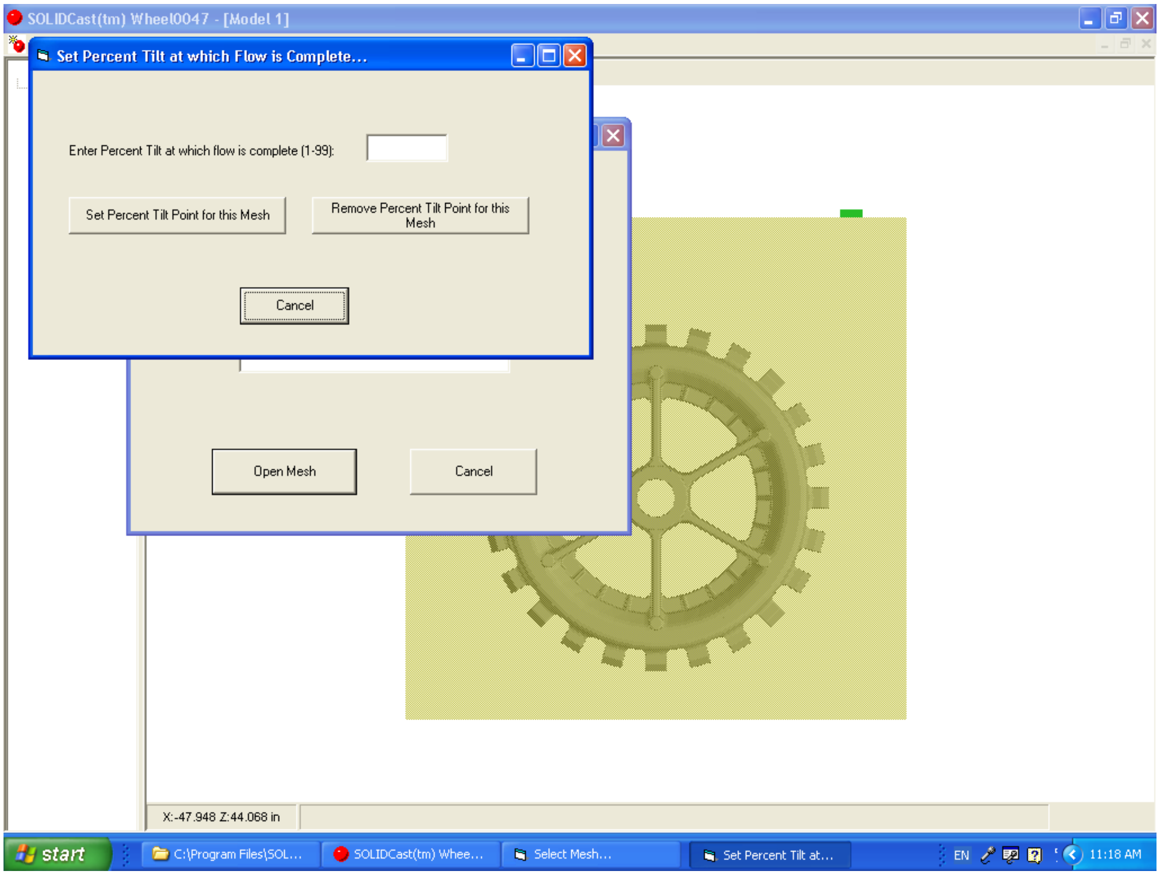


Use the cursor to highlight the mesh you want to adjust, then click on the Open Mesh button. The system will then begin analyzing the mesh, as shown here:



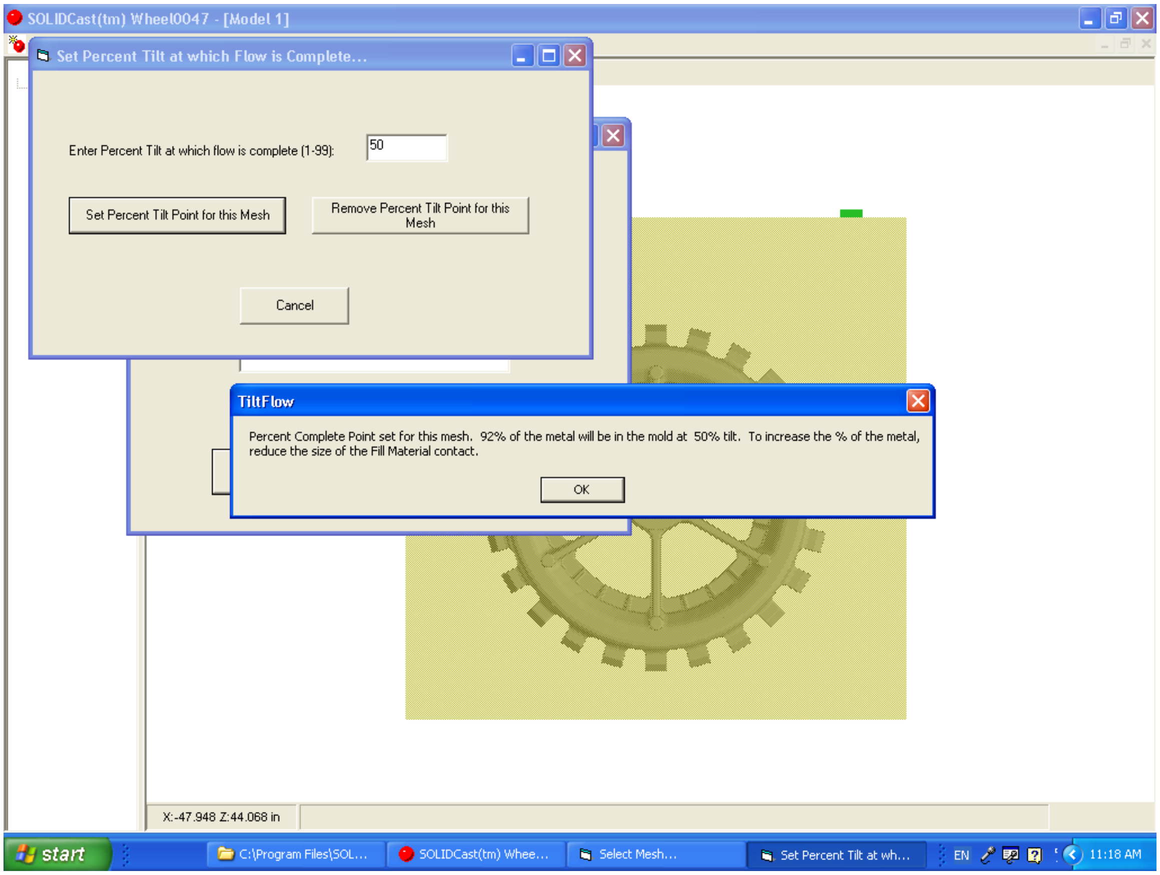
In some cases there may be a problem with the mesh. For example, if the Fill Material in your model does not directly make contact with something that can fill, either Casting Material or Riser Material, then the following message will be displayed. This could also happen if you do not have any Fill Material in the model.

In most cases the mesh will analyze properly, and you will be presented with a window similar to that seen on the next page.



Now, enter the Percent Tilt (NOT the angle) at which you want the mold to be filled. For example, you could enter 50 percent, which would be the equivalent of 45 degrees for a 90-degree tilt. Then click the Set Percent Tilt Pour for this Mesh button.

A message is now displayed which says how much of the metal will be in the mold at the given percent tilt. In this case, the mold will be 92% full at 45 degrees. Note that this percentage could be increased by decreasing the size of the Fill Material contact.



Now click the OK button to exit.

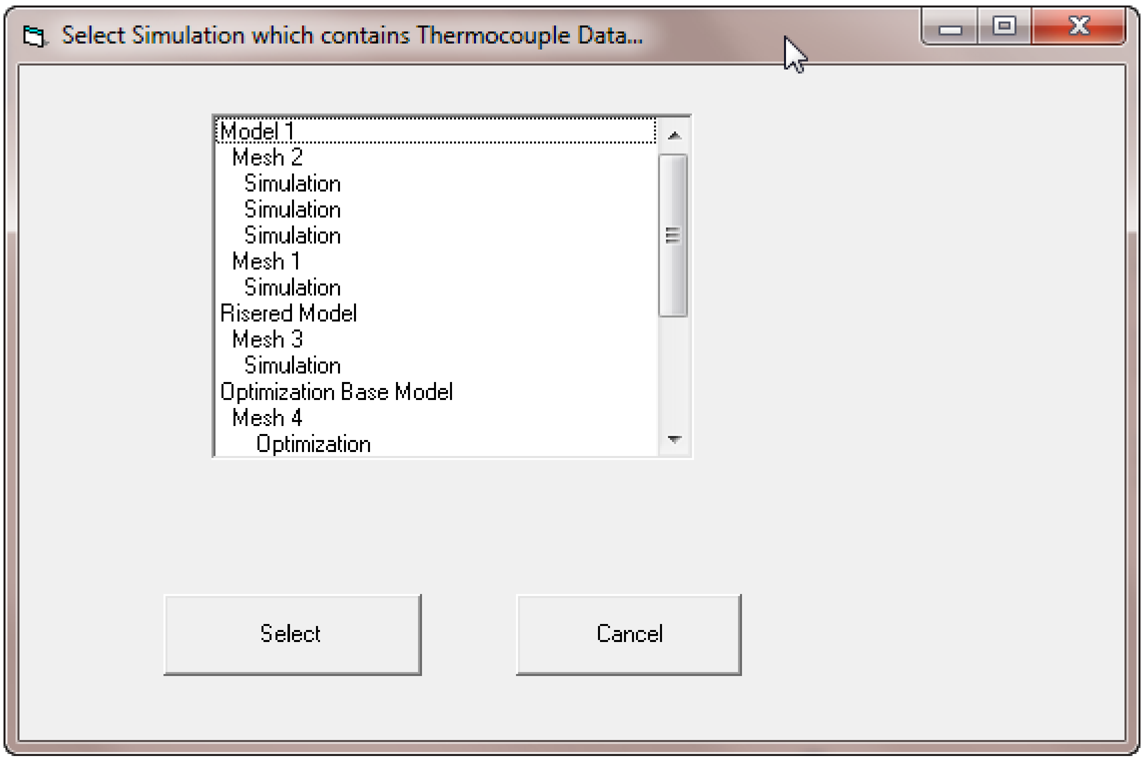
After setting this fill point, go ahead and run a FLOWCast simulation as normally done. The mold will fill as indicated by the Early Fill Tilt Pour Utility.

UTILITY: Extract Thermocouple Data

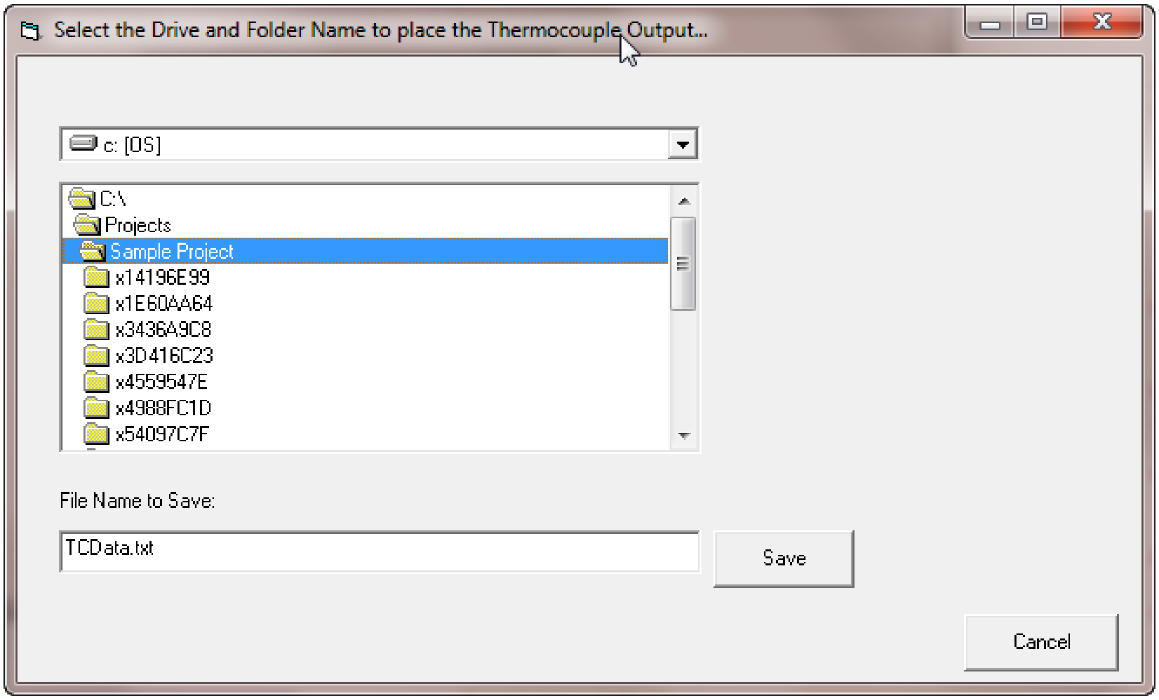
This utility will take time-temperature data created by SOLIDCast, and export it out into a text file, for use in plotting in other software packages, such as spreadsheets.

To use the utility, you must first have run a simulation with thermocouples active. See the Data Capture (Temperature) utility instructions for details.

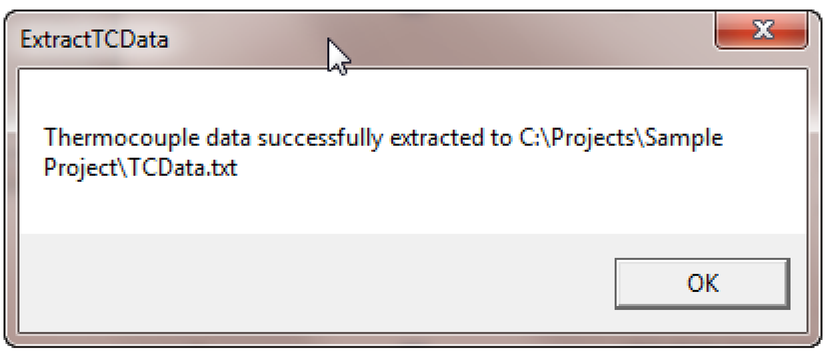
To run the utility, load the project and model, then select Tools…Extract Thermocouple Data. You will see the following:



Highlight the simulation that has saved thermocouple data, then press the Select button. The system will then display the screen shown on the next page.



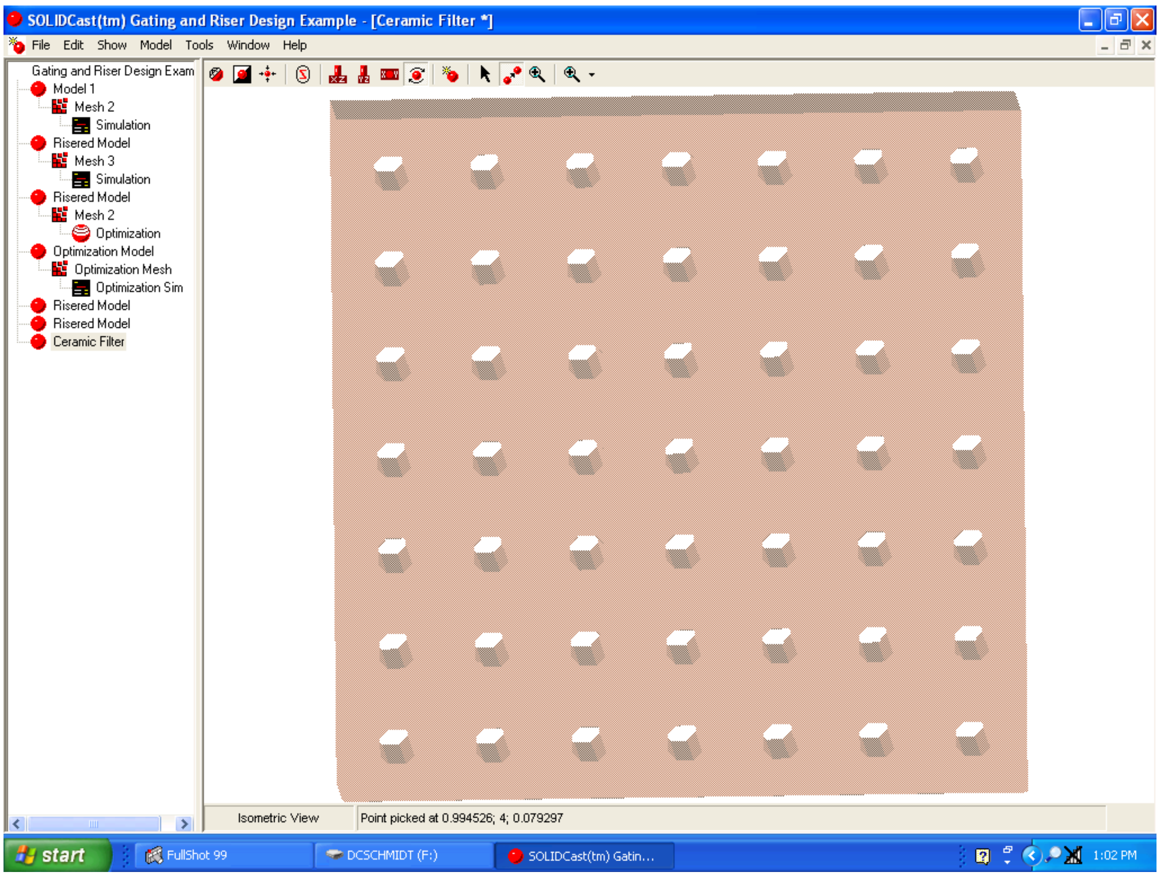
Navigate to where you want the data saved to, then click the Save button. If all goes well you should receive the following message:



You can now load the text file into a spreadsheet or other program for plotting. Click OK to return to the Model Building screen.

UTILITY: Filter Shape

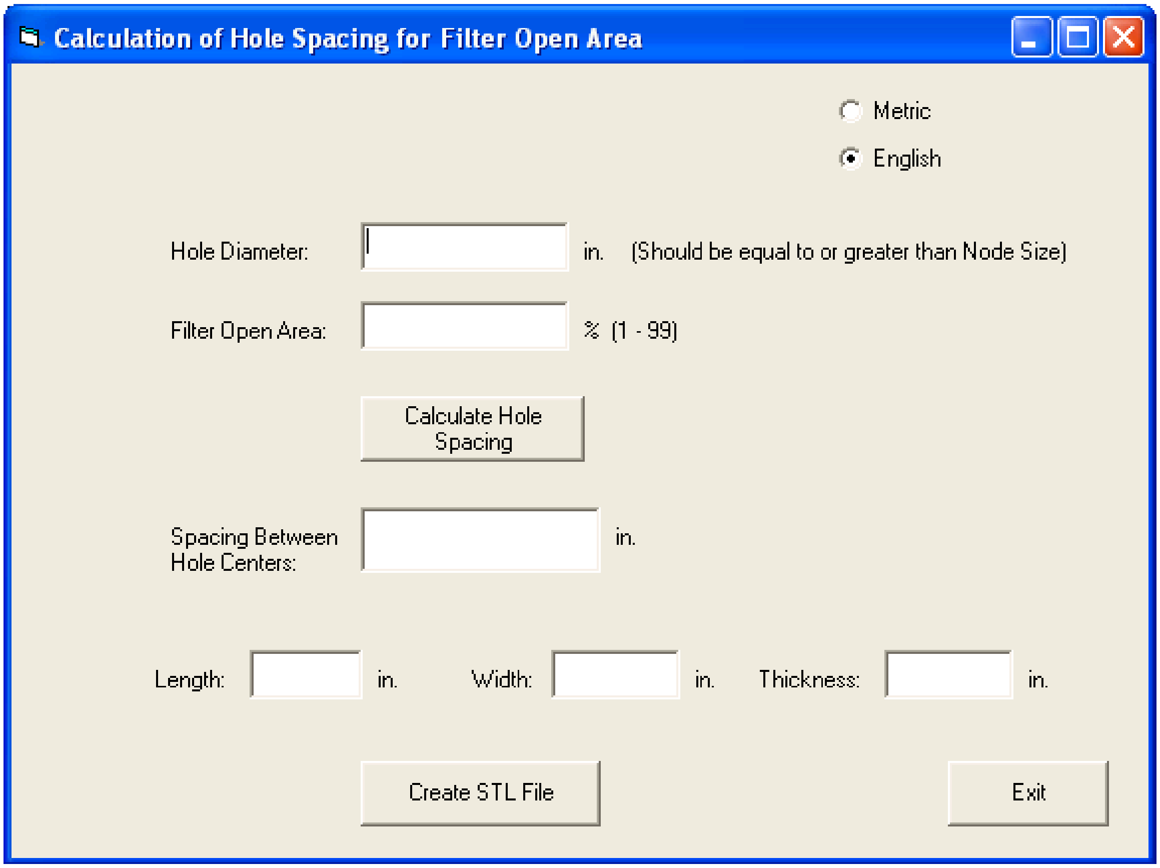
This utility helps you to create an STL file consisting of a rectangular block with cylindrical holes. This object, when added to a model and given ceramic thermal properties, can be used to simulate a filter. The figure below shows what a filter created with Filter Shape would look like in the SOLIDCast model builder.



A typical ceramic filter would have thermal properties similar to these:

Thermal Conductivity:	0.9 BTU/hr-ft-F	1.557 W/m-K
Specific Heat:	0.2 BTU/lb_m	836.8 J/kg-K
Density:	130 lb/ft^3	2082.34 kg/m^3

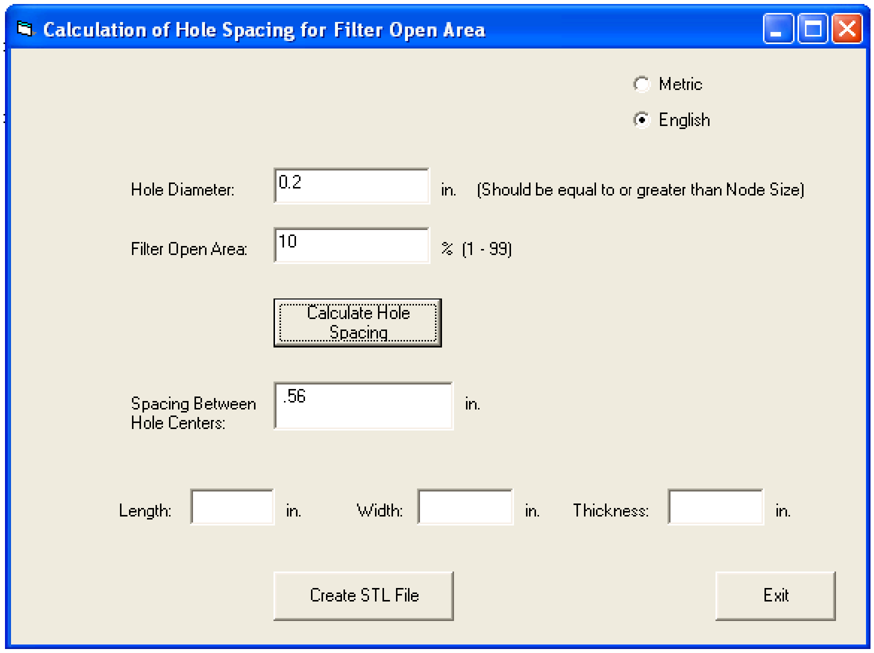
To run this utility, first load your model. Then select Tools…Filter Shape. You should see a window that looks like this:



Note that you can select the units you want, either millimeters or inches, by clicking on the appropriate button on the upper right side of the window.

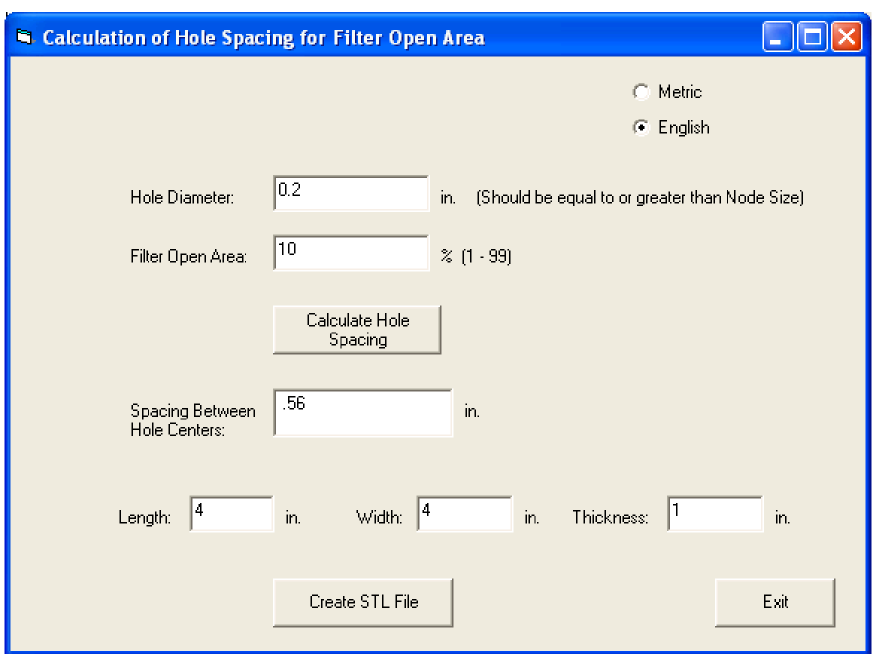
The first part of the utility calculates the center-to-center hole spacing for a grid of holes through the filter material, based on hole size and percent open area in the filter. Note that the diameter of each hole should be larger than the node size you will be using the mesh the model, so that the holes are actually created in the mesh. In the example shown here we will use a hole diameter of 0.2 inches (5mm). The filter open area, in percent, will be 10%. That is, holes occupy 10% of the cross-sectional area of the filter. This number can vary from 1% to 99%.

After entering the hole diameter and filter open area, click on the Calculate Hole Spacing button. You should see something similar to this:

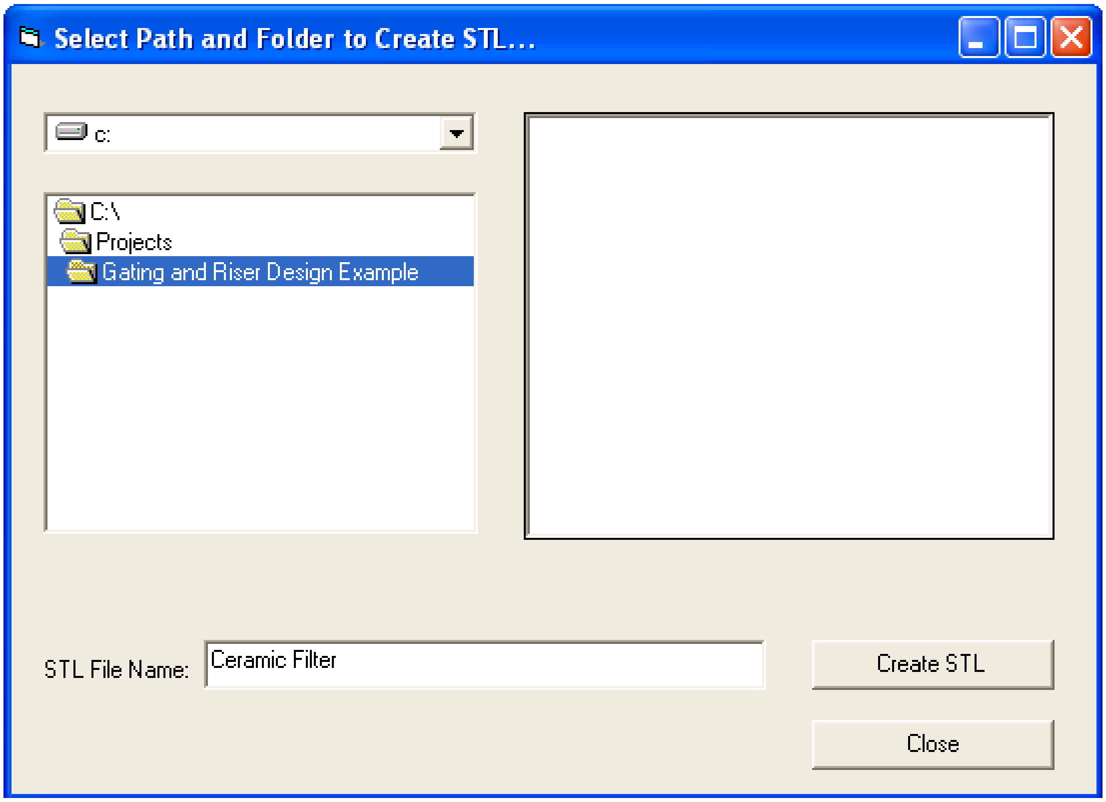


In this example we have a center-to-center hole spacing of 0.56 inches.

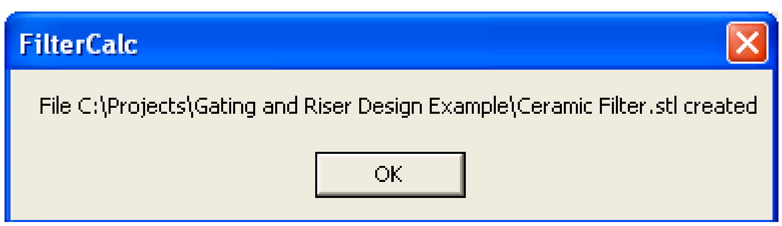
To create a solid object of the filter, in the form of an STL file, we need enter the length, width and thickness of the ceramic filter. In this example, we will enter 4 inches (100mm), 4 inches (100mm) and 1inch (25mm) for our dimensions. The completed form would look like this:



Once the dimensions are entered, click on Create STL File. You will see a window similar to this, where you can name and place the new STL file.



Click on the Create STL and the file will be created and stored. When this completes, the system will display:



Click on OK and you will be returned to the model builder.

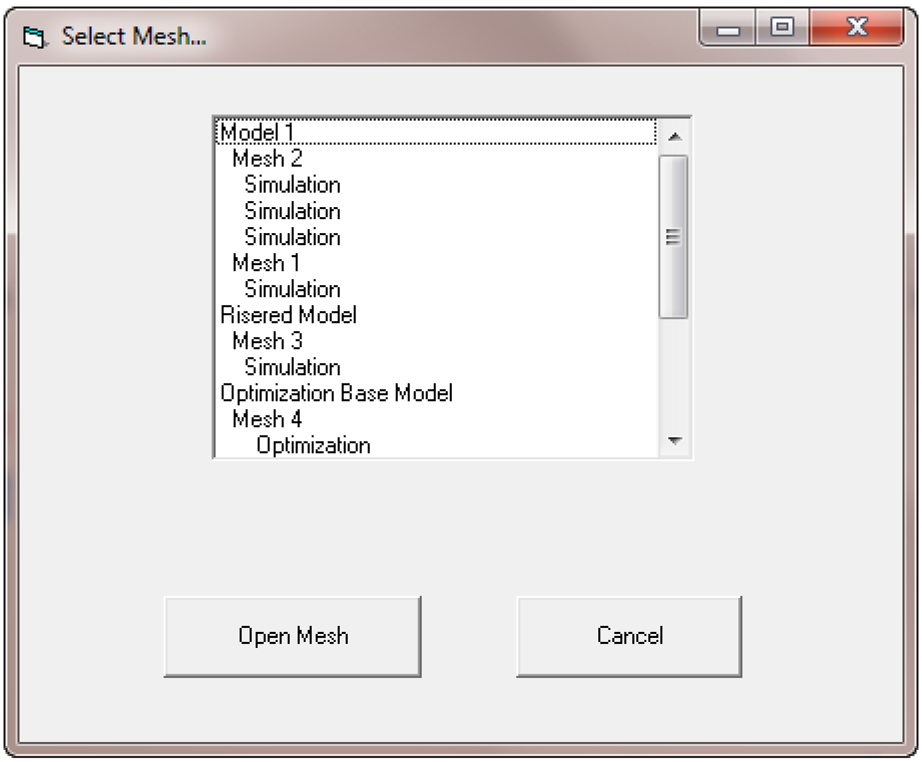
You now have an STL file that can be loaded into your model, and given the properties of a ceramic filter. The filter should overlap either Casting Material or Riser Material. Make sure that the filter has a smaller (closer to 1) priority number than the metal it is imbedded in. This way, when the model is meshed, you will get the filter in the mesh with metal in the holes.

UTILITY: Four Point Fill

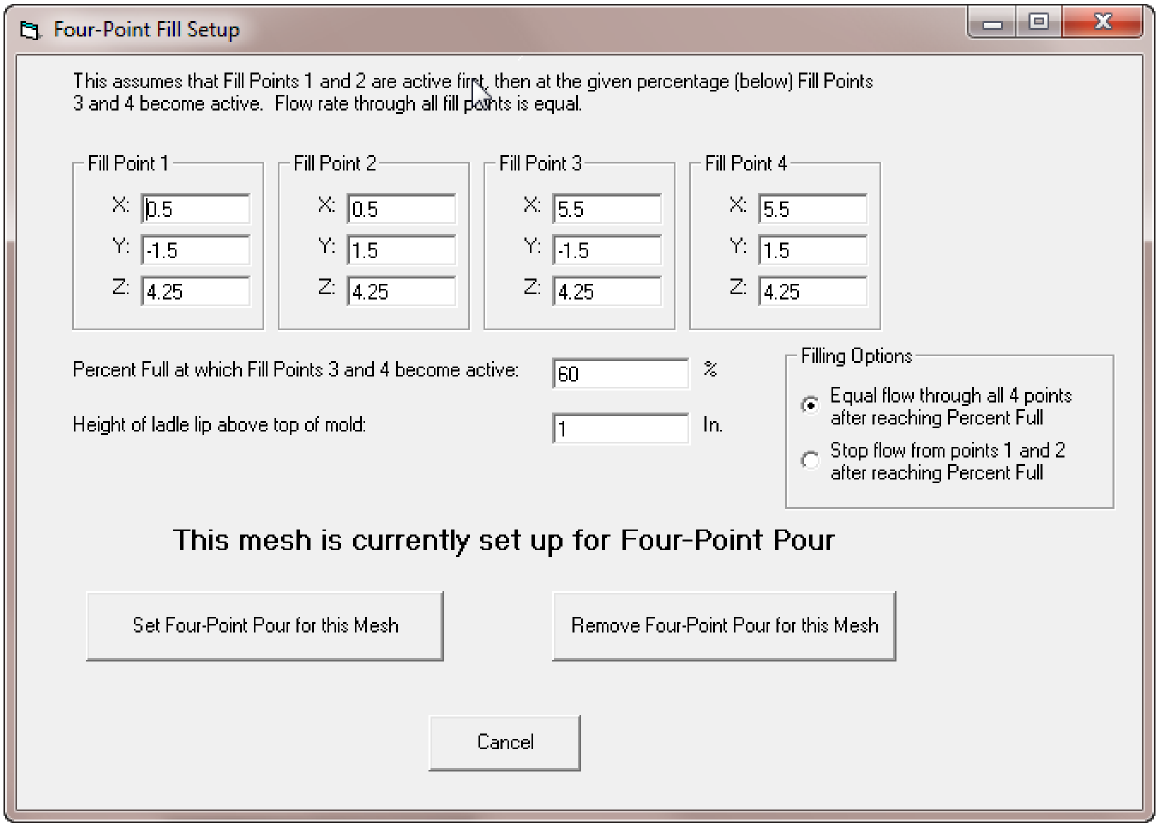
In this utility, pouring starts at two points. Then, at a given percent full, pouring begins at two additional points so that in the second stage of pouring all four pour points are active. In this scenario, the incoming flow rate at each pour point is equal, which means that the total rate of flow in the second stage of pouring is twice that of the first stage, and the mold fills faster during the second stage than during the first. Note that this utility only works for FLOWCast filling and not with the SOLIDCast simplified filling algorithm.

A requirement for this utility is that your model be built with four separate pieces of Fill Material. They must be in contact with areas that can be filled; i.e., made of either Casting or Riser Material.

To use Four Point Fill, first load your model, then click on Tools…Four Point Fill. You should see a screen similar to this:



Scroll down and highlight the mesh you want to work with, then click Open Mesh. You should then be brought to the setup screen, which is shown on the next page.



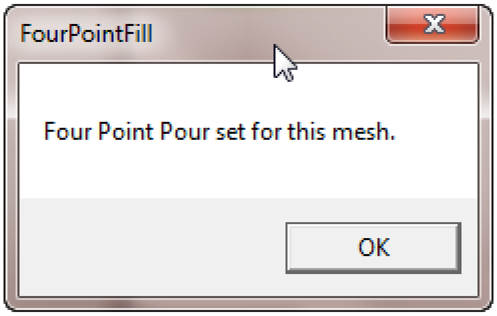
The default scenario is that filling begins at Fill Points 1 and 2 simultaneously, and at the same rate. Then, at the Percent Full point entered on the worksheet, Fill Points 3 and 4 also become active and all 4 points fill at the same rate, effectively doubling the pouring speed.

A second possible scenario is to have Fill Points 1 and 2 start, then, when Fill Points 3 and 4 become active, Fill Points 1 and 2 stop filling. These options are selected by clicking on the appropriate button under Filling Options.

Locations of the Fill Points are entered in X, Y, Z coordinates, using inches or mm, depending on what units you are using in SOLIDCast.

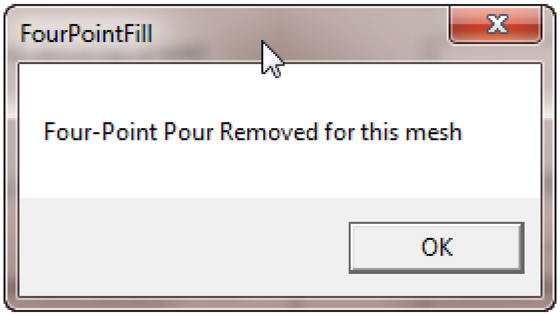
The last entry is the height of the ladle lip above the top of the mold.

Once all the data has been entered, click on Set Four-Point Pour for this Mesh. The data will be saved and the system will respond like this:



Note that the worksheet lets you know the current status of the mesh, regarding Four-Point Pour.

If you do not want Four-Point Pour applied to this mesh, click the button that says Remove Four-Point Pour for this Mesh. The system will respond with:

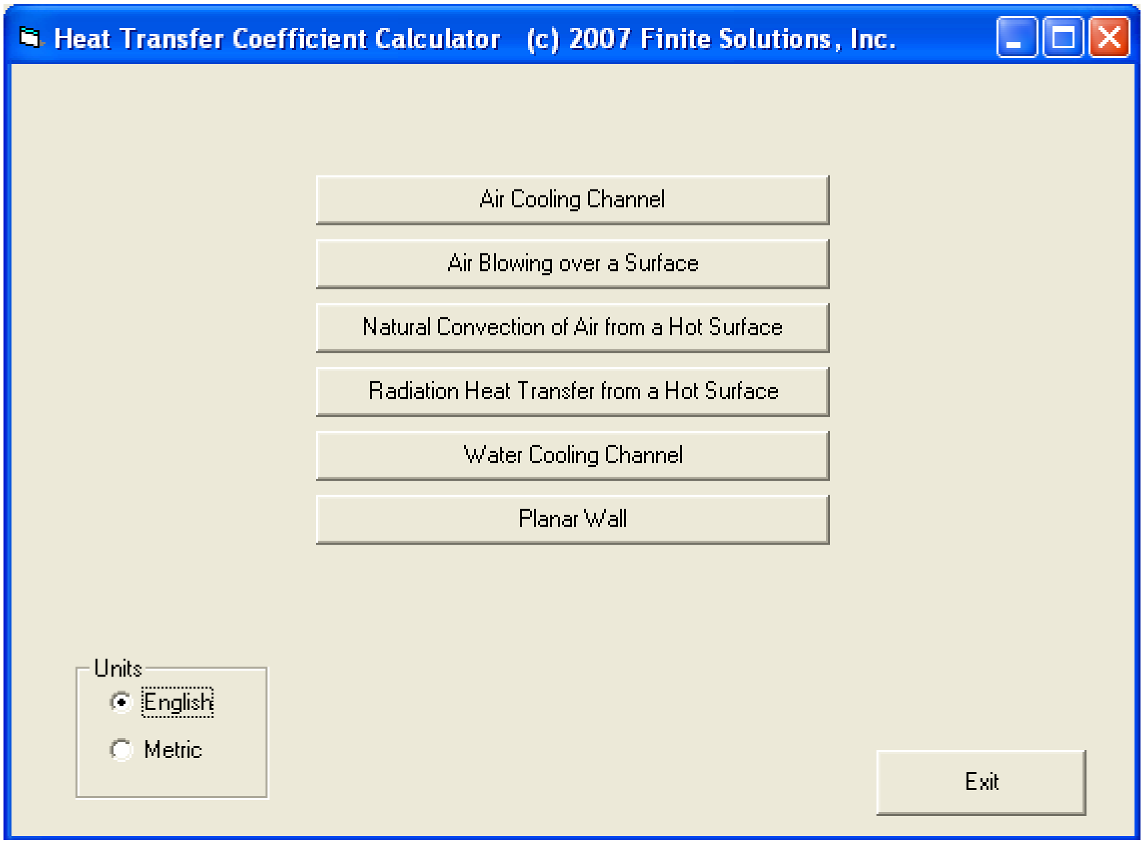


In either case, once you click the OK button you will be returned to the Model Building screen.

UTILITY: HTC Calculator

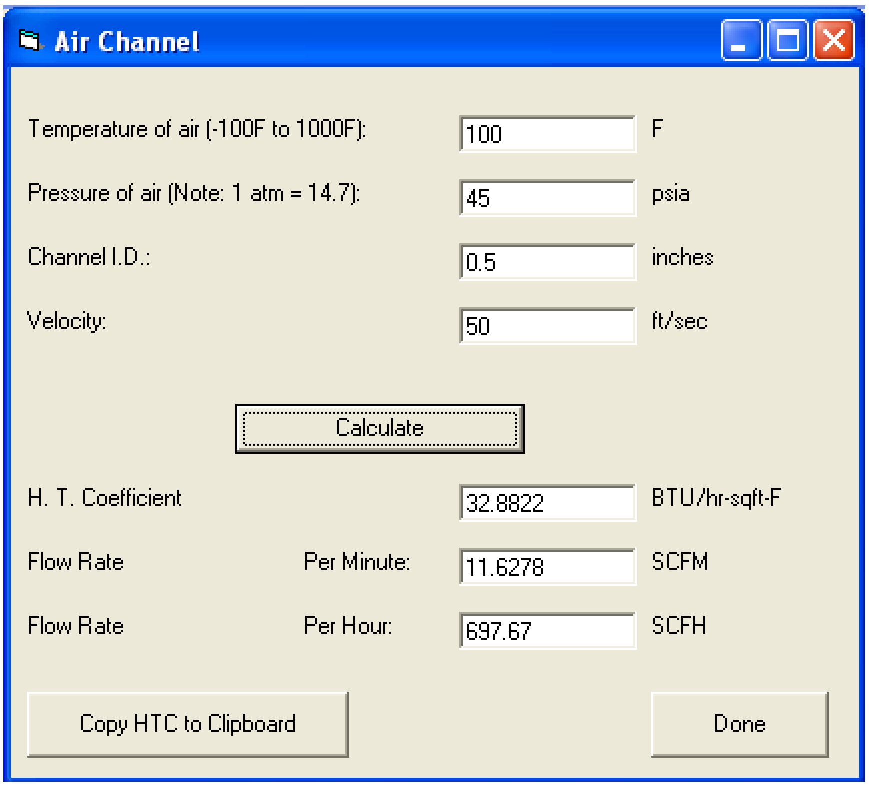
This utility helps you to calculate the appropriate values for Heat Transfer Coefficients(HTCs) for use in a SOLIDCast simulation. The utility considers various situations, including convection and radiation from hot surfaces, water and air cooling, as well as coating coefficients for mold sprays.

To run this utility, first load your model. Then select Tools…HTC Calculator. You should see a window that looks like this:



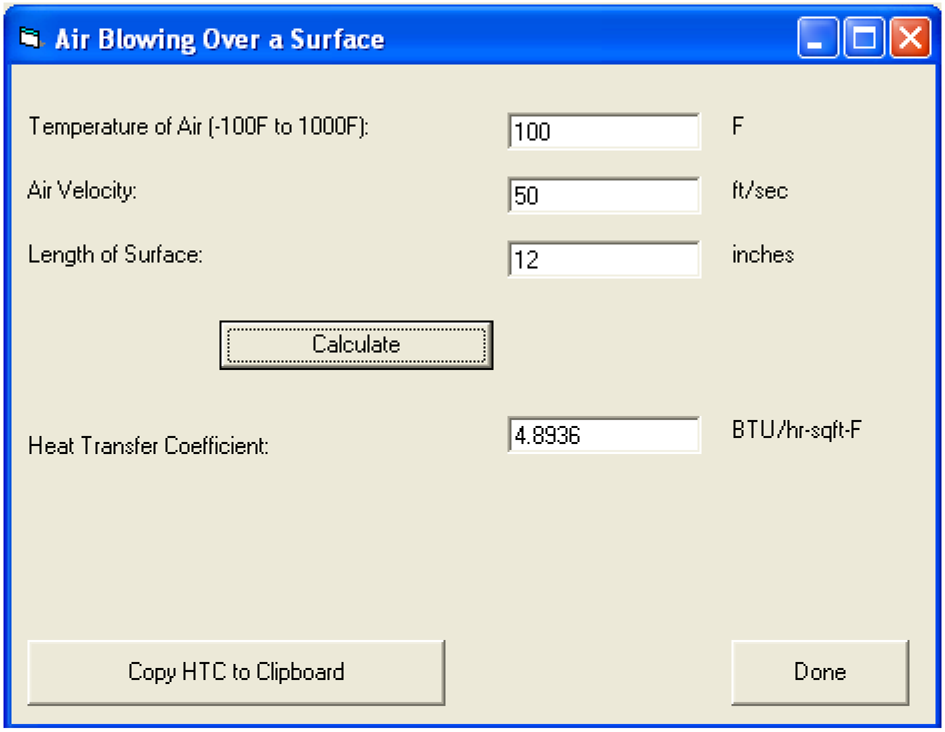
Note that you can shift between English and Metric units, by clicking the appropriate button in the lower left corner of the window. The different options are shown on the following pages.

Air Cooling Channel – This function is used in the permanent mold, or gravity die process, when air is forced through a channel in a metal die to locally cool that portion of the die. Required inputs are the average air temperature in the channel, the air pressure, the diameter of the channel and the air velocity. Outputs include the effective HTC for the interface between the channel and the die, and the air flow rate per minute and per hour, as shown here:



Note that you can use the button in the lower left corner to copy the calculated HTC value to the Windows Clipboard. Once you have done this you can use the Windows Paste command to enter that value into the HTC table in the Materials List. An example of this is shown at the end of this section.

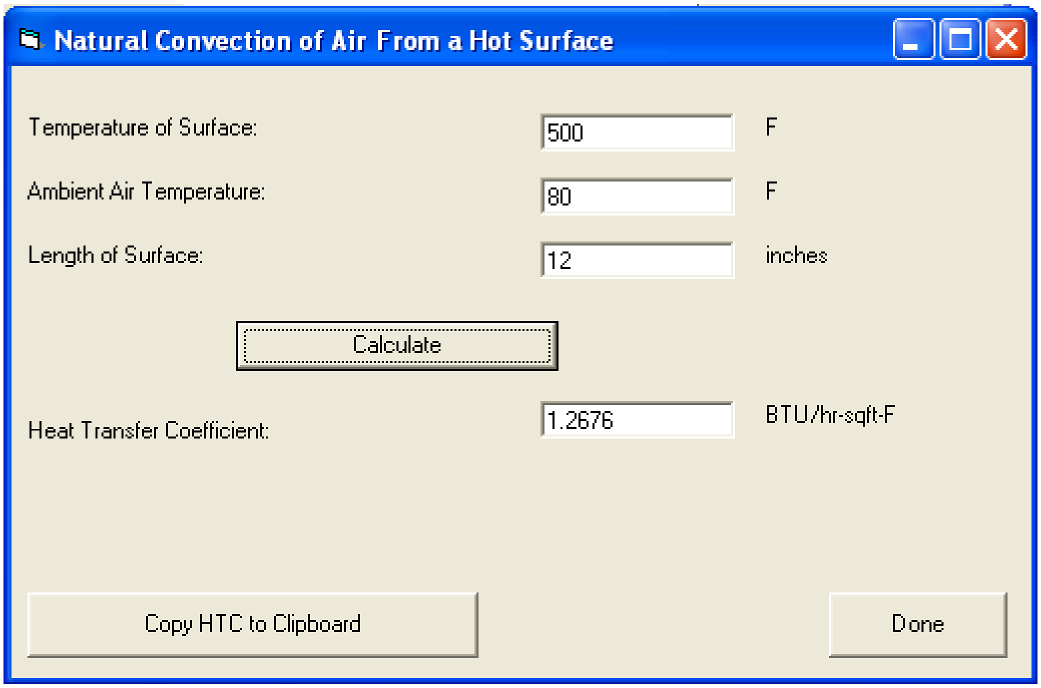
Air Blowing Over a Surface – This function is used to calculate an HTC based on forced convection of air that is blown over a surface using a fan. Required inputs are the air temperature, air velocity and the length over which the air is blowing. The calculated output is the effective HTC, as shown here:



Note that you can use the button in the lower left corner to copy the calculated HTC value to the Windows Clipboard. Once you have done this you can use the Windows Paste command to enter that value into the HTC table in the Materials List. An example of this is shown at the end of this section.

In the case of forced convection, the HTC is normally added to the calculated radiation HTC and the total value is used.

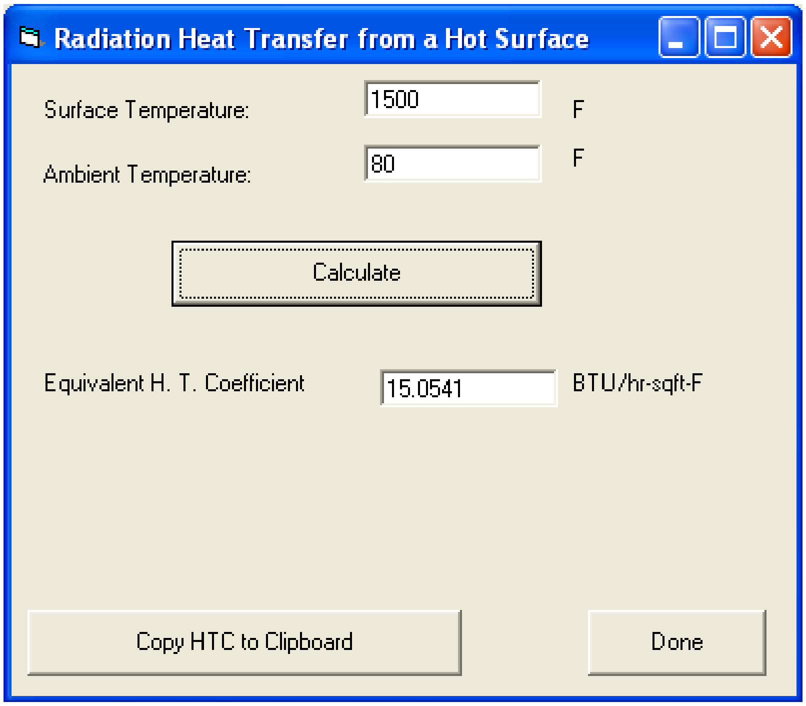
Natural Convection of Air From a Hot Surface – This function is used to calculate the effective HTC from the unforced convection of air that heats up on the hot surface of a die or investment shell. This value is typically lower than that for forced convection. Required inputs are the surface temperature, air temperature and the length of the surface that the air is moving over. The output is the effective HTC, as shown here:



Note that you can use the button in the lower left corner to copy the calculated HTC value to the Windows Clipboard. Once you have done this you can use the Windows Paste command to enter that value into the HTC table in the Materials List. An example of this is shown at the end of this section.

In the case of natural convection, the HTC is normally added to the calculated radiation HTC and the total value is used.

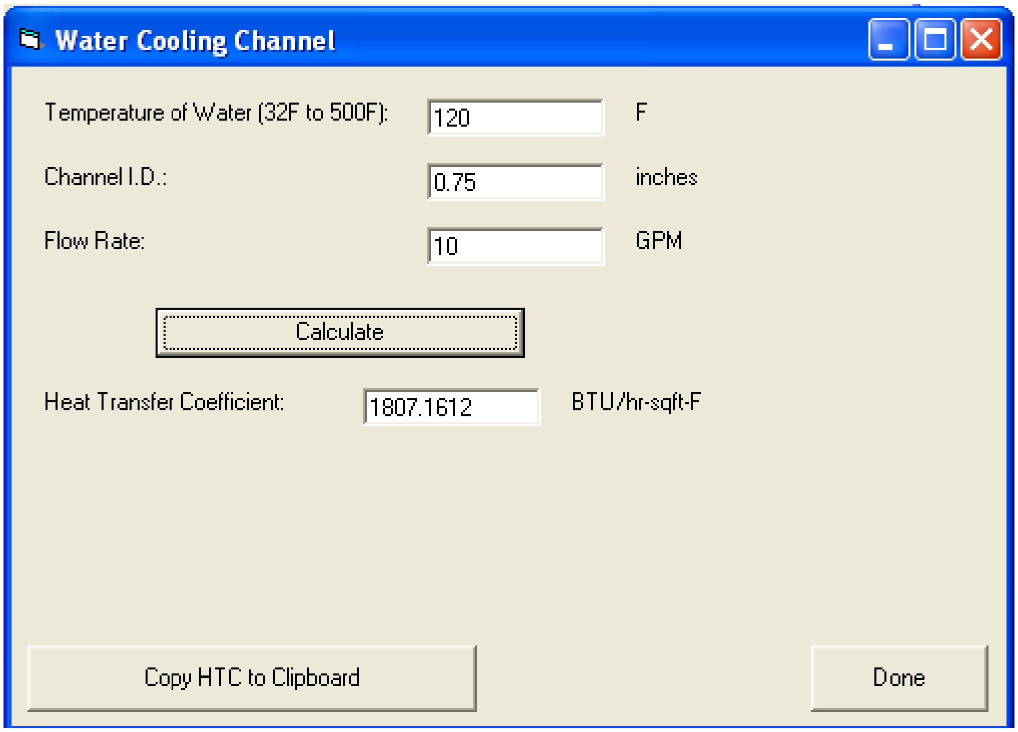
Radiation Heat Transfer from a Hot Surface – This function is used to calculate the effective HTC due to radiation heat transfer from a hot surface. Radiation is strongly dependent on temperature, and becomes much more important to consider as the metal and mold temperatures rise. Required inputs are the surface temperature and the ambient air temperature. Calculated output is the effective HTC, as shown here:



Note that you can use the button in the lower left corner to copy the calculated HTC value to the Windows Clipboard. Once you have done this you can use the Windows Paste command to enter that value into the HTC table in the Materials List. An example of this is shown at the end of this section.

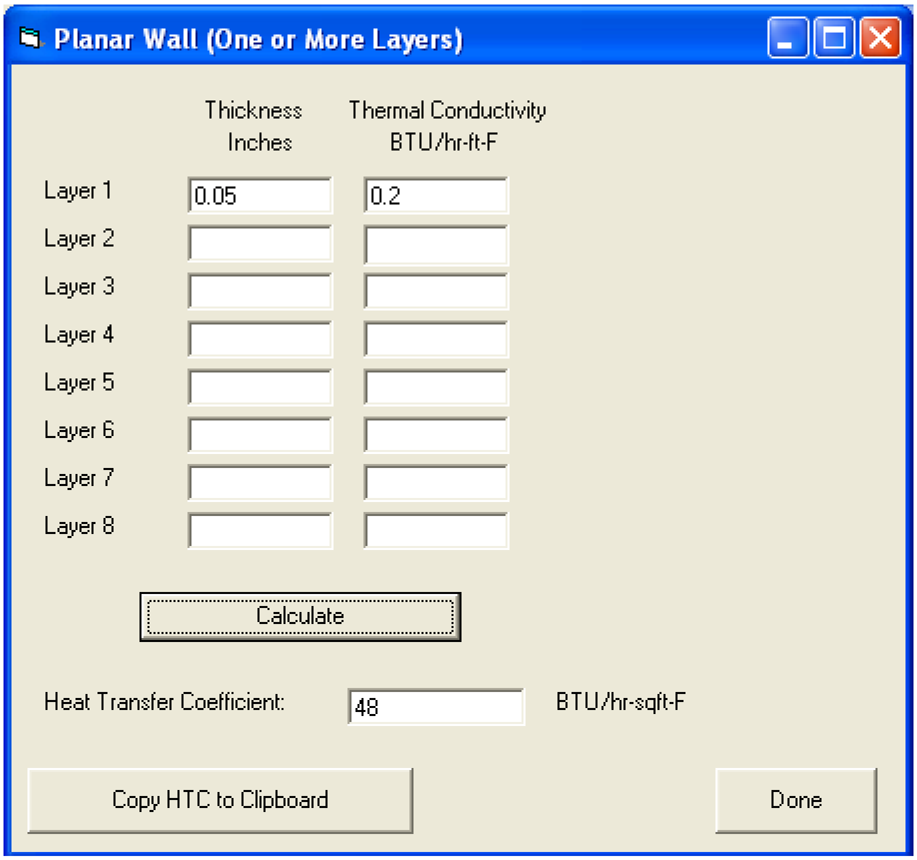
In the case of radiation, the HTC is normally added to the calculated natural or forced convection HTC and the total value is used.

Water Cooling Channel – This function is used in the permanent mold, or gravity die process, when water is forced through a channel in a metal die to locally cool that portion of the die. Required inputs are the average water temperature in the channel, the diameter of the channel and the water flow rate. The calculated output is the effective HTC, as shown here:



Note that you can use the button in the lower left corner to copy the calculated HTC value to the Windows Clipboard. Once you have done this you can use the Windows Paste command to enter that value into the HTC table in the Materials List. An example of this is shown at the end of this section.

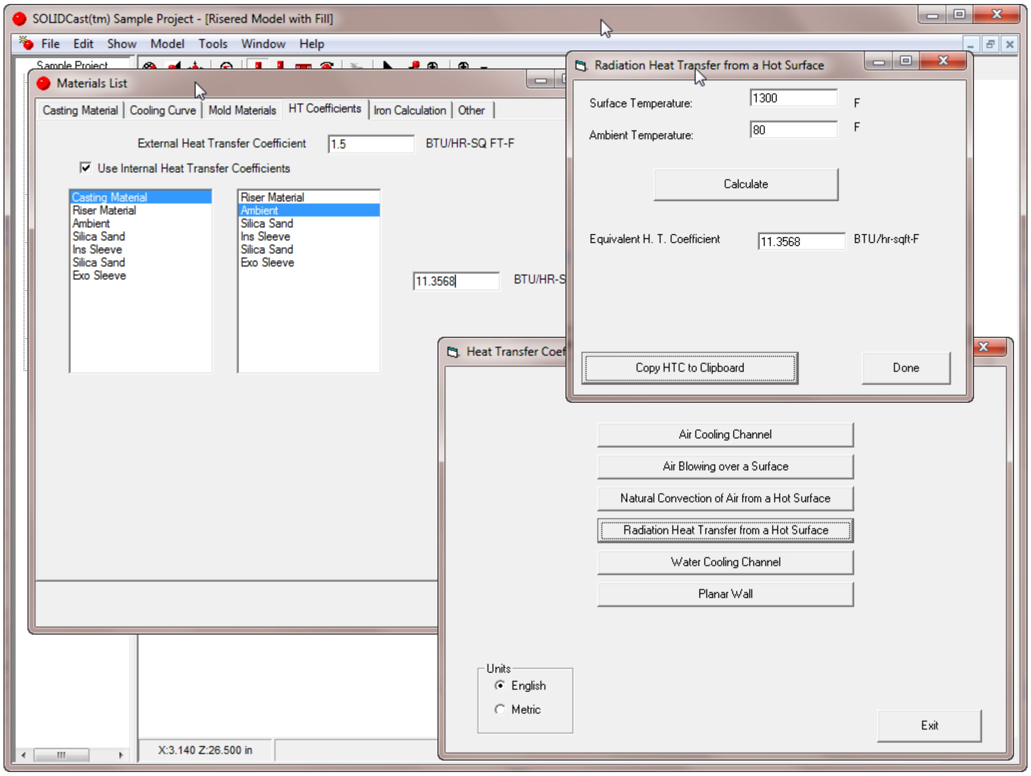
Planar Wall of One or More Layers – This function is typically used to calculate the effect of mold sprays or washes used to coat die surfaces in a permanent mold. The required inputs are the thickness of the coating layer and the Thermal Conductivity of the coating material. Note that you can calculate the cumulative effect of multiple layers of different coating materials. The calculated output is the overall effective HTC, as shown here:



Note that you can use the button in the lower left corner to copy the calculated HTC value to the Windows Clipboard. Once you have done this you can use the Windows Paste command to enter that value into the HTC table in the Materials List. An example of this is shown on the next page.

Using Calculated HTC Values – HTC values are entered into a project using the HT Coefficients Tab of the Materials List. Since the HTC Calculator is a stand-alone program, it can be run while the Materials List is open. Once a calculation is done, you can use the Copy HTC to Clipboard button to copy the result to the Windows Clipboard. Then you can highlight the specific entry on the HT Coefficients Tab and use the Windows Paste command to paste the calculated value.

The screen shot below shows the HT Coefficients Tab on the left, after pasting the result of a radiation calculation into the interface between Casting Material and Ambient Air.



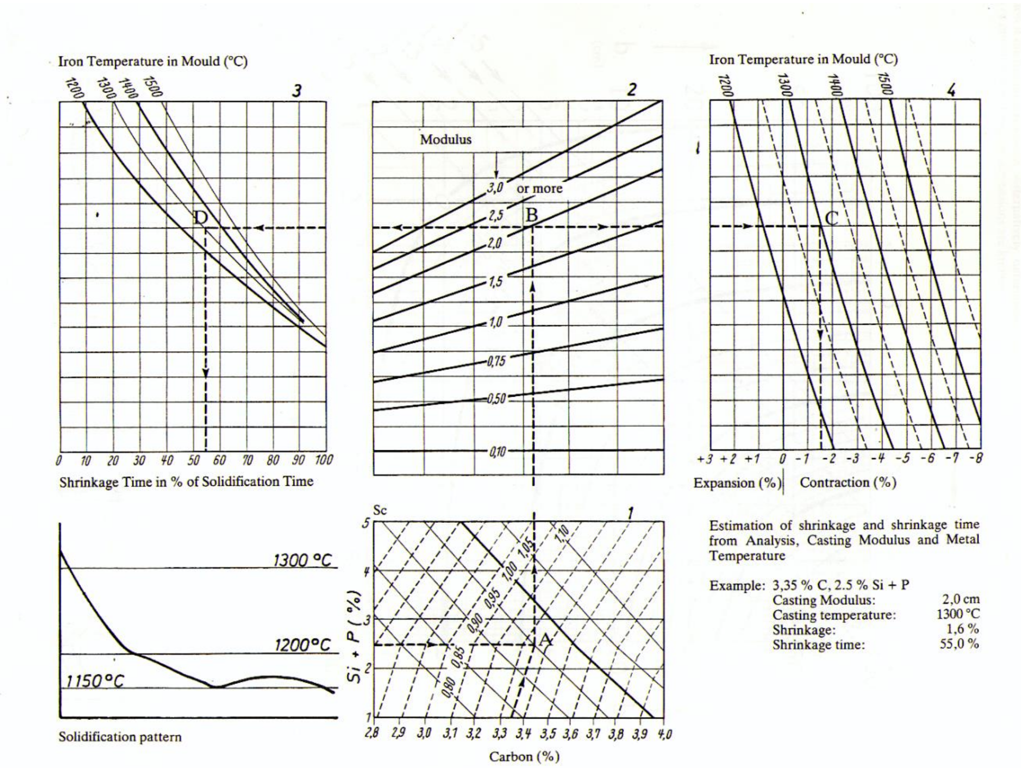
Once you are done with HTC calculations, click on the Exit button to return to the Model Building window.

UTILITY: Iron Property Calculator

This utility allows you to calculate the % contraction/expansion and CFS point for a given cast iron, based on the VDG Nomogram published by the German Iron Society. This utility also performs risering calculations for gray and ductile cast irons. Risering calculations for white cast iron, which shrinks throughout solidification and does not have graphite expansion, can be done using the SOLIDCast Riser Design Wizard.

These calculations can also be reached via the Iron Calculation Tab of the Materials List.

The printed version of the VDG Nomogram is shown here:



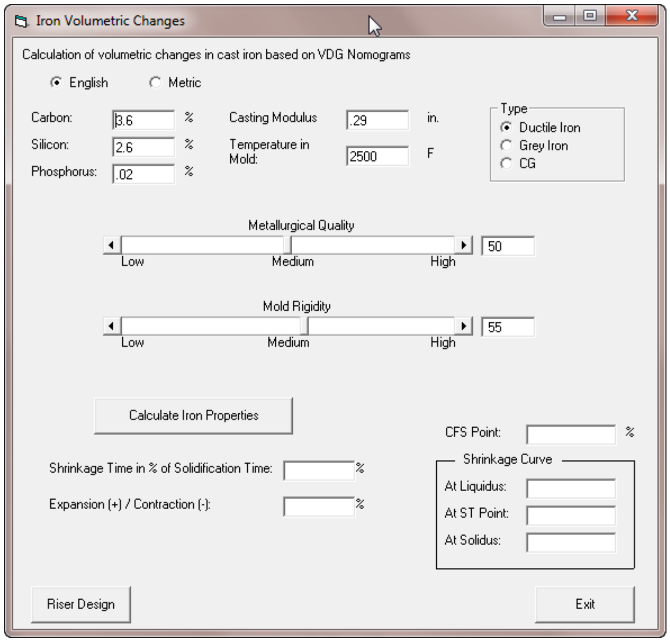
By selecting the composition (Carbon, Silicon and Phosphorus) the following the curve up to the Modulus value and across to the Temperature in Mold, you can determine the Shrinkage Time in % of Solidification, and the net amount of Metal Expansion or Contraction.

Note that this does not take mold wall movement into account. Additional metal contraction should be considered for non-rigid molds.

Also, this approach assumes that the temperature of the iron is the predominant temperature in the mold after pouring, not the pouring temperature.

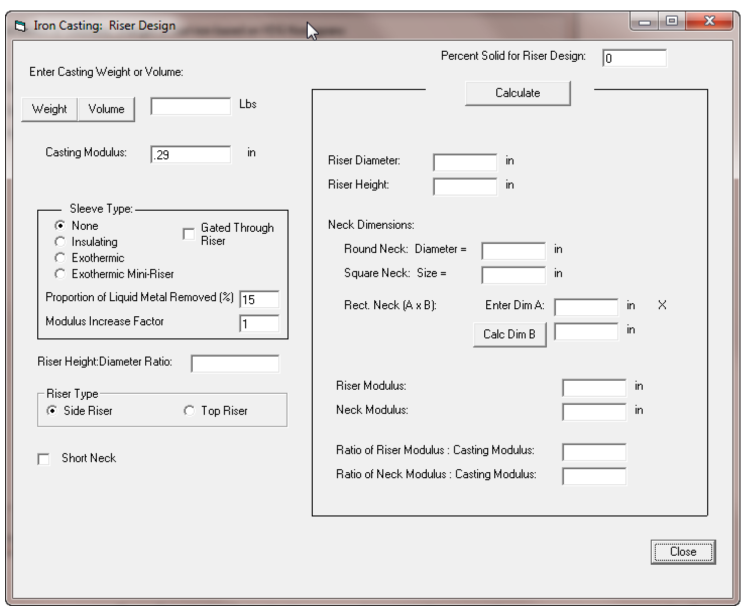
The Modulus value for a casting can be calculated and viewed using the Riser Design Wizard.

To start the utility, load your model, then go to Tools…Iron Property Calculator. The main screen for the Iron Property Calculator is shown here:



The worksheet will normally contain data from the last time the calculator was used. Once calculations have been done, you can go directly to Riser Design calculations for cast irons by clicking on the Riser Design Button.

The main screen for the Riser Design function is shown here:

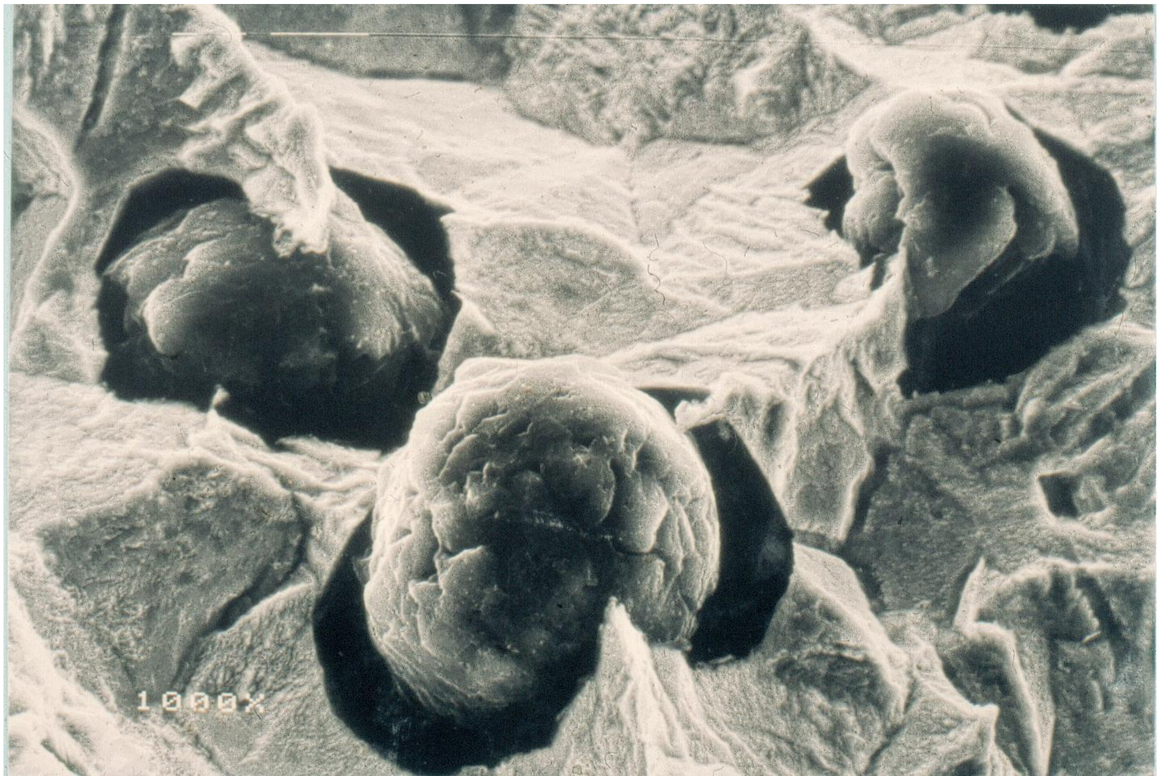


Before we go through the utility specifically, let's have a short discussion of the principles of riser design for cast irons.

RISER DESIGN PRINCIPLES FOR CAST IRONS

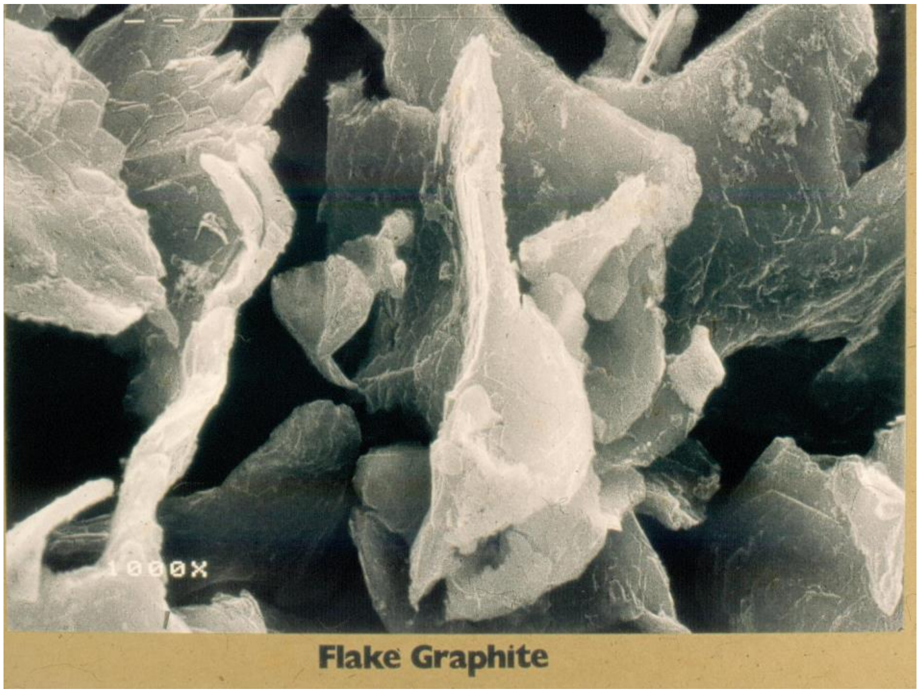
The solidification of iron castings (ductile, or nodular, iron and gray iron) is unique among cast metals, due to the precipitation of graphite as the iron solidifies. The graphite takes various forms depending on the type of iron. For example, in ductile iron (nodular iron), the graphite is primarily spheroidal, which gives the iron its characteristic ductility since the spheroids tend to reduce localized stresses under load.

The figure below shows typical graphite spheroids in a ductile iron:



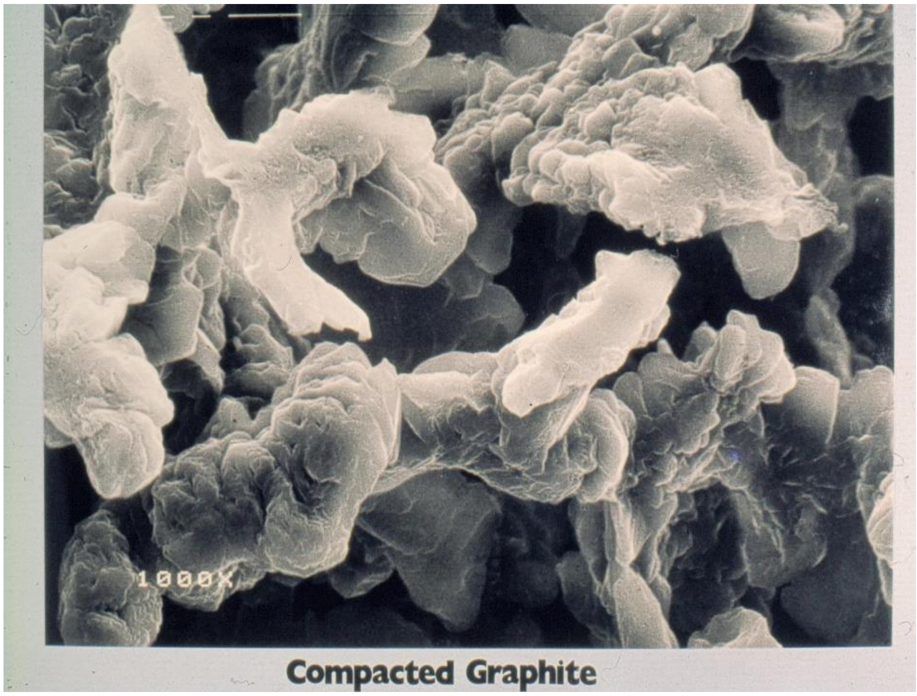
Spheroidal Graphite

In gray iron, the graphite tends to be in the shape of flakes, which results in gray iron's great compressive stress but relatively low tensile stress. Here is a photomicrograph showing gray iron's flake graphite structure:



Flake Graphite

And in compacted graphite iron, both the graphite structure and mechanical properties tend to be somewhat intermediate between ductile and gray irons. CG structure can be seen here:

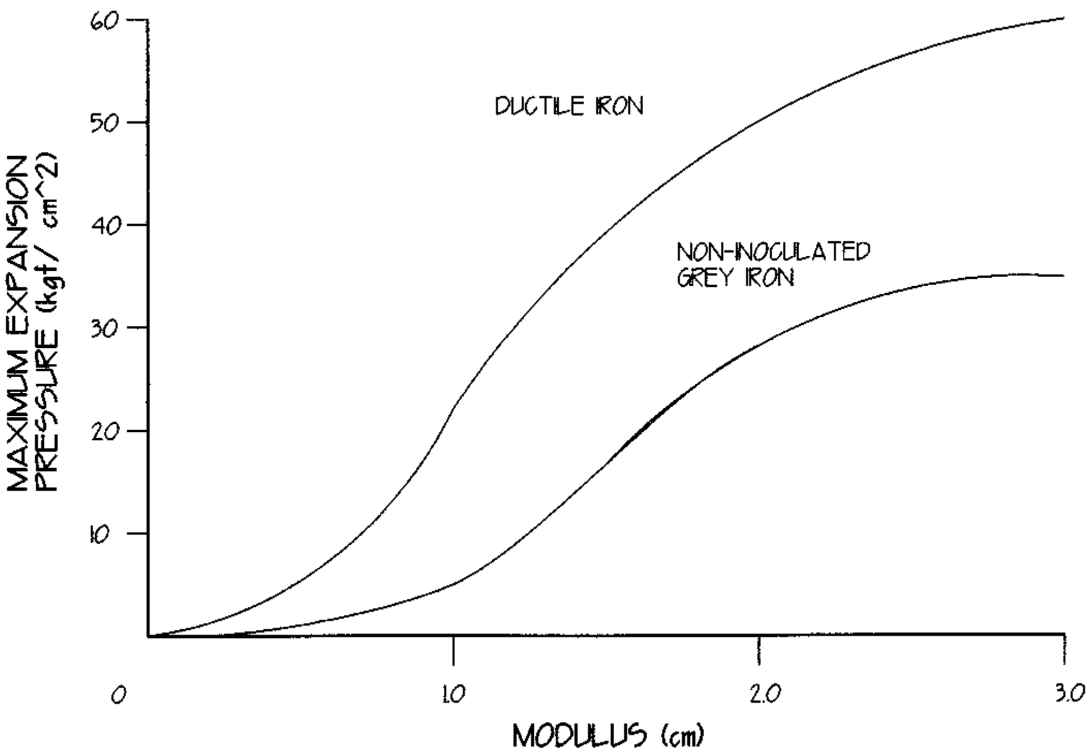


Compacted Graphite

The precipitation of graphite during solidification causes some expansion to occur, because the graphite is much less dense than the surrounding iron. Therefore, we have several forces at work. The liquid iron tends to contract as do almost all liquids when cooled. During solidification, the austenitic iron also contracts as do most metals upon solidification. However, the precipitating graphite causes an expansion pressure that can be used to our advantage if the feeding system is properly designed.

No matter which type of cast iron we are pouring, the secret of good design is to provide a feeding system to compensate for the liquid shrinkage and then allow the expansion (due to graphite precipitation) to provide enough pressure to produce a sound casting.

There are some differences in expansion pressure between ductile iron and gray iron as shown in the following chart:



Regardless of the type of iron casting we are designing, there is a set of basic procedures that can be followed which will help us to ensure high-quality, sound iron castings. These procedures are outlined in the following section, and then a full example will be shown, using the Iron Property Calculator and Riser Design utility. These procedures will help to minimize the number of sample castings required, reduce lead times to get into production, and result in consistent-quality castings which will make our customers happy and our foundry more profitable.

RISER DESIGN PROCEDURES FOR CAST IRONS

Step 1

Run a simulation of just the casting, surrounded by mold material, without gates or risers. We call this ‘freezing naked’. This simulation provides data on how the casting design and geometry affect solidification, and forms the basis for thermal modulus calculation and riser design.

Step 2

Run the SOLIDCast Riser Design Wizard and select "Calculate and Display Casting Modulus" to find out the maximum effective modulus of the casting. Take note of this value and exit the wizard.

Step 3

Select Tools…Iron Property Calculator to calculate the Shrinkage Time (ST) and net Percent Expansion (+) or Contraction (-) of the iron based on chemistry, modulus and temperature in the mold. This is based on the VDG Nomograms for iron properties. Or go to Model…Materials List..Iron Calculation and make adjustments there. The advantage of that method is that the curves of the iron will be automatically adjusted by the software.

Step 4

The calculations provide the net expansion of the iron considering the metal, dilation of the mold and metallurgical quality of the metal.

Step 5

RULE: Recommended practice would be to use a "hot" riser, i.e., to use a side riser and gate directly into the riser if so that the amount of heat in the riser, and its ability to provide feed metal, is maximized. We would typically recommend a sleeve (either exothermic or insulating) to retain the heat for top risers, since it is difficult to gate into these.

Step 6

The proportion of liquid metal that can be supplied by a riser can be estimated by knowing its condition. For example, a hot side riser can typically provide about 20% of its metal for feeding, while a typical cold side riser might provide around 14% of its metal. A sleeved riser can provide anywhere from 33% to 35% depending on its condition. Exothermic mini-risers have been known to provide up to 70% of their metal for feeding.

Step 7

The formula relating the available volume of metal in a riser to the volume of the casting can be expressed as:

$$V_f = \frac{V_c * S}{x}$$

Where

V_f = Riser Volume
V_c = Casting Volume
S = Feeding Requirement (including Mold Dilation)
x = Proportion of Liquid Metal Removed from Riser

From this the diameter of the riser can be calculated if an assumption of riser Height:Diameter is made.

Step 8

The riser neck should be sized so that its modulus guarantees that it will freeze at the point that the liquid shrinkage is done and any subsequent expansion will be controlled and contained within the casting to prevent shrinkage porosity formation. This is done using the following formula:

$$M_n = \sqrt{(ST/100)} * M_c$$

Where

M_n = Modulus of the Neck (Transfer Modulus)
ST = Shrinkage Time
M_c = Modulus of the Casting

Step 9

Also, in order for the riser to provide sufficient liquid melt during the shrinkage period, its modulus should be 20% greater than the neck modulus (transfer modulus), which means the riser size should satisfy the equation:

$M_r = 1.2 * M_n$

Or

$M_r = 1.2 * \sqrt{(ST/100)} * M_c$

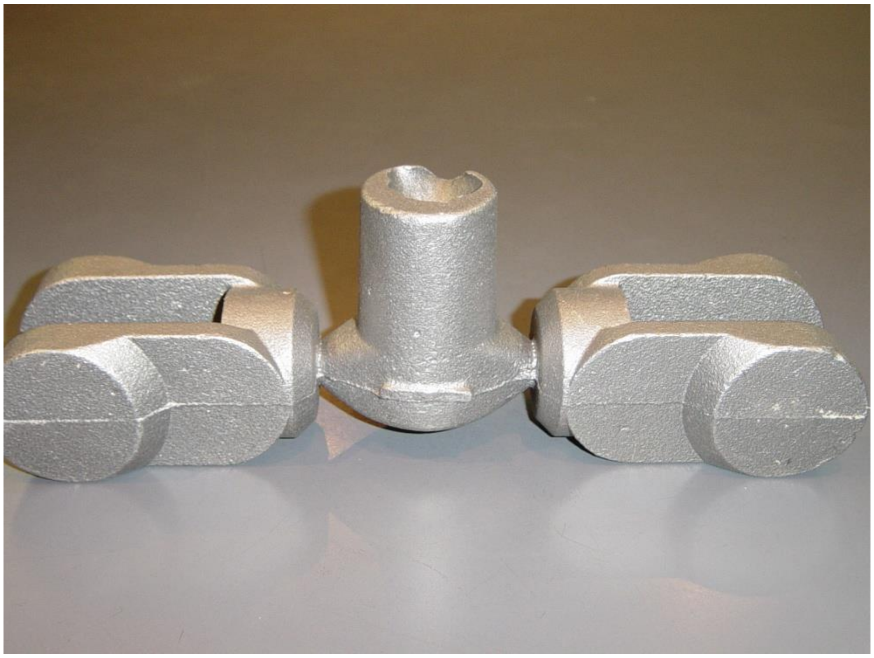
So that if the riser is sized to satisfy the liquid feeding requirement but it does not satisfy this modulus requirement, its size must be increased to satisfy this modulus requirement.

Step 10

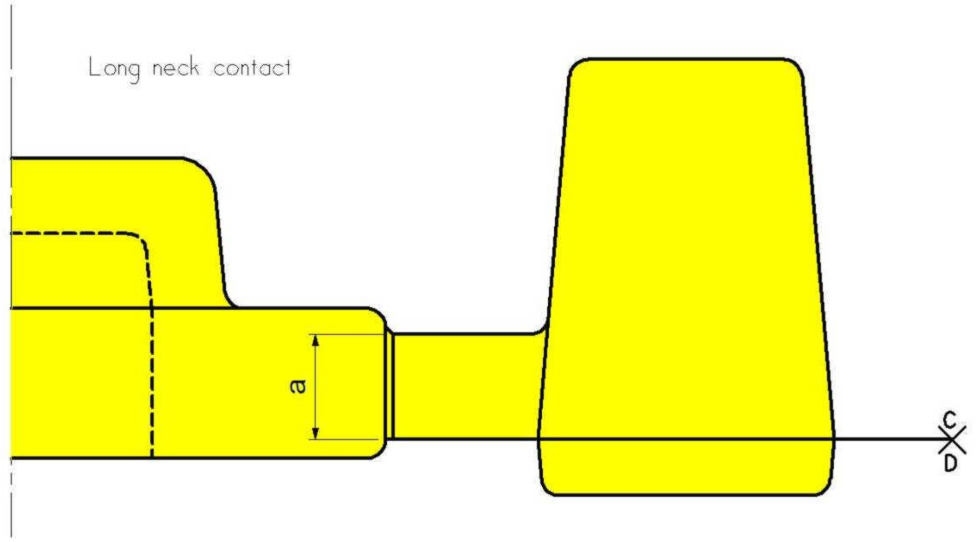
If the riser is close enough to the casting so that mold heatup between the casting and riser can be taken into account, the neck modulus can be reduced by multiplying by a factor of 0.6.

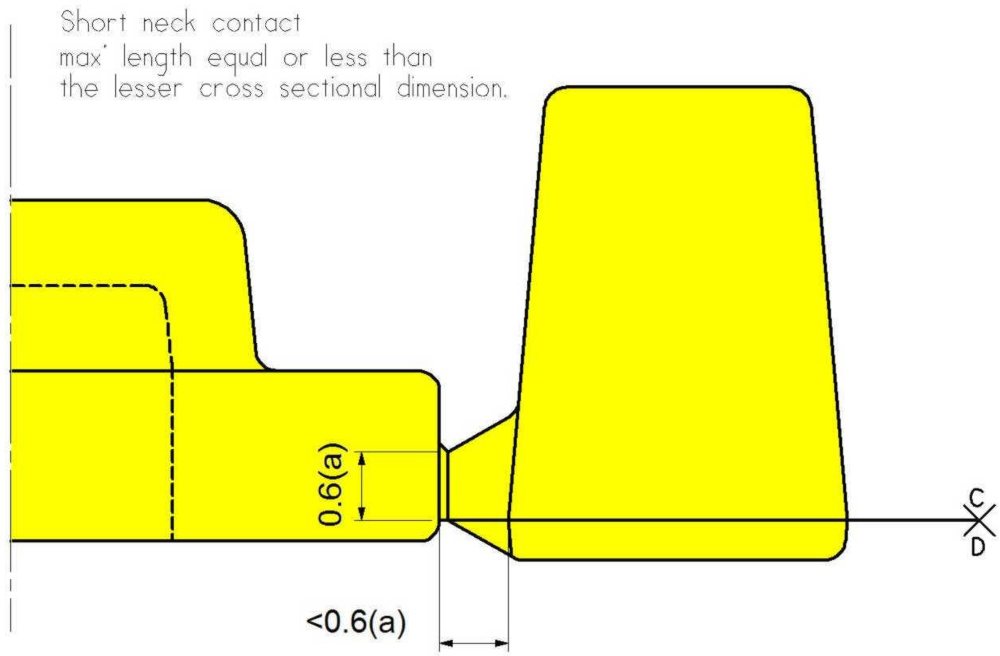
RULE: In order to be considered a short neck, the distance between the casting and the riser should be less than the minimum dimension of the riser neck.

The figure below shows a typical short neck where the riser is closer to the casting than the short dimension on the riser contact:



Schematic diagrams of short and long necks are shown here:





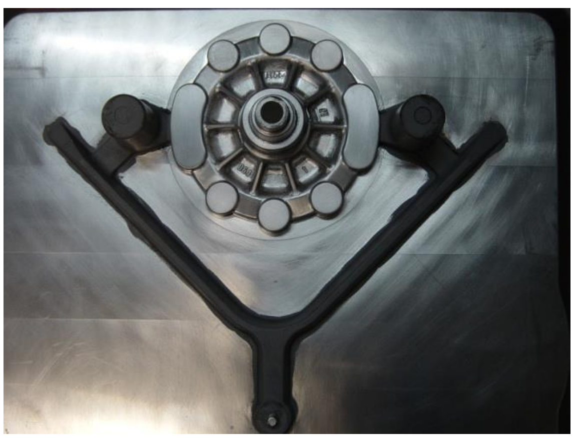
Step 11

How many risers are required for a casting?

RULE: Only one riser should be used for each feeding zone within a casting. Feeding zones can be visualized in SOLIDCast by plotting the neck modulus, also called the Transfer Modulus. If more than one riser is used for a single feeding zone, in almost all cases only one of the risers will pipe and the other riser(s) will skin over and not pipe, but will create a thermal hot spot underneath at which some shrinkage porosity will be likely to appear.

Shown below are some examples that appeared to have required more than one riser, but in reality had only one feeding zone:






Note that in these over-risered situations, it is not always the same riser that pipes first. Variables in the pouring and casting process may favor one riser in one mold and another riser in the next mold. However, in each case, once the first riser begins to pipe, the other riser(s) skin over and cease to function.

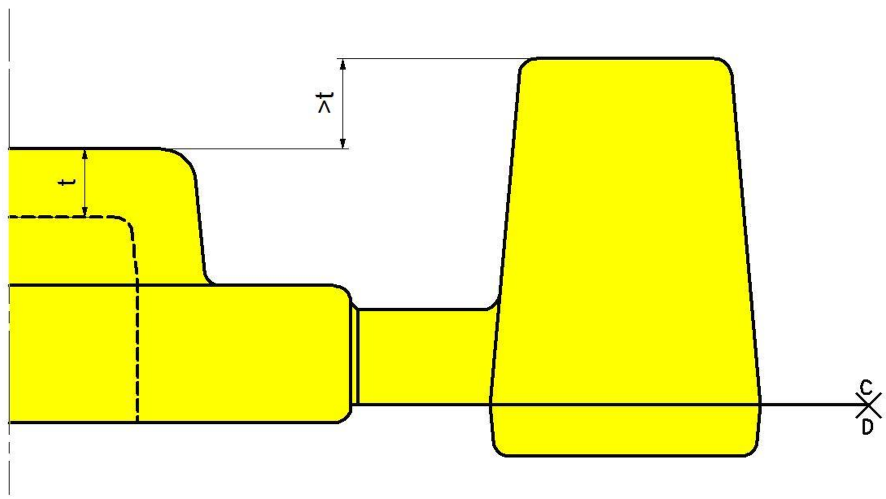
Step 12

SOLIDCast uses the THERMAL, or EFFECTIVE MODULUS to calculate the location and extent of feeding zones within the casting. This is superior to the traditional measure of modulus, which is the strictly the geometric Volume:Surface Area Ratio, as it is able to take into account heat saturation of mold and core pockets as well as heat extraction by chills and, if desired, temperature distribution due to filling. In other words, it accounts for the dynamics of metal casting, including alloy type, molding method and more.

Step 13

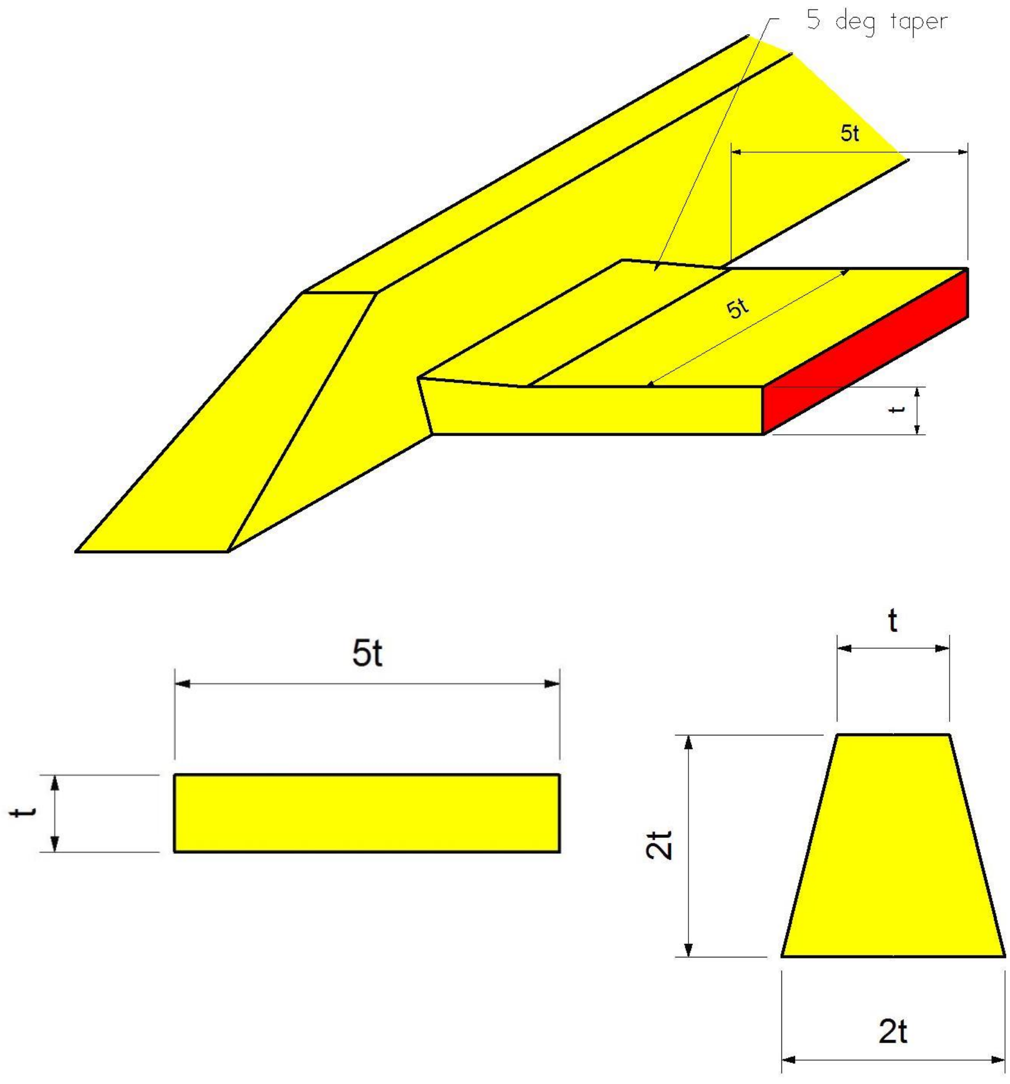
The Iron Property Calculator Riser Design program assumes that side risers are cylindrical with a hemispherical bottom, while top risers are cylindrical in shape.

RULE: The tops of the risers should be above the highest point of the casting for gray and ductile iron castings, by at least the minimum section thickness.



Step 14

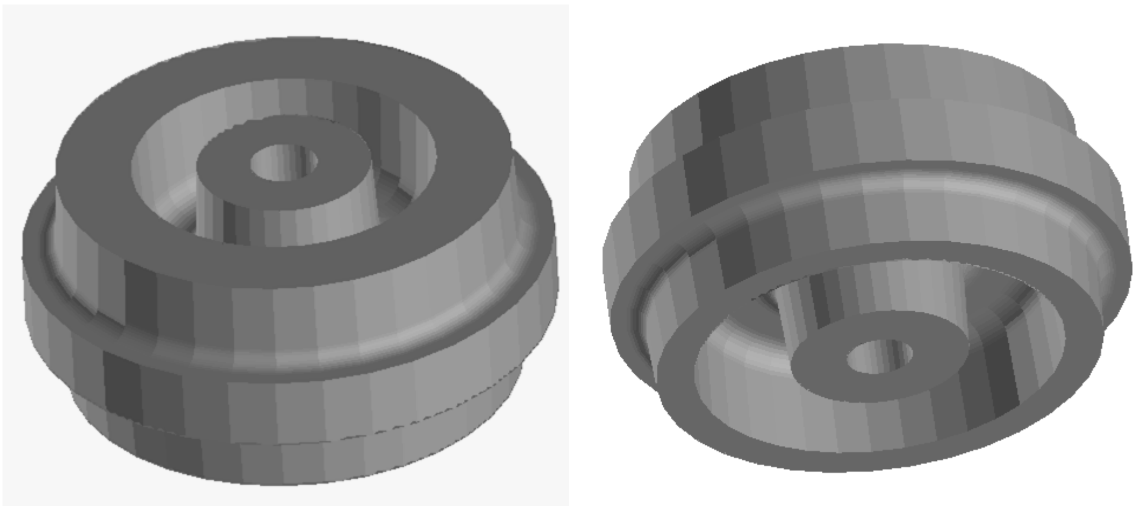
Gating should be designed to freeze relatively quickly after the liquid metal has filled the mold cavity. In general, this means that the gate attachment to the casting should have a 5:1 ratio of width to height to ensure relatively quick freezing so that expansion pressure can be contained. Remember, control of expansion pressure is our ultimate goal in feeding cast iron.



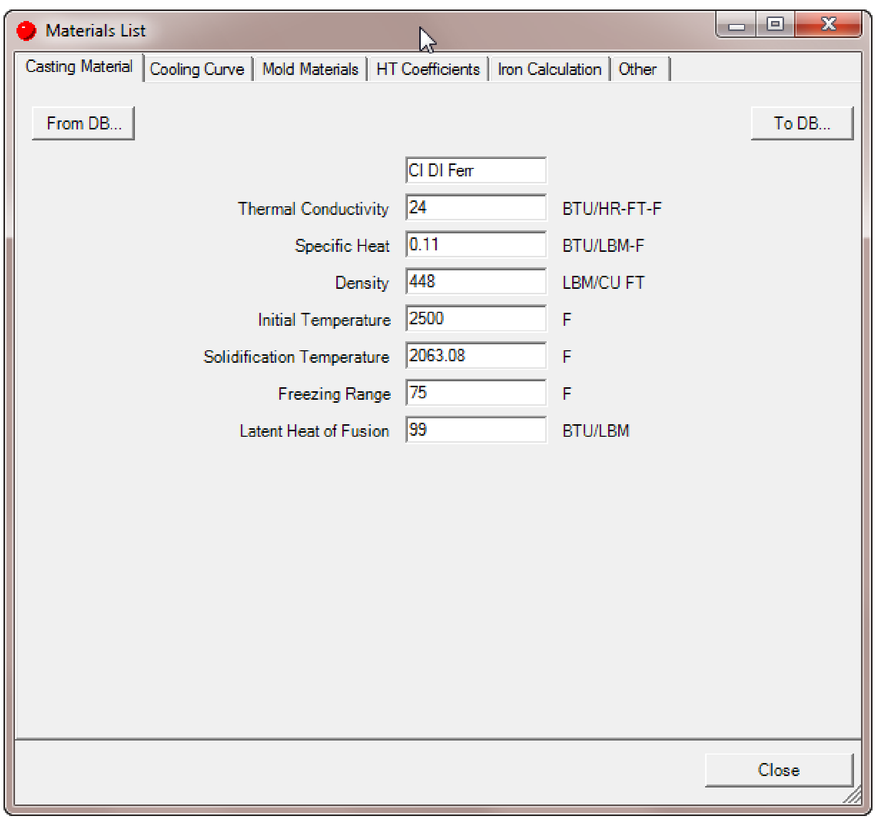
All of these calculations can be performed quickly and easily in SOLIDCast.

CAST IRON RISER DESIGN EXAMPLE

As an example, consider the task of designing a feeding system for the following ductile iron casting. The basic casting shape as imported from a CAD system in SOLIDCast is shown here:

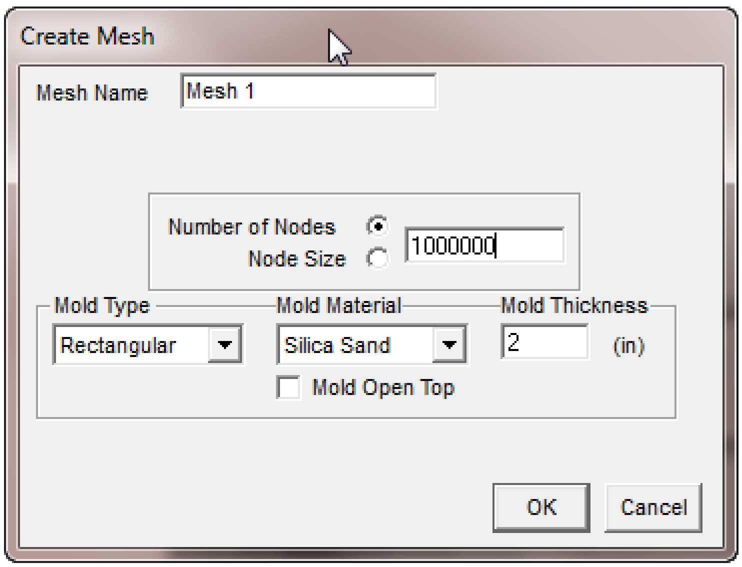


Ferritic Ductile Iron is selected as the base Casting Material from the SOLIDCast database. Standard properties are shown here:

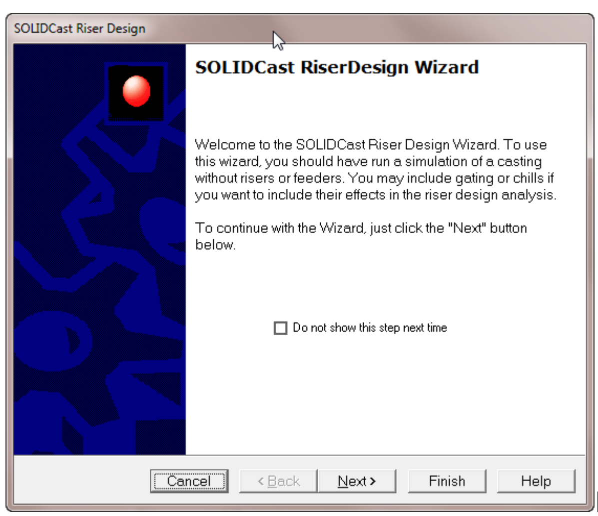


Note that you should always adjust the Initial(Pouring) Temperature to the value you will be using in your own shop. The Pouring Temperatures listed in the SOLIDCast database are simply placeholders, and are not to be taken as recommending values. Pouring temperature will vary by individual casting types and sizes.

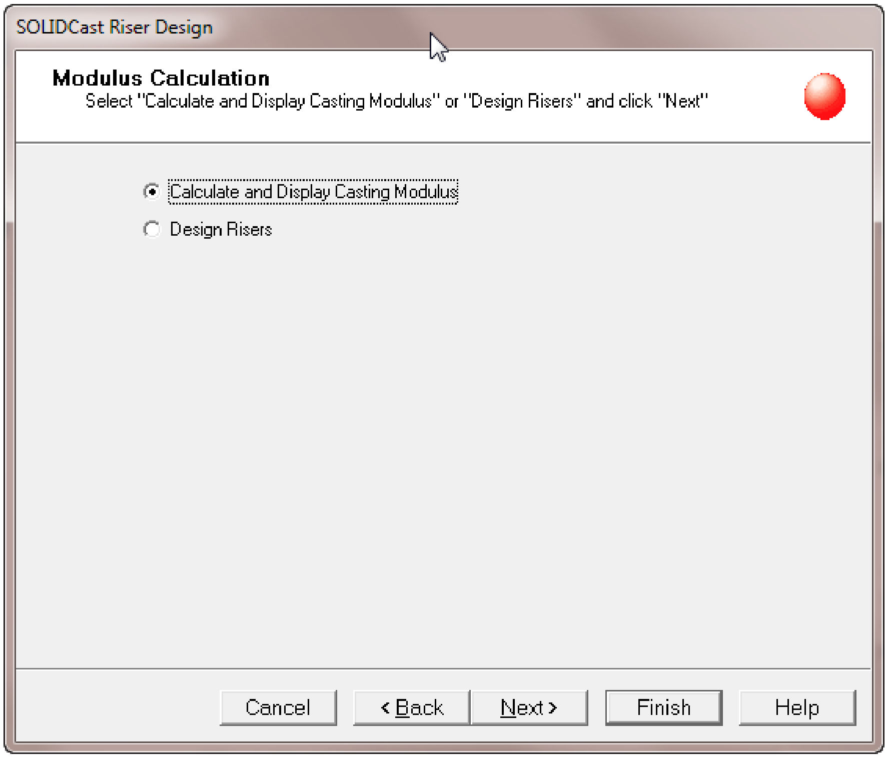
The casting is meshed with no risers or gates, so that a “thermal” simulation can be run for calculation of the Thermal Modulus of the casting. In general, you can use a fairly coarse mesh at this time, so that the simulation can run very quickly. If possible, use planes of symmetry to reduce model size. Also, since no rigging system is present in the model, make sure that the check box labeled Mold Open Top is cleared, so that the mold material will be meshed over the top of the casting, too.



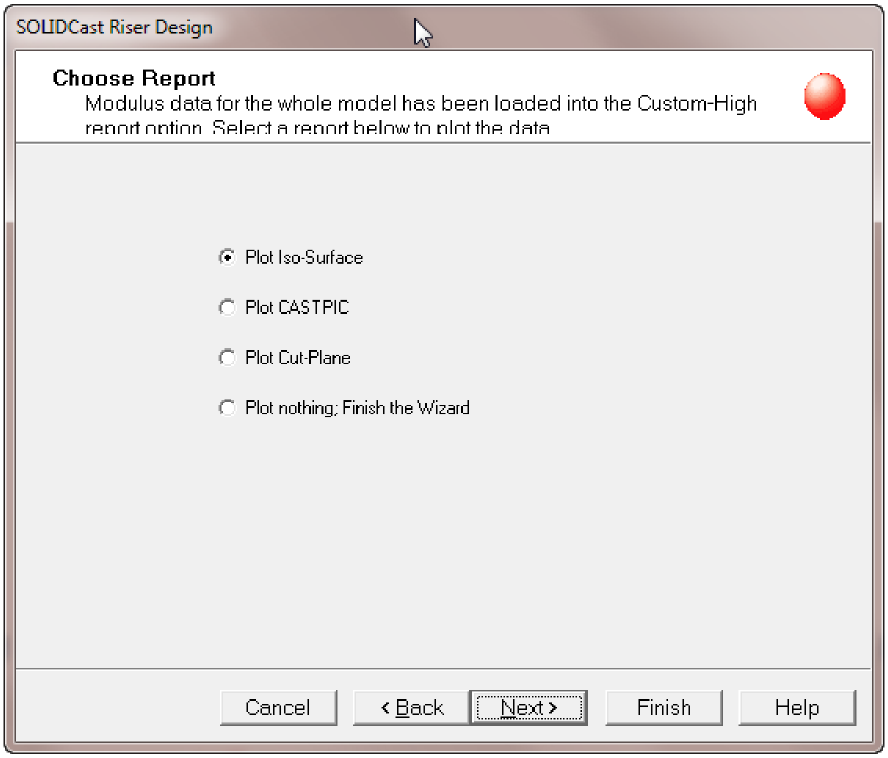
Once the simulation is complete, load simulation results by double-clicking on the simulation icon in the Project Tree. Then select Simulation…Riser Design Wizard. We are going into this wizard to get the maximum thermal modulus information, which is an input to the Iron Property Calculator. These steps are shown on the following pages.

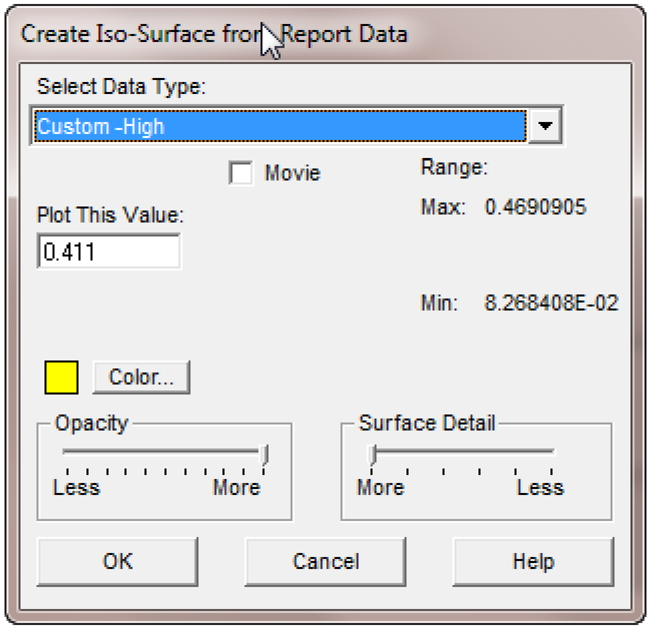


Select Calculate and Display Casting Modulus, then click Next.



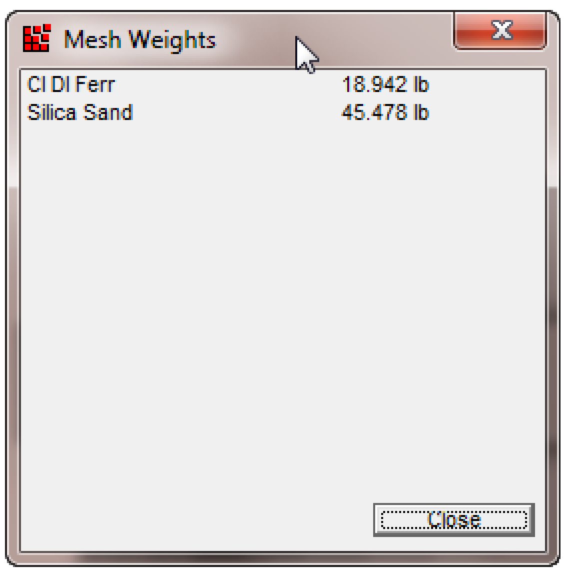
Select Plot Iso-Surface, then click Next.



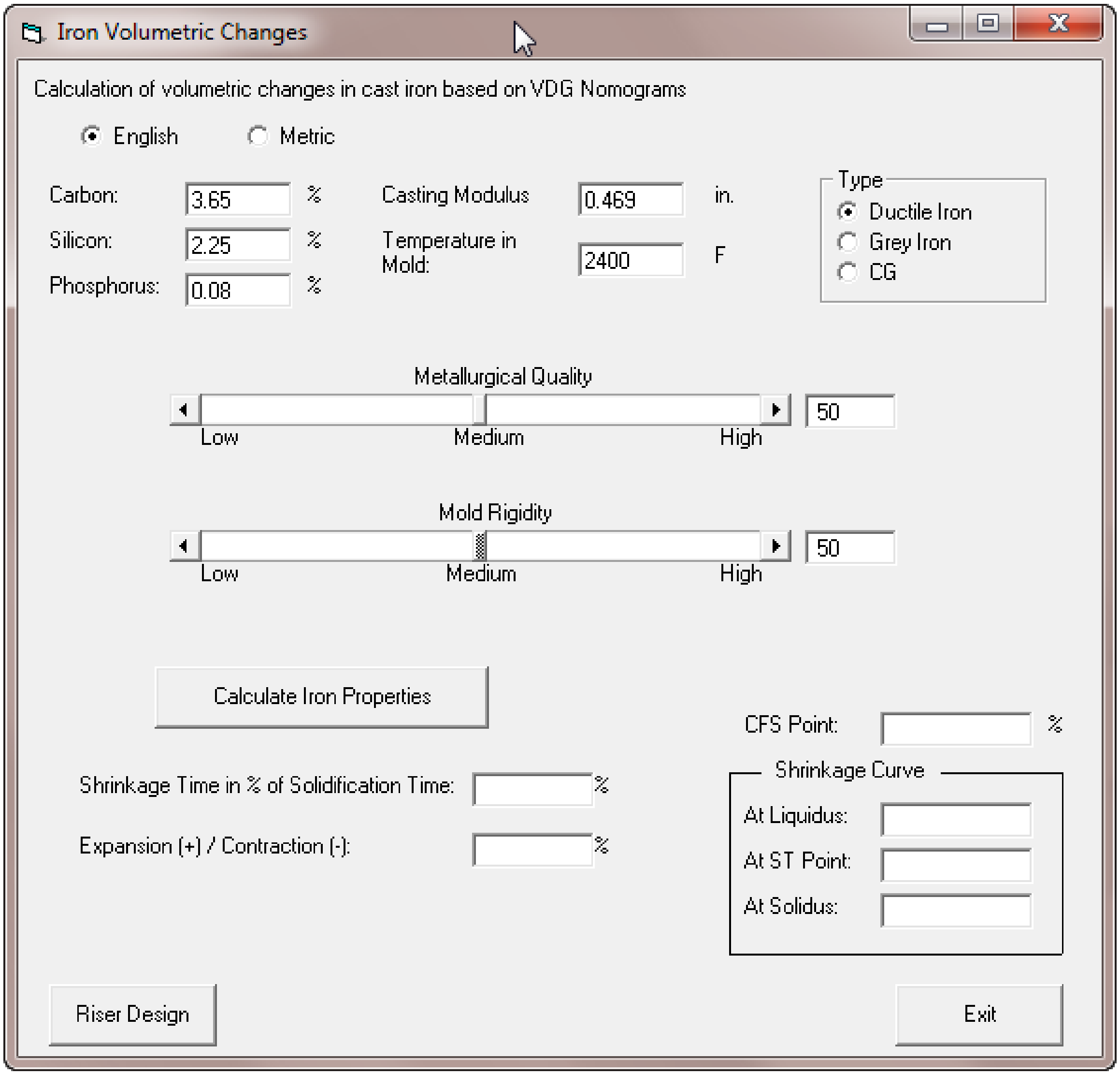


From the Iso-Surface Plot Menu, we can read that the maximum Modulus of this casting is about 0.469 in. At this point, this is all the information we need from the wizard, so we can just press Cancel to avoid making the plot at this time.

Another item of information we'll need is the weight of the casting. This can be easily obtained by selecting Mesh…Weights from the main menu. Here we can see that the casting weighs 18.942lb.



Now we are ready to calculate the properties of the iron, and the required riser size. From the main menu, select Tools…Iron Property Calculator. You will see a screen similar to that shown on the next page.



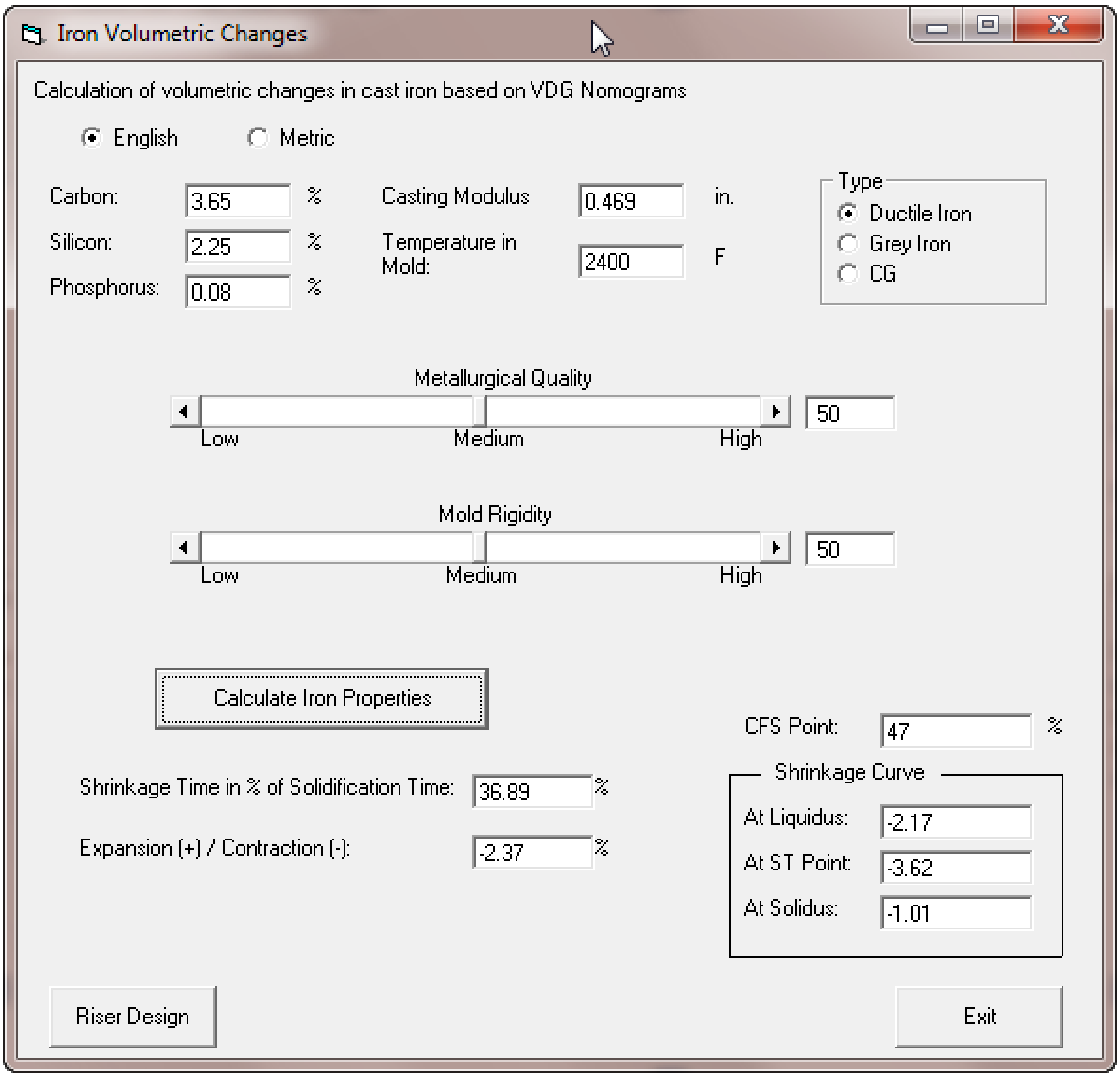
Enter the Carbon, Silicon and Phosphorus content of the iron. In this example we will have entered C = 3.65%, Si = 2.25% and P = 0.08%.

Enter the Casting Modulus as previously calculated by the Riser Design Wizard. In this example we have entered 0.469in.

Enter an estimate of the metal Temperature in Mold. If you have not done a filling simulation, you can typically estimate a temperature of 75-150F less than the pouring temperature. We have used a temperature drop of 100F, to 2400F, from the pouring temperature of 2500F.

Two additional items to be considered are the Metallurgical Quality and the Mold Rigidity. Metallurgical Quality affects when the Critical Fraction Solid Point is reached, and how much expansion takes place. Mold Rigidity affects the overall amount of shrinkage created by mold dilation. These factors are relative, and can vary from 1 to 100. For this example we have used mid-range values of 50 for each one.

Clicking the Calculate Iron Properties button causes the system to display the Shrinkage Time (ST) in terms of % Solid, the net amount of Expansion (+) or Contraction (-) which occurs up to that Shrinkage Time, the Critical Fraction Solid (CFS) Point, and the three main points on the Shrinkage Curve. This data can be used later to manually adjust the curves by going to the Cooling Curve Tab on the Materials List. Or, if the calculations were done on the Iron Calculation Tab of the Materials List, the curves will be updated automatically.



Now that we have calculated the properties of the iron, we can design a riser for this casting. To do this, click on the Riser Design button at the bottom left side of the window.

First, we enter the casting weight as previously calculated by the system in the Mesh menu. If you know the volume of the casting, you can click on the Volume button and enter that rather than a weight.

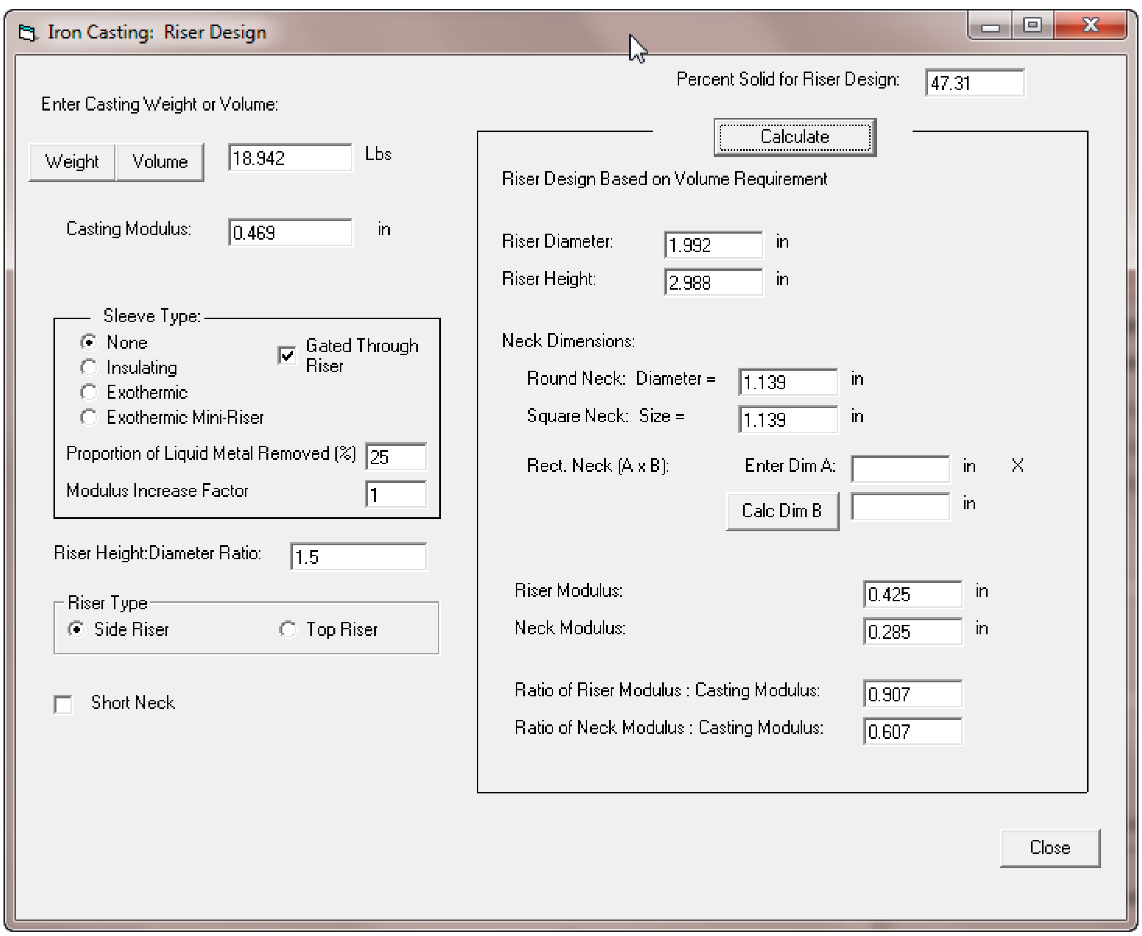
The Casting Modulus will have already been entered by the Iron Property Calculator.

The next item to choose is sleeve type. You can select either a sand riser (no sleeve), an insulating riser, an exothermic riser or an exothermic mini-riser. You can also select whether the casting is gated through the riser. The proportion of liquid metal removed from the riser is automatically adjusted. We have chosen a sand riser which is hot, that is, we are gating through the riser.

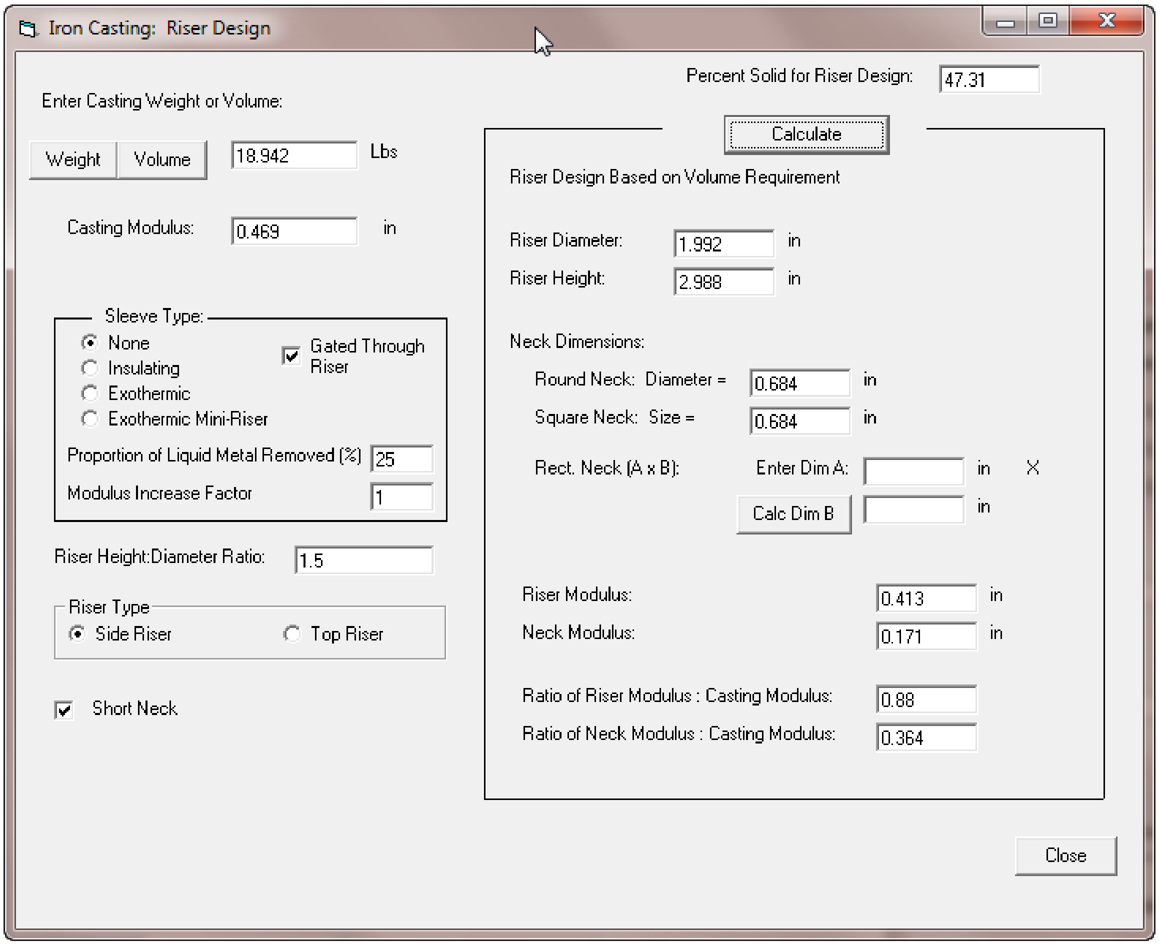
Next select the ratio of Height to Diameter that you want to use for the riser design. We have chosen a 1.5:1 HD Ratio.

Finally, select either a Top or Side Riser. Here we have selected a Side Riser.

Press the Calculate button to display the required riser and neck size, as well as the Modulus of the neck and the riser. Note that the Neck Modulus is also referred to as the Transfer Modulus and can be used to indicate how many feeding zones (and now many required risers) there are for this casting.

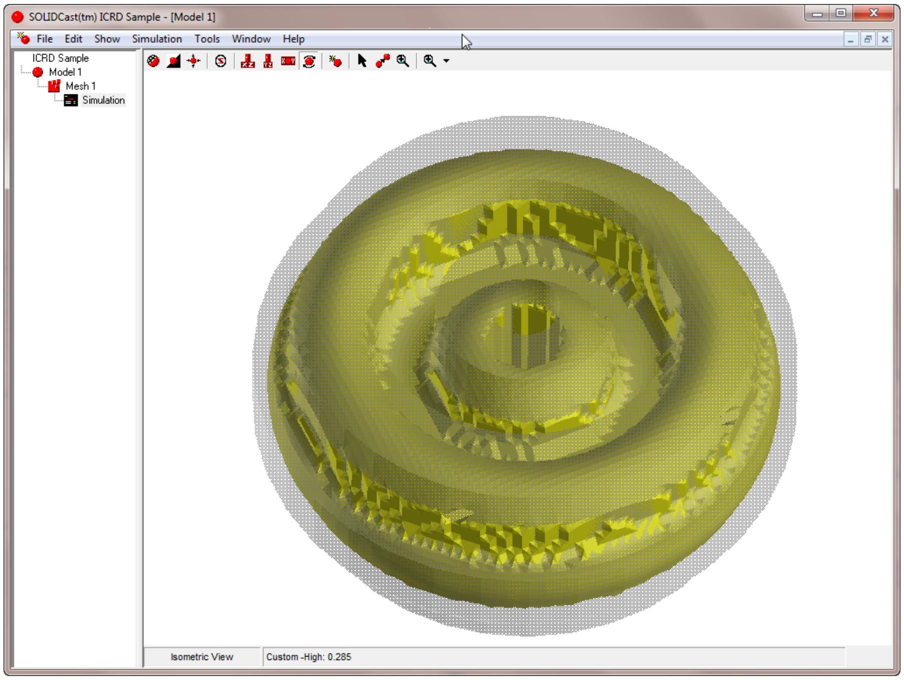


Notice that by checking the Short Neck option and repressing the Calculate button, you can calculate a neck size for a riser which is very close to the casting (closer than the minimum dimension of the neck). This reduction is possible due to the additional heating of the next by the metal.



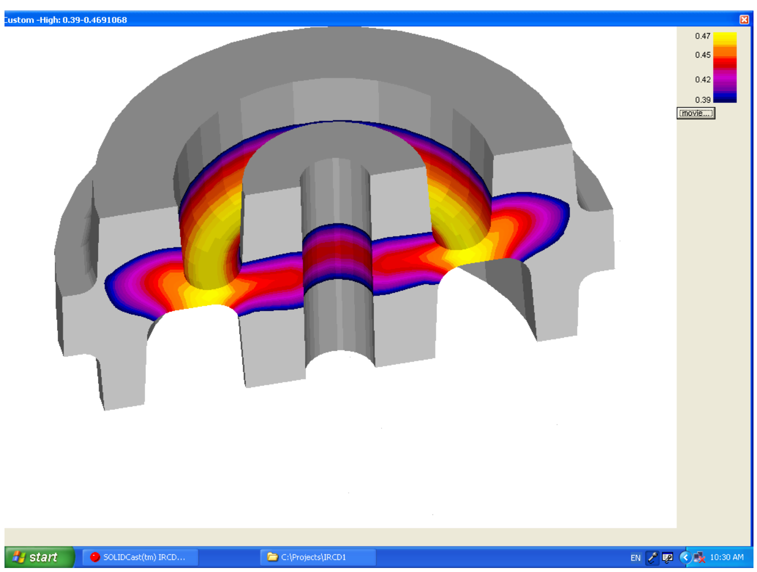
Press the Calculate button to display the required riser and neck size, as well as the Modulus of the neck and the riser. Note that the Neck Modulus is also referred to as the Transfer Modulus and can be used to indicate how many feeding zones (and now many required risers) there are for this casting.

To find the number of feeding zones on the casting, go back to Simulation…Riser Design Wizard, and plot the Neck, or Transfer, Modulus. In our example the value is 0.285. The plot is shown here:

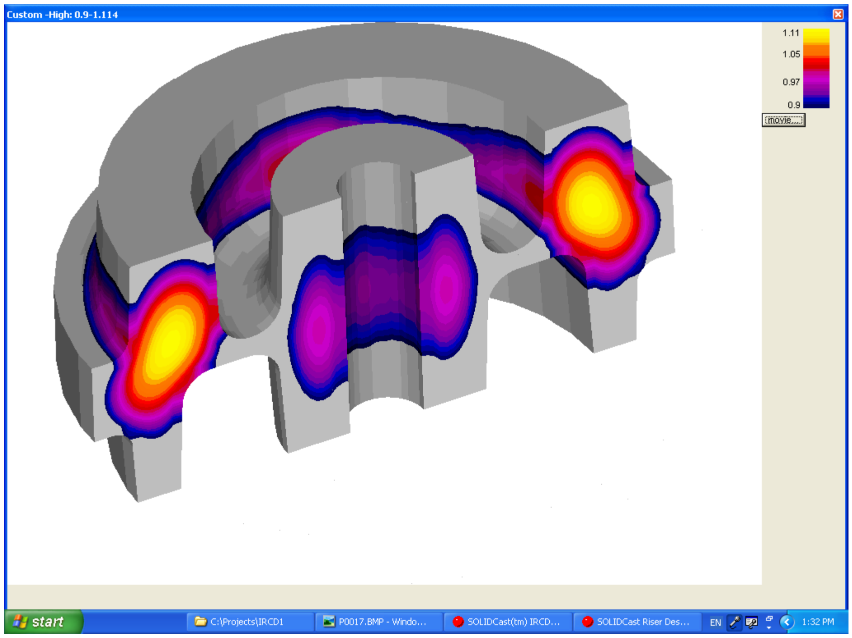


Plotting the Transfer Modulus as an Iso-Surface Plot shows that the entire casting is one feeding zone, so only one riser is required and the riser calculations are complete. If we had seen more than one feeding zone, then we would have had to design a separate riser for each zone.

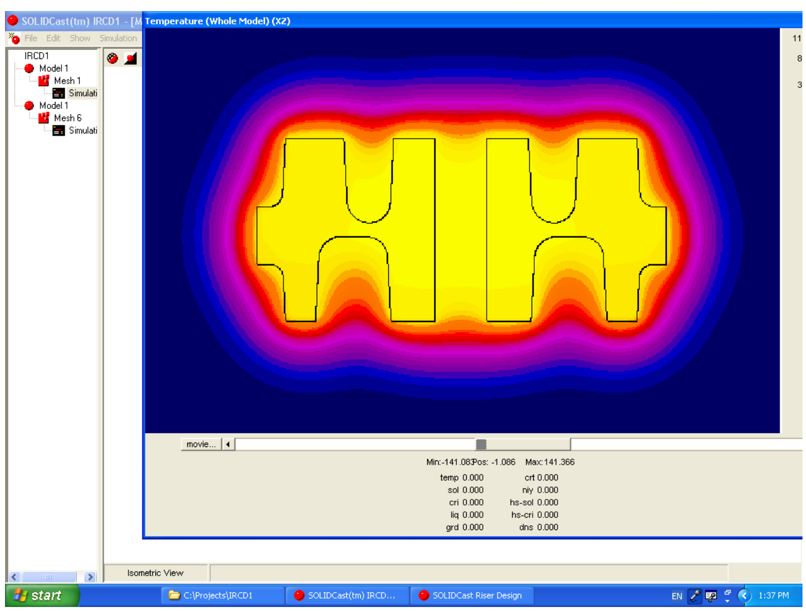
This figure shows another image of the Transfer Modulus using CastPic plotting. It also shows one zone that means one riser is required for this casting.



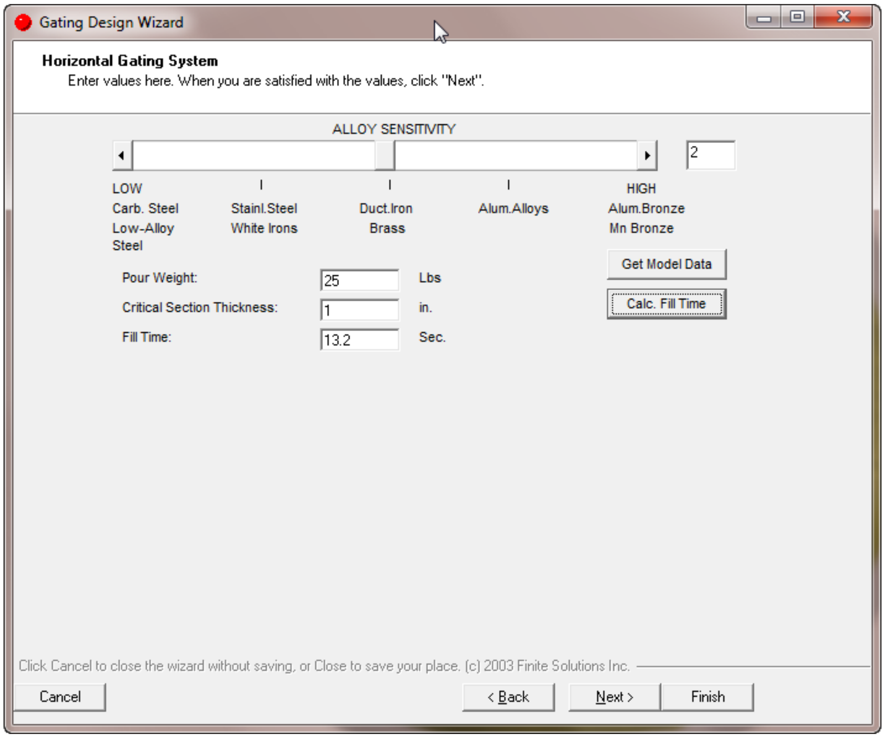
The traditional Geometric Modulus calculation using Volume/Surface Area Ratio would have indicated two separate feeding areas, one in the central hub and one around the outer rim as shown below. Why is there a difference?



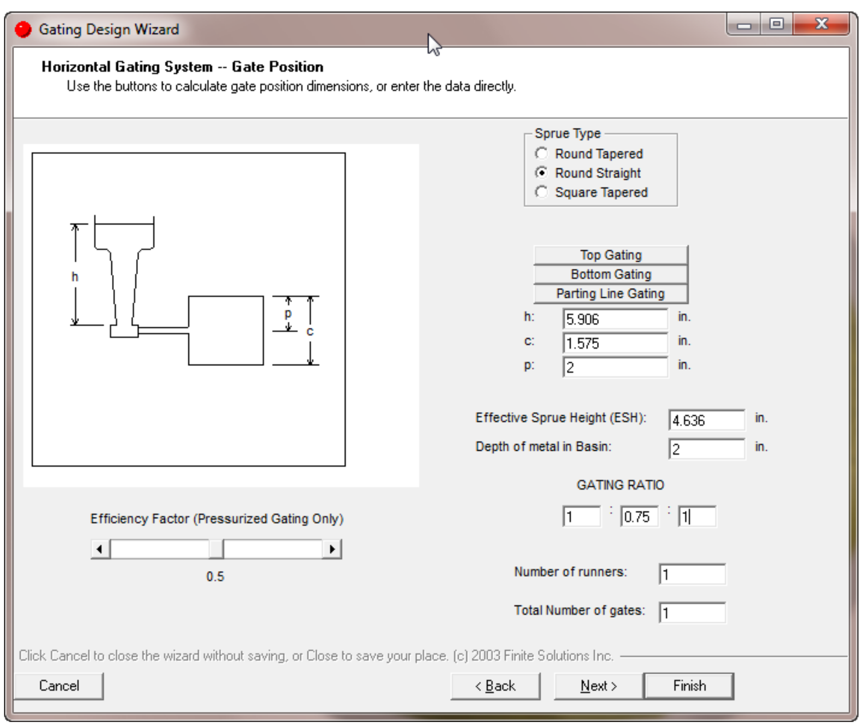
The figure below is a cross-sectional view through the casting and mold and shows temperature at the end of solidification. You can see that the mold material becomes saturated with heat in the “pocket” areas between the inner hub and the outer rim, which keeps the thinner sections hot. This effect would not be captured by performing the old-fashioned Volume/Surface Area Modulus calculations, but is automatically taken into account when performing the Thermal Modulus function in SOLIDCast, because dynamic thermal effects in the mold are simulated.



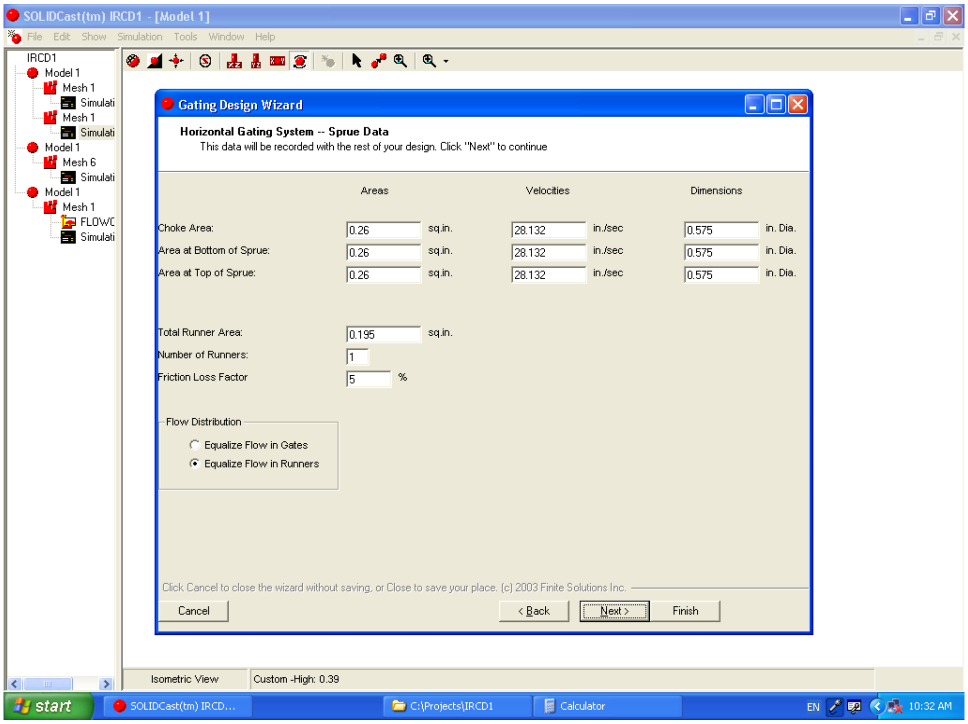
Now that we have the riser and neck dimensioned, we also need to dimension the sprue, runner and gate, as well as estimate a fill time. The SOLIDCast Gating Wizard calculates an Optimal Fill Time of about 13 seconds for this casting (assuming a single casting).



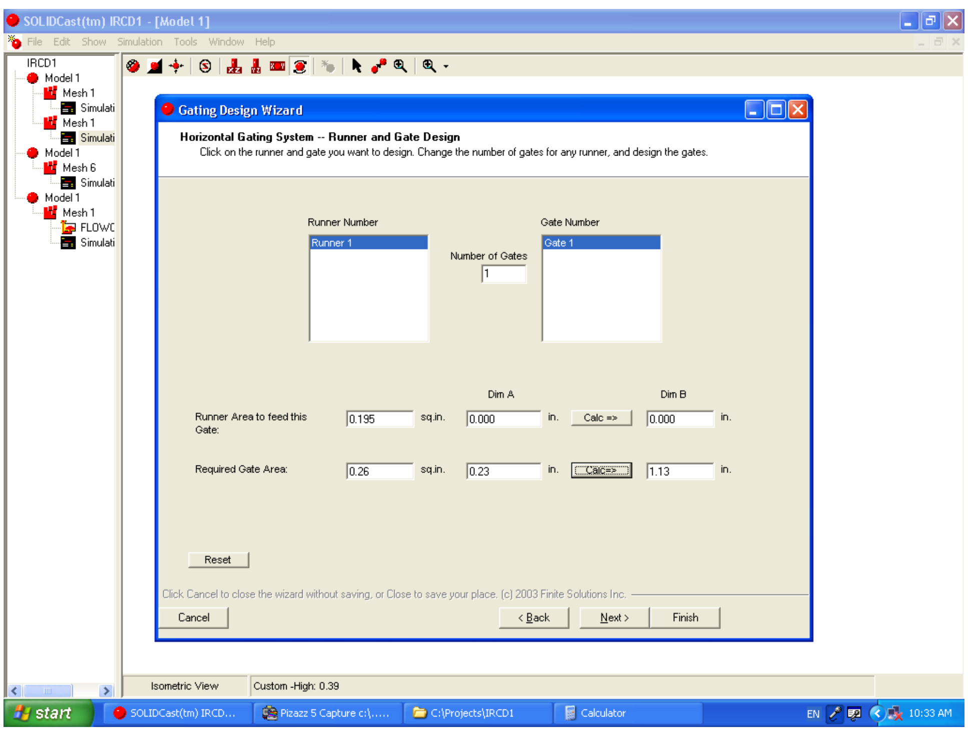
And by describing the geometry of our proposed gating system…



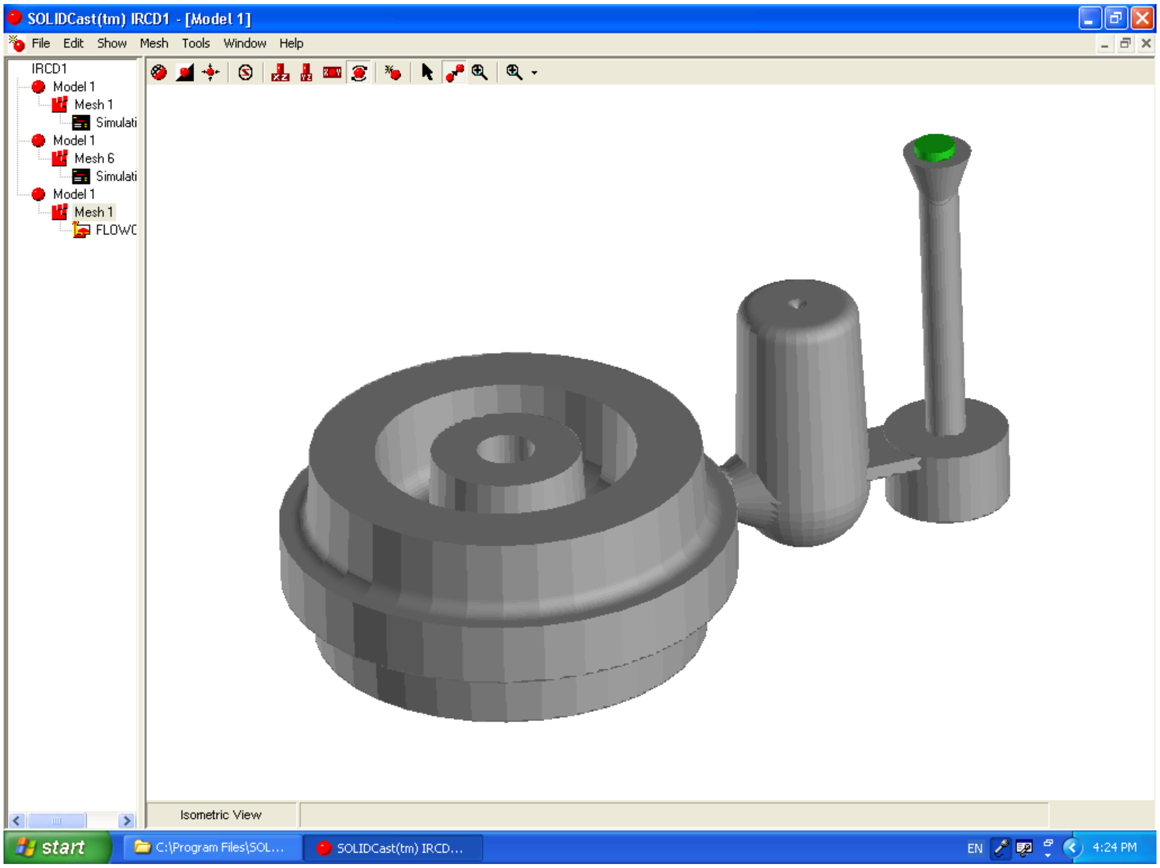
The Gating Design Wizard calculates a sprue diameter of 0.575 inches…



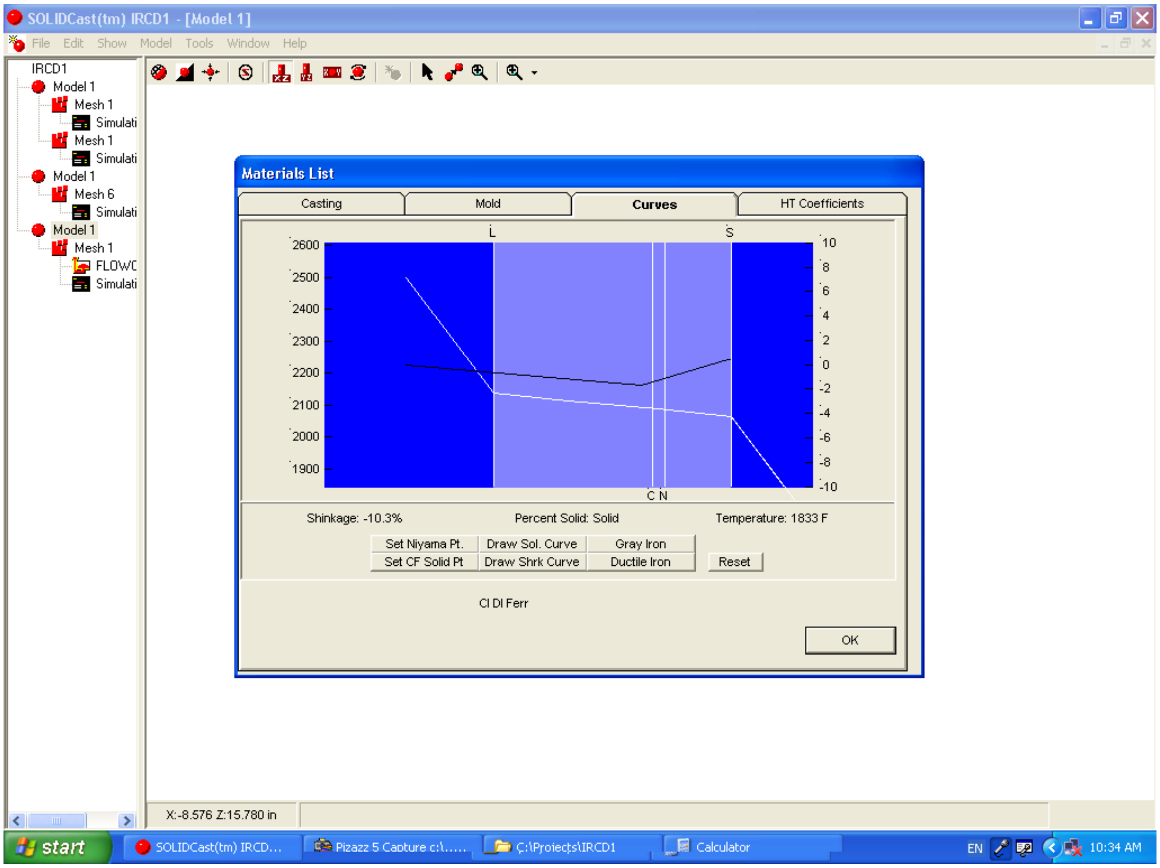
…And inlet gate dimensions of 0.23 in. x 1.13 in., which should insure that the gate freezes quickly for control of expansion pressure.



Now we can use all of the calculated dimensions to create a simple system for gating and feeding this casting, which would appear as follows:



Next we use the results of the Iron Property Calculator to adjust the shrinkage curve parameters for the exact conditions of this iron chemistry, temperature and modulus value. Remember that if you do these calculations on the Iron Calculation Tab of the Materials List, the changes will be made to the curves automatically.



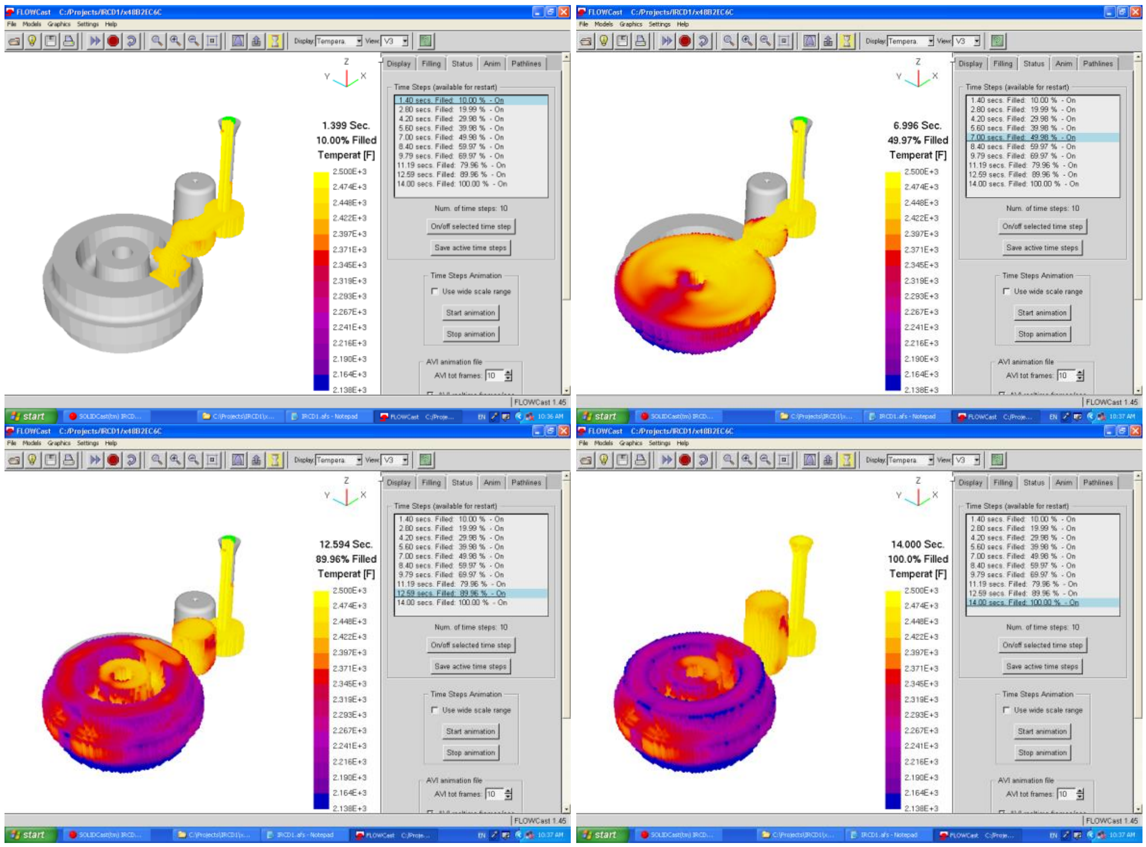
Click on the Draw Shrk Curve button to adjust the black Shrinkage Curve. Shrinkage Time of 63%, amount of shrinkage = -1.1% plus Mold Dilation of 1% for a total shrinkage of -2.1%. Move the cursor until the Shrinkage value reads –2.1% and the Percent Solid is listed as 63%. Click the left mouse button.

Then add expansion of approximately 0.5% per 10% solidification. In this example we would be moving from 63% to 100%, a difference of 37%. At the rate of 0.5% per 10% solidification, this would be an expansion of 1.85%, which, when added to –2.1% equals –0.25%. Shrinkage percent is only listed in 0.1% increments, so move the mouse so that the Shrinkage value reads –0.2% and the Percent Solid is listed as 100%. Click the left mouse button, then click on Done.

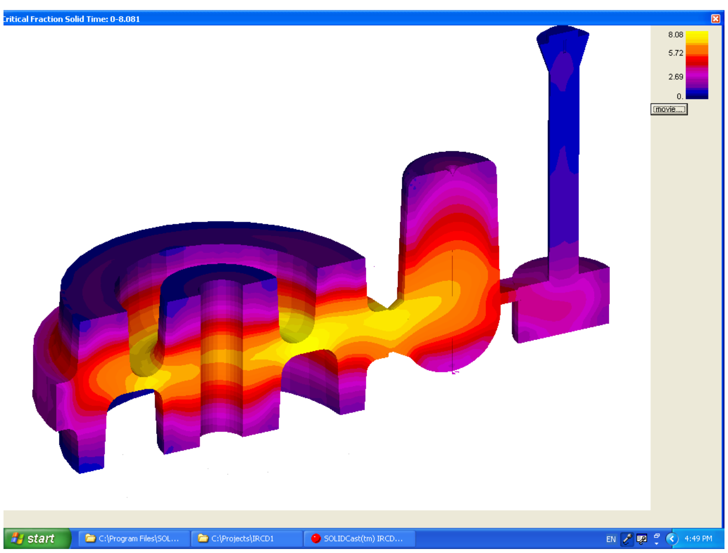
Set the CFS Point approximately 5% to the right of the ST Point. To do this, click on the Set CF Solid Pt button, move the cursor until Percent Solid reads, in this case, 68%. Click the left mouse button. If you use Niyama Calculations, you can set the Niyama point approximately 5% to the right of the CFS Point.

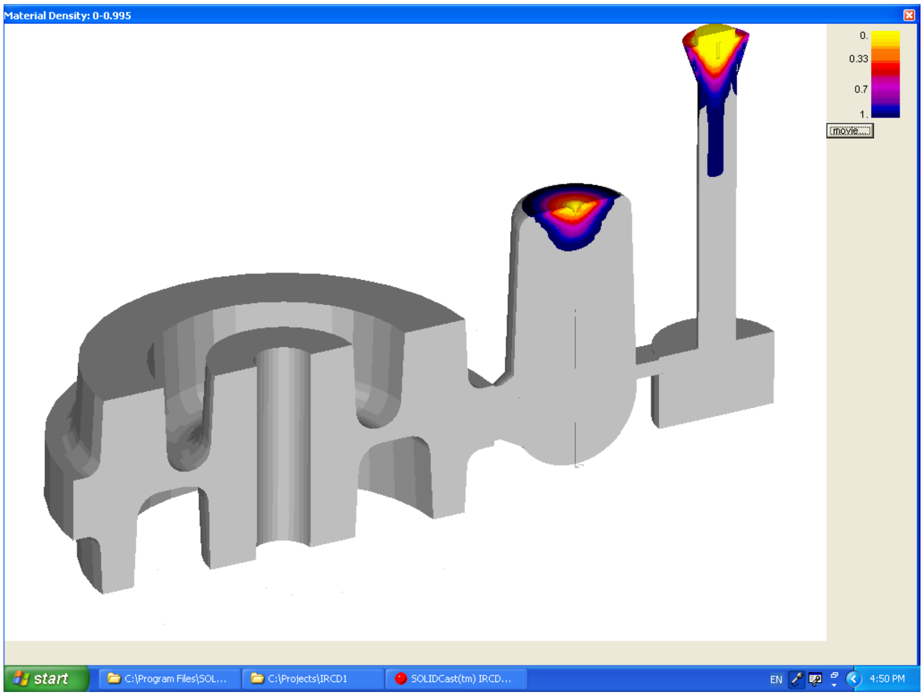
The system is now ready for an accurate simulation of the rigged casting.

Using FLOWCast, we first perform a fluid flow mold filling simulation of the casting, pouring metal down the sprue, through the gate and riser and into the casting cavity. The figures below show various phases of the filling process:

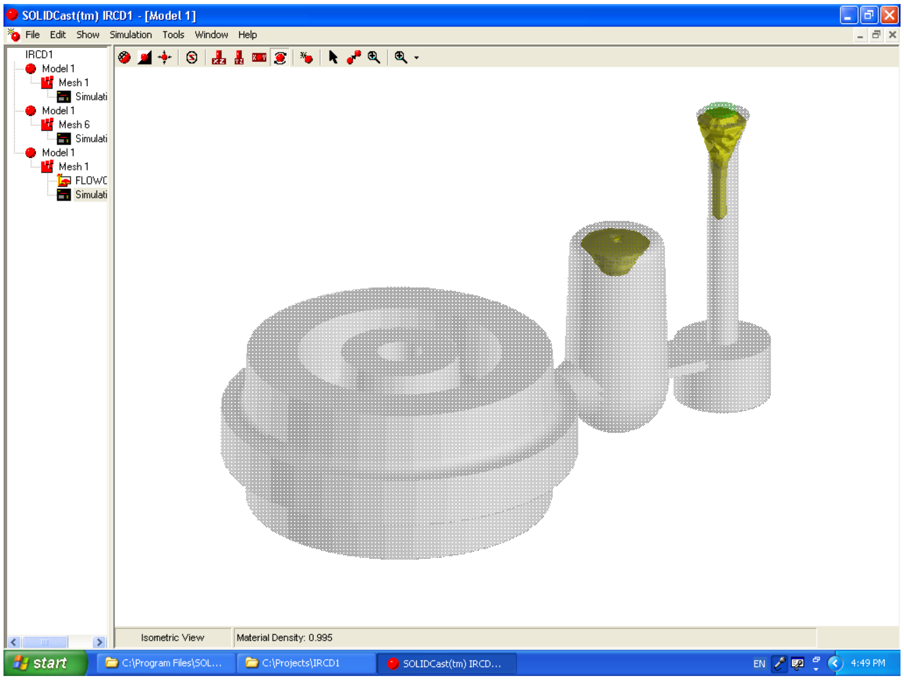


Finally, we use SOLIDCast to perform a simulation of the solidification of the casting and to predict the soundness of the final part. Progressive Solidification is shown here:





Shrinkage prediction using CastPic plotting, sliced through the center of the gating system.

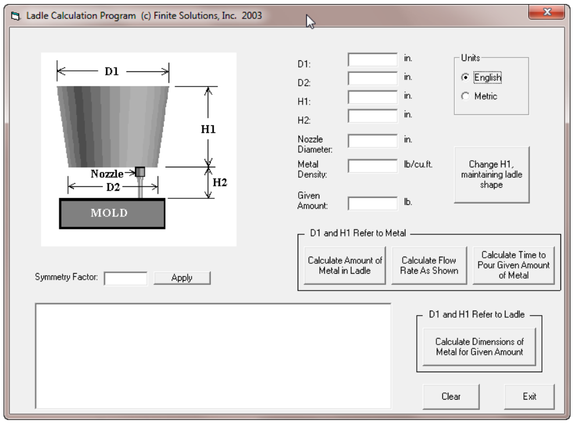


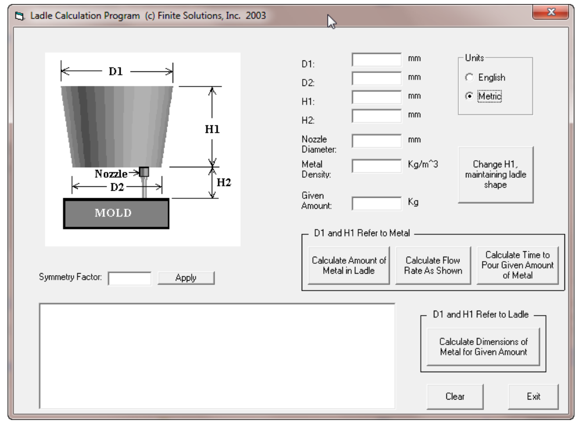
Shrinkage prediction using Iso-Surface plotting, an X-Ray view.

The final result is a sound casting, effectively designed using the SOLIDCast Riser and Gating Design tools, and verified using SOLIDCast and FLOWCast simulation.

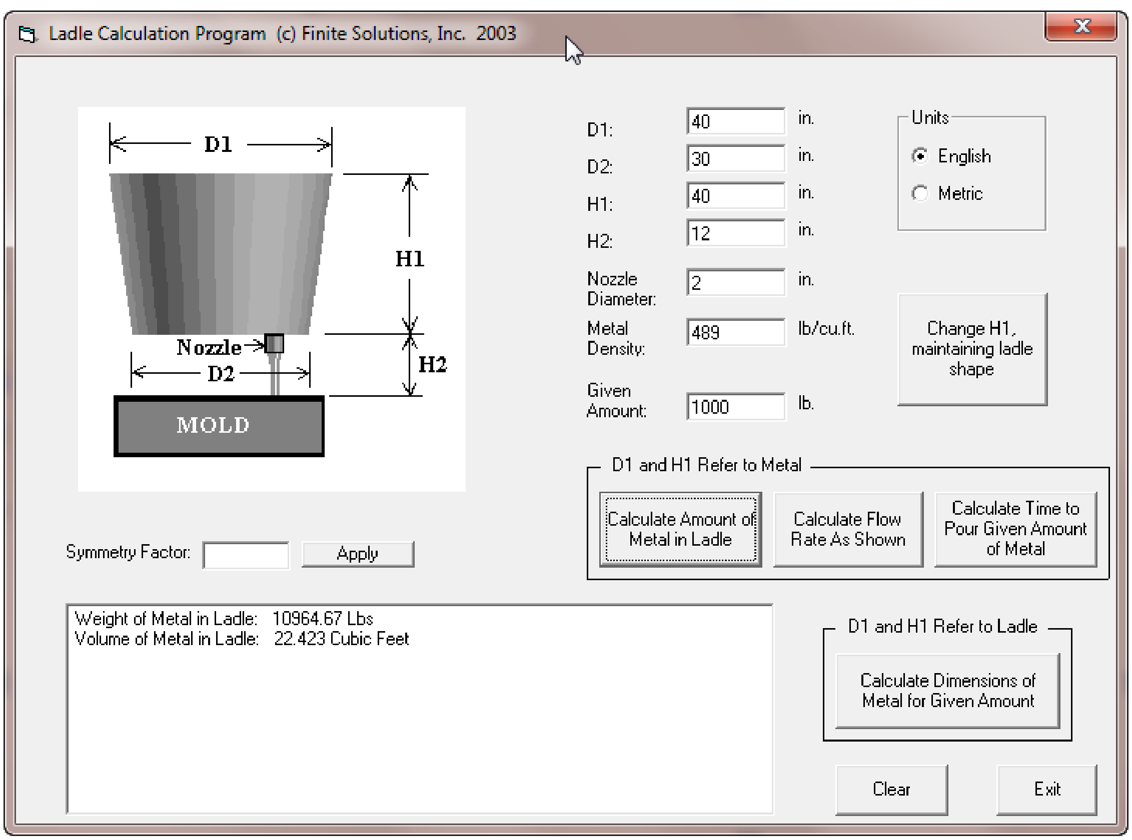
UTILITY: Ladle Calculations

The Ladle Calculations utility allows you to calculate volumes, flow rates and fill times from a bottom pour ladle. The utility can be run in either English or Metric units, as shown here:

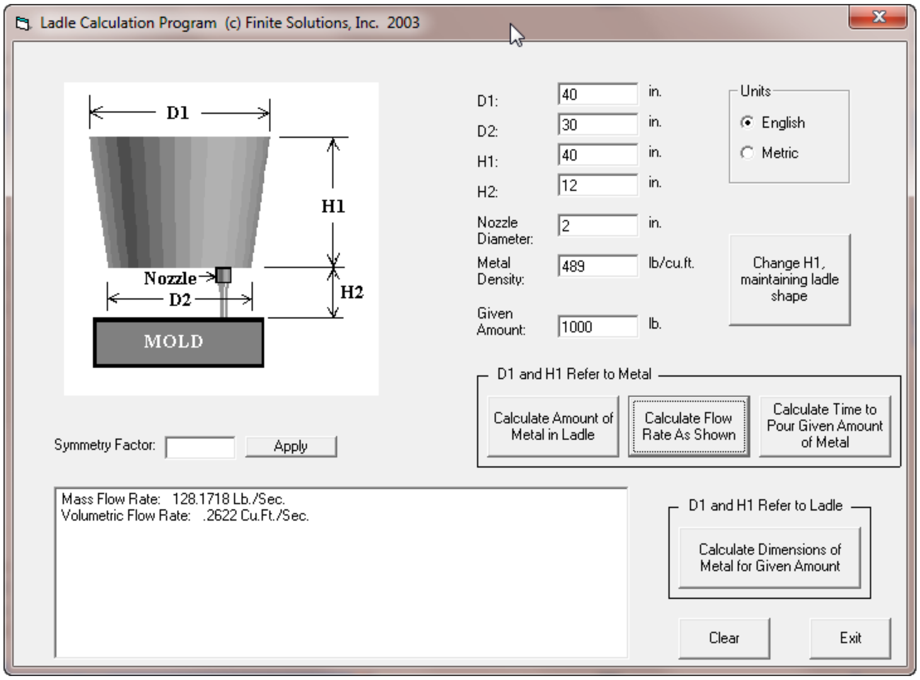




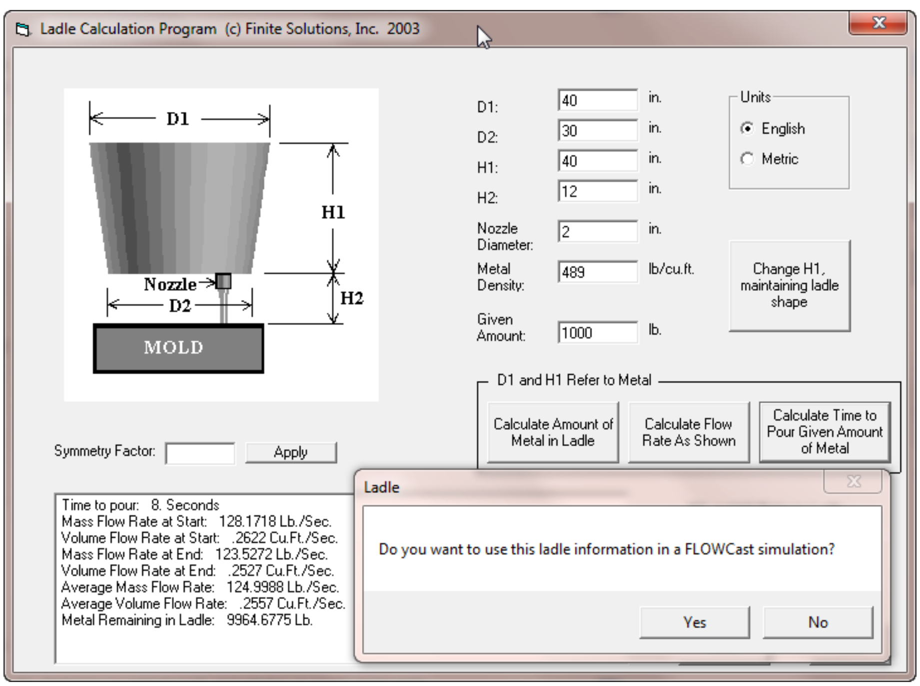
To run Ladle Calculations, first load a model, then go to Tools…Ladle Calculations.To use this utility, enter the dimensions of the metal in the ladle as shown, plus the density of the metal. The first option is to calculate the weight and volume of metal in the ladle, as shown here:



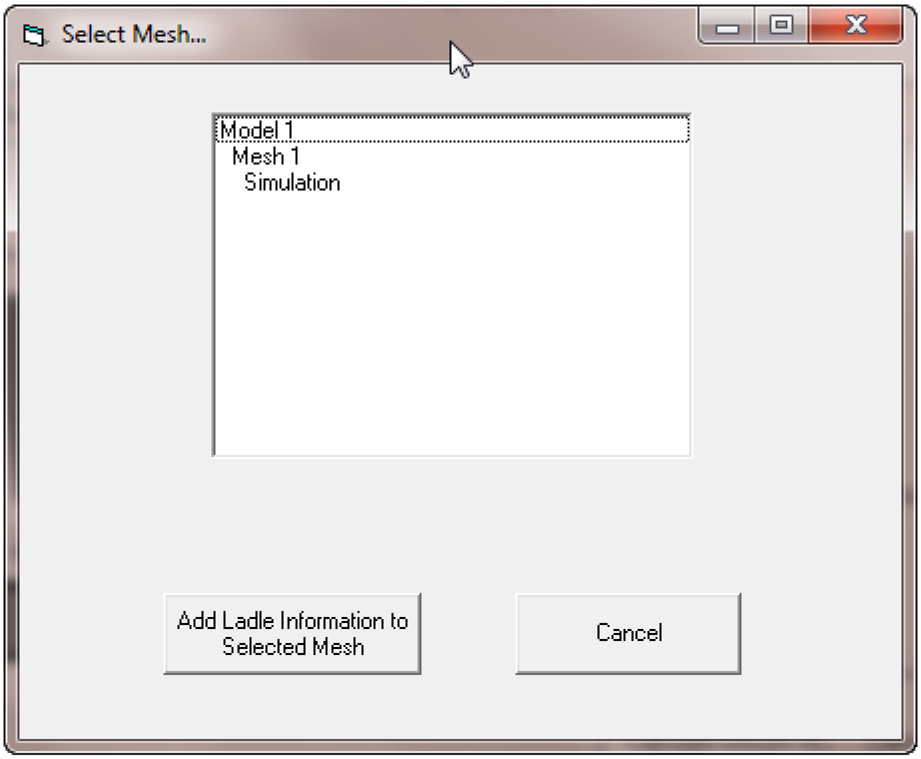
The second option is to calculate and show what would be the current rate of flow from the nozzle under the conditions shown, as seen here:



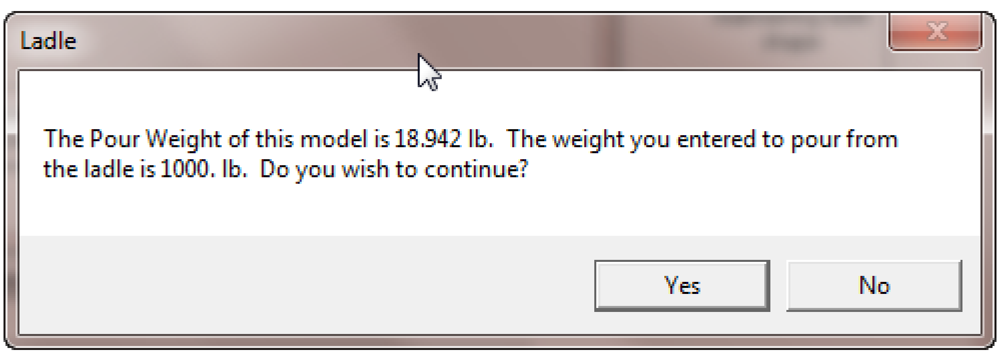
The third option allows you to enter an amount of metal that you want to pour from this ladle. After this entry, there is a button to click which calculates the time to empty this much metal from the ladle, the flow rates at start and finish of the pour, and the average flow rate. An example of this calculation is shown here:



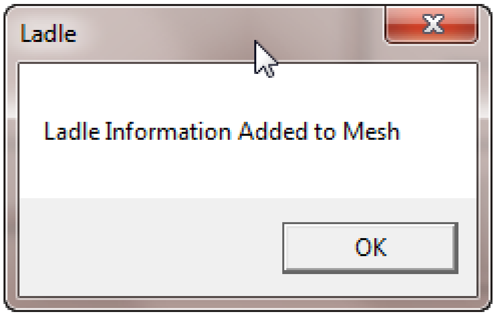
Once you have calculated the time it takes to pour a given amount of metal, the system will ask whether you want to save the variable flow rate data to a file for use in a fluid flow simulation using FLOWCast, as shown in the previous figure. If you choose yes the system will display the Project Tree, as shown here:



Use the cursor to highlight the desired mesh, then click on Add Ladle Information to Selected Mesh. If there is a problem, for example, you select the wrong mesh, then you may get an error message like this:

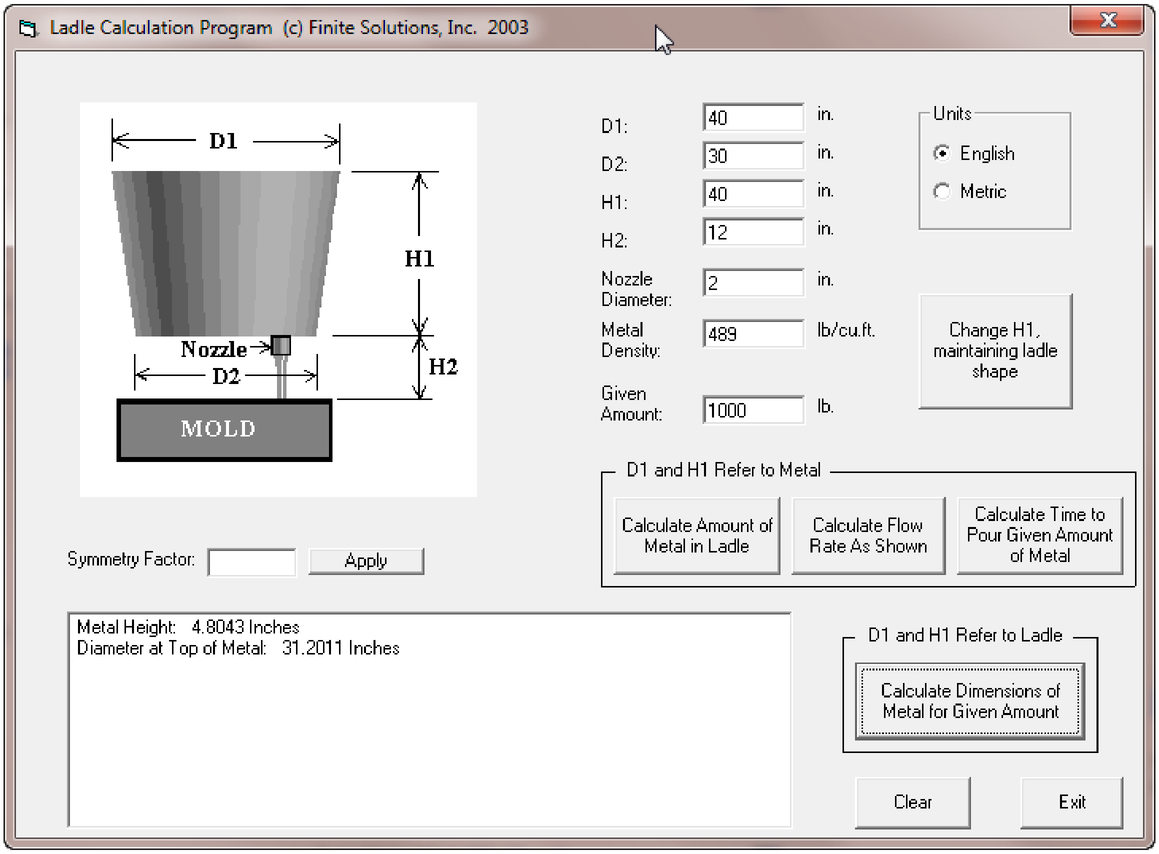


If you select Yes, or if the system did not encounter a problem, then the data will be added to the mesh folder and the system will display:



When you enter FLOWCast, go to the Filling Tab and check the box labeled Variable Flow Rate. FLOWCast will then use the data generated using Ladle Calculations.

The final option allows you to calculate the 'shape', or dimensions, of a given amount of metal poured from the ladle. In the example shown below, you would lose almost 5 inches of metal height in the ladle during a pour of 1000 lbs.



Use of the Symmetry Factor

Note that in the middle of the Ladle Calculation screen there is a box labeled Symmetry Factor and a button labeled Apply. The purpose of the Symmetry Factor is to correct the ladle calculations for the case where the model is divided by one or two planes of symmetry.

For example, if the model is symmetrical about the X axis and a Lower X or Upper X Plane of Symmetry is applied, then the volume of metal in the mesh is one-half the complete volume of the model. This means that it is necessary to adjust the dimensions and volume of metal in the ladle, i.e., to come up with an “equivalent ladle” which contains one half the volume of metal but pours the given amount of metal in the same time. The Symmetry Factor should be 0.5 if you are dividing the model in half with a central plane of symmetry and 0.25 if you have two planes of symmetry and are simulating one quarter of the model. Once you enter this and click the Apply button, the ladle dimensions are adjusted to make the flow rates come out to an “equivalent ladle” for ½ or ¼ the amount of metal.

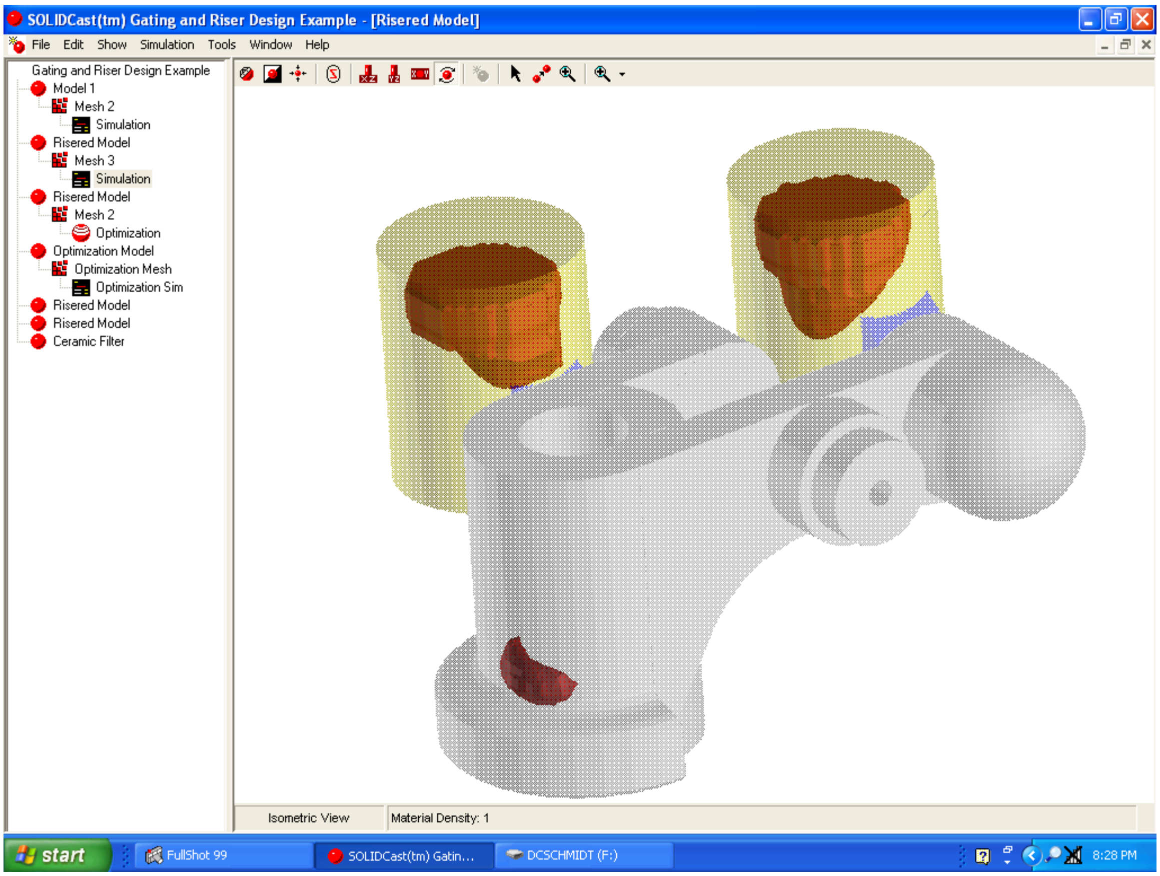
For example, if you have one plane of symmetry and the model is cut in half by the plane of symmetry, enter 0.5 and click the Apply button. You will see the ladle dimensions change to represent an “equivalent ladle” which will deliver one half the amount of metal in the same time.

An important note: Do not press Apply again or the ladle size will be cut in half again.

UTILITY: Mask Riser Density

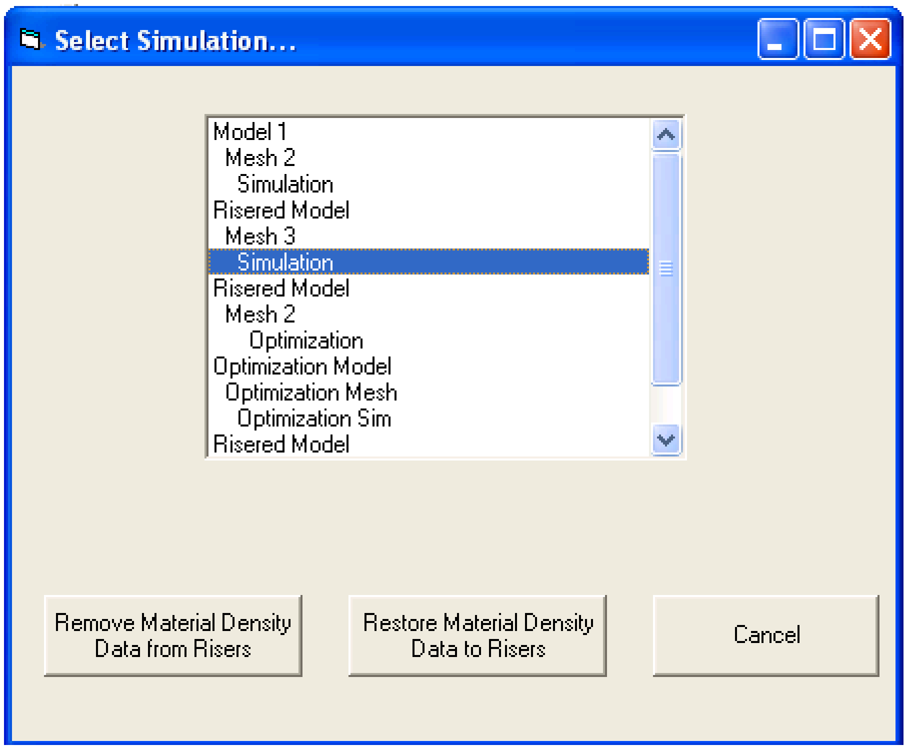
This utility is used to hide the Material Density Factor data contained in parts of the model made of Riser Material. When casting and rigging geometry gets complicated, this can be quite useful to see where the shrinkage indications are in the casting only.

As an example let's look at the following Iso-Surface plot of the Material Density Factor(MDF):



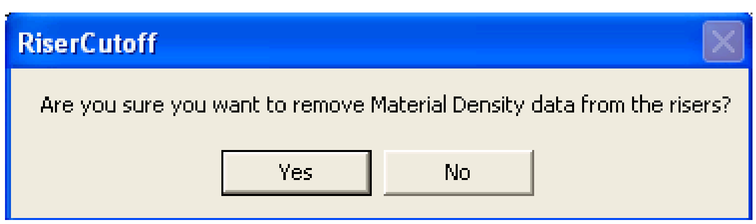
This plot shows shrinkage indications both in the casting and in the risers. In order to focus on the shrinkage in the casting, use the Mask Riser Density utility.

To use this utility, load a model, then select Tools…Mask Riser Density. You should see a listing of the Project Tree, as shown on the next page.

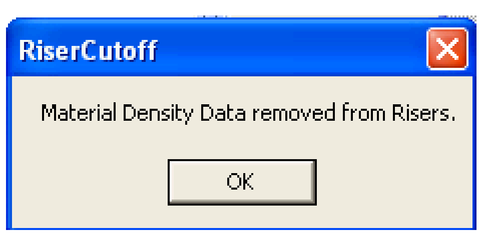


Use the cursor to highlight the name of the simulation that you want to mask MDF information in the risers, and click on the Remove Material Density Data from Risers button. Note that there is also a button to Restore Material Density Data to Risers. The system does not delete the data, just masks it for plotting purposes.

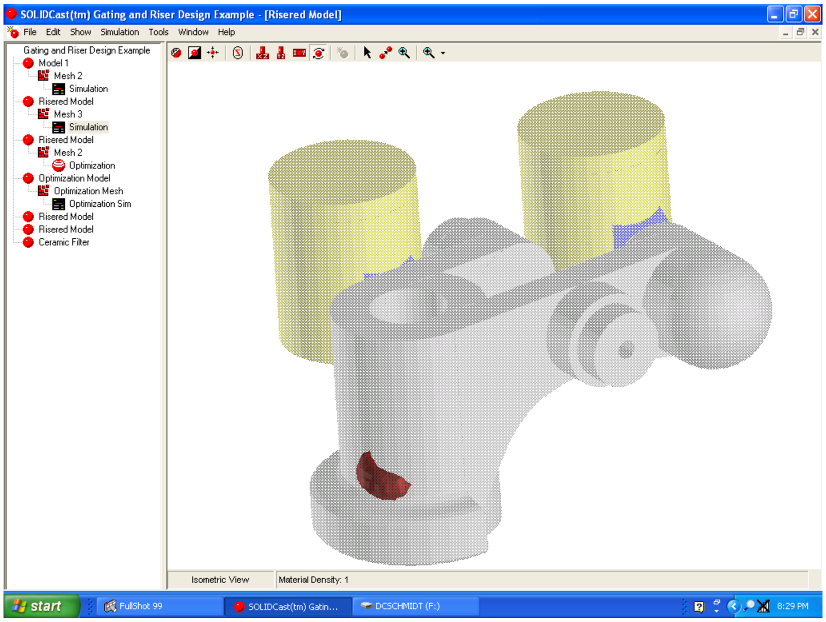
The system will ask for confirmation prior to running the operation, as shown here:



Confirm by clicking Yes and the system will respond with:

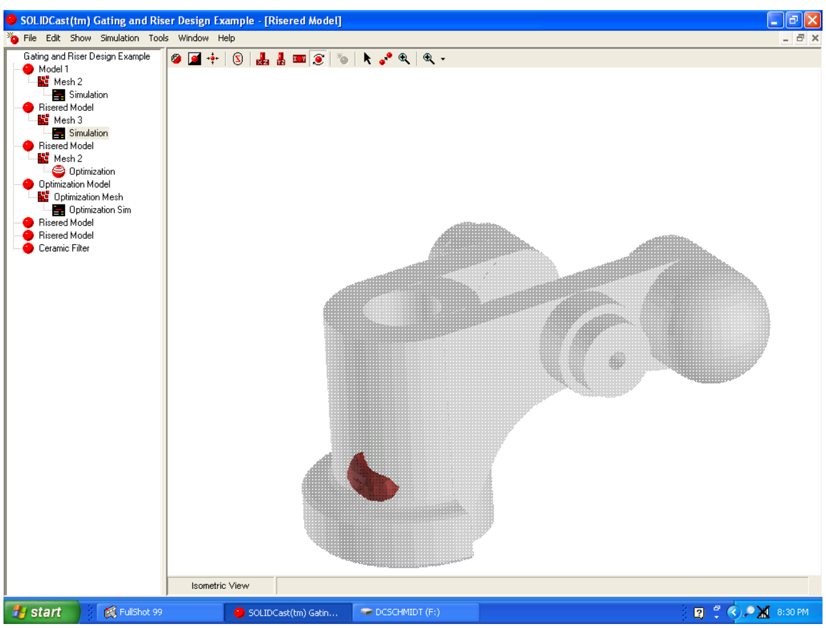


If we now do the same iso-surface plot as before we masked the riser data, we get a display that looks like this:



It is now very easy to see the shrinkage in the casting, and not be bothered with shrinkage in the risers.

You can also use the Select icon to select the risers, then hide the risers using the Show Menu. The following figure shows the plot with both data masking and the risers hidden:



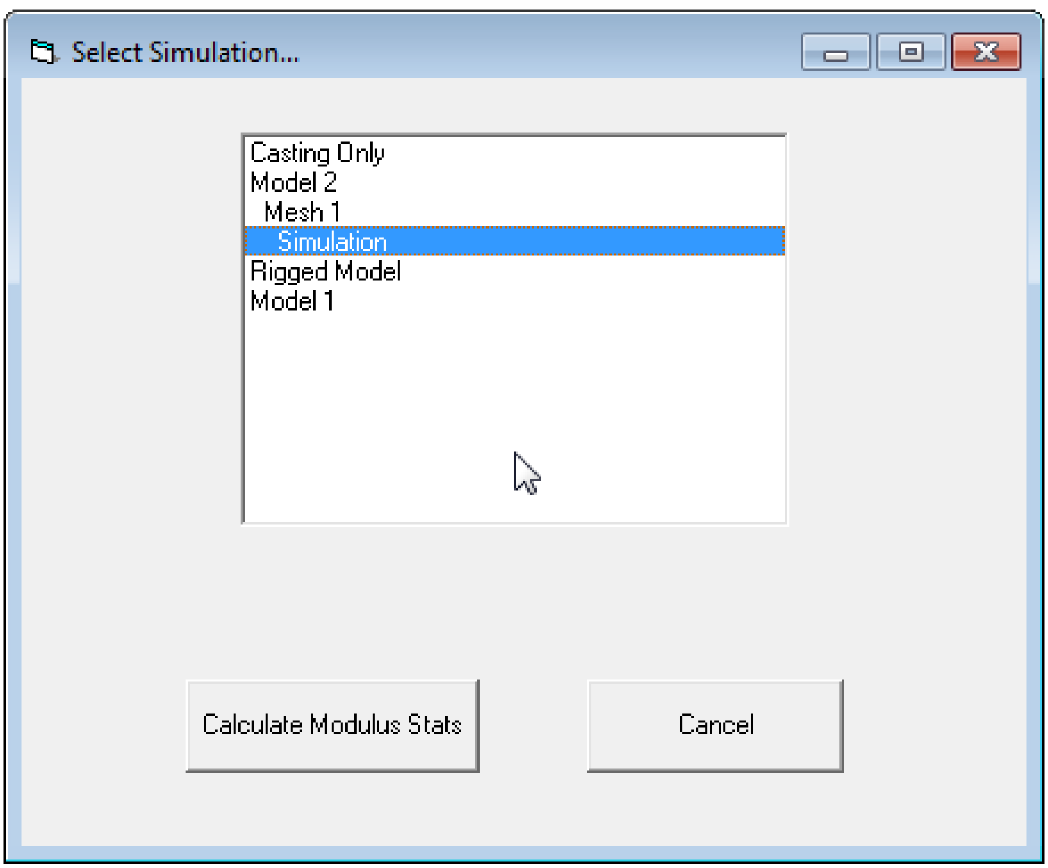
Remember that if you want to see the riser data in the future, just run Mask Riser Density… again and select Restore Material Density Data to Risers.

UTILITY: Modulus Statistics

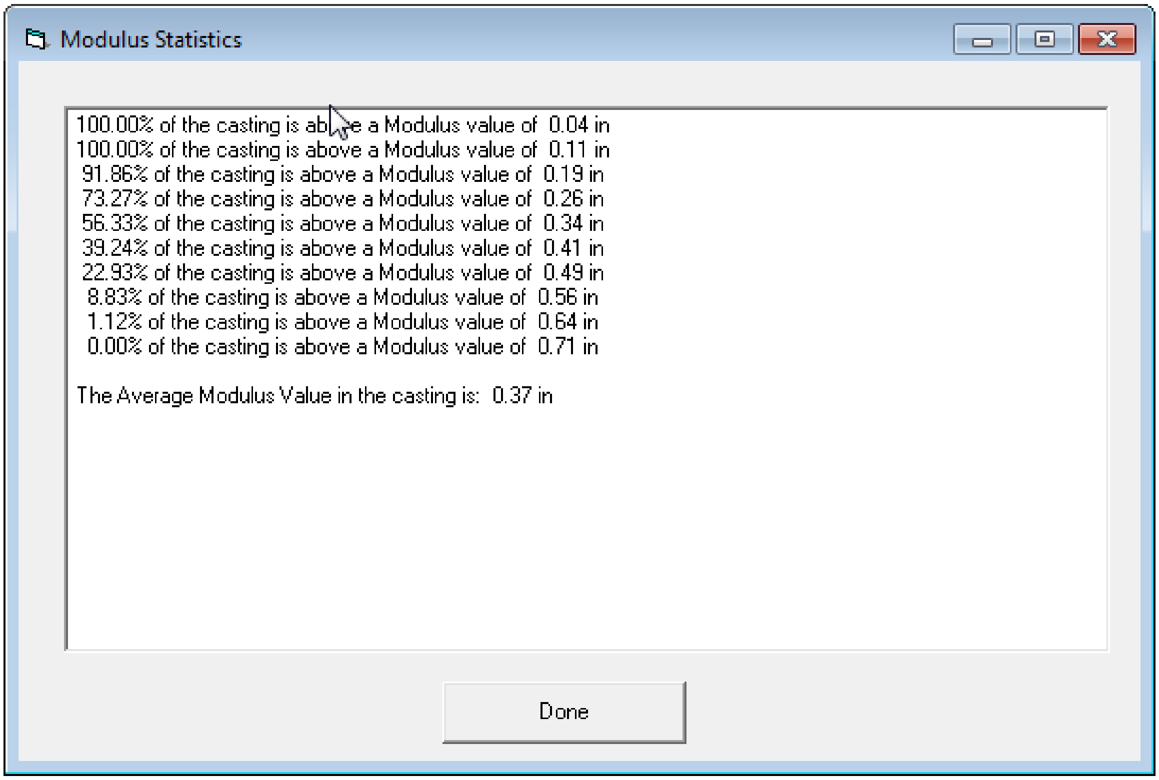
The Modulus Statistics tool is primarily intended for use with heavy-section iron castings. Typically we recommend that a "naked casting" simulation be run as the first step in designing a feeding system for iron castings. Using the Iron Property Calculation and the Riser Design function for iron casting, if the maximum Modulus is high enough then the Riser Design program may indicate that there is "Net Expansion" and no riser is required. This typically happens when the maximum Modulus is in excess of 1 inch (2.54 cm).

Since a given casting may have both heavy and lighter sections, it is important to find the overall "average" Modulus to determine whether there will actually be enough expansion to produce a casting without risers. We therefore recommend that, if the Riser Design function indicates "Net Expansion" after entering the maximum Modulus value, the user obtain the average Modulus value for the casting and then re-run the Iron Property Calculation and the Riser Design function with the average Modulus Value. The Modulus Statistics tool can be used to display the average Modulus value for any casting. It also shows, as a matter of interest, the distribution of Modulus values within the casting.

To start the utility, load a model then go to Tools…Modulus Statistics. You will see a window of the current project tree:



Highlight the simulation you want to work with, then click the button that says Calculate Modulus Stats. The system will run through the modulus distribution and will then display the average modulus and the overall distribution of modulus values, as shown here:

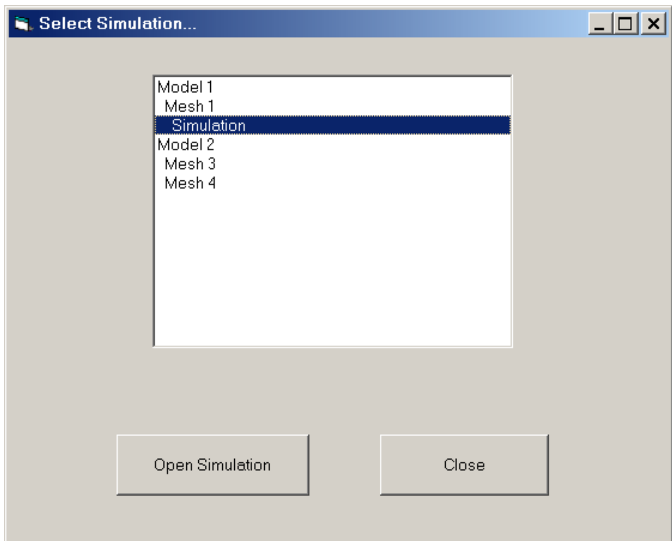


After using the average Modulus value, the Riser Design function may or may not show “Net Expansion”. If a riser is required, then the user can calculate a riser size in the normal way. If no riser is required when using the average Modulus, then riserless casting can be considered. It is necessary to ensure that mold rigidity and iron quality are sufficient for riserless casting; also, it is recommended in some cases that “safety risers” be considered even when a casting is produced in a riserless process.

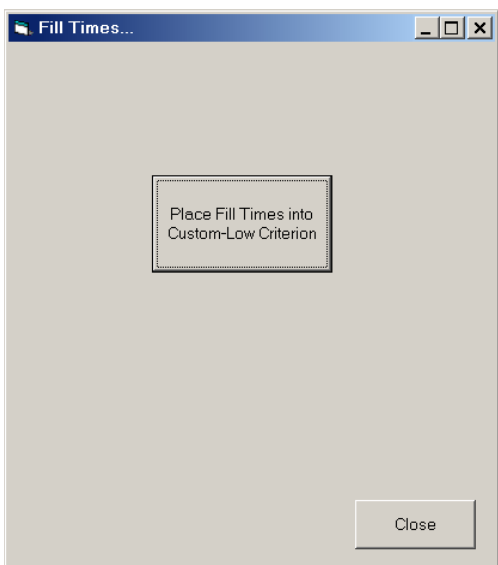
UTILITY: Plot SOLIDCast Fill Times

This utility places progressive fill times from a simulation into the Custom function so that they can be plotted. This assumes that Fill Material has been included in a model and a mold filling simulation has been performed using the SOLIDCast Fill Algorithm. If you have run a fluid flow simulation using FLOWCast, there are a number of movies and plots that can be done within FLOWCast.

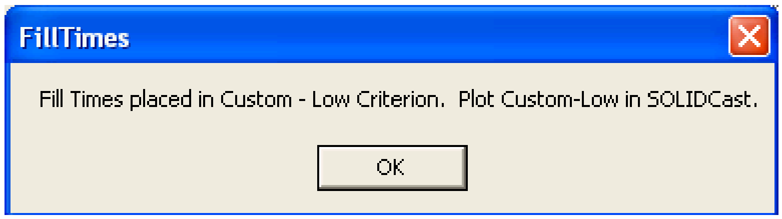
To use this utility, load a model, then select Tools…Plot SOLIDCast Fill Times. You will see the Project Tree displayed. Select the Simulation entry on this Project Tree for which you wish to view and plot Fill Times:



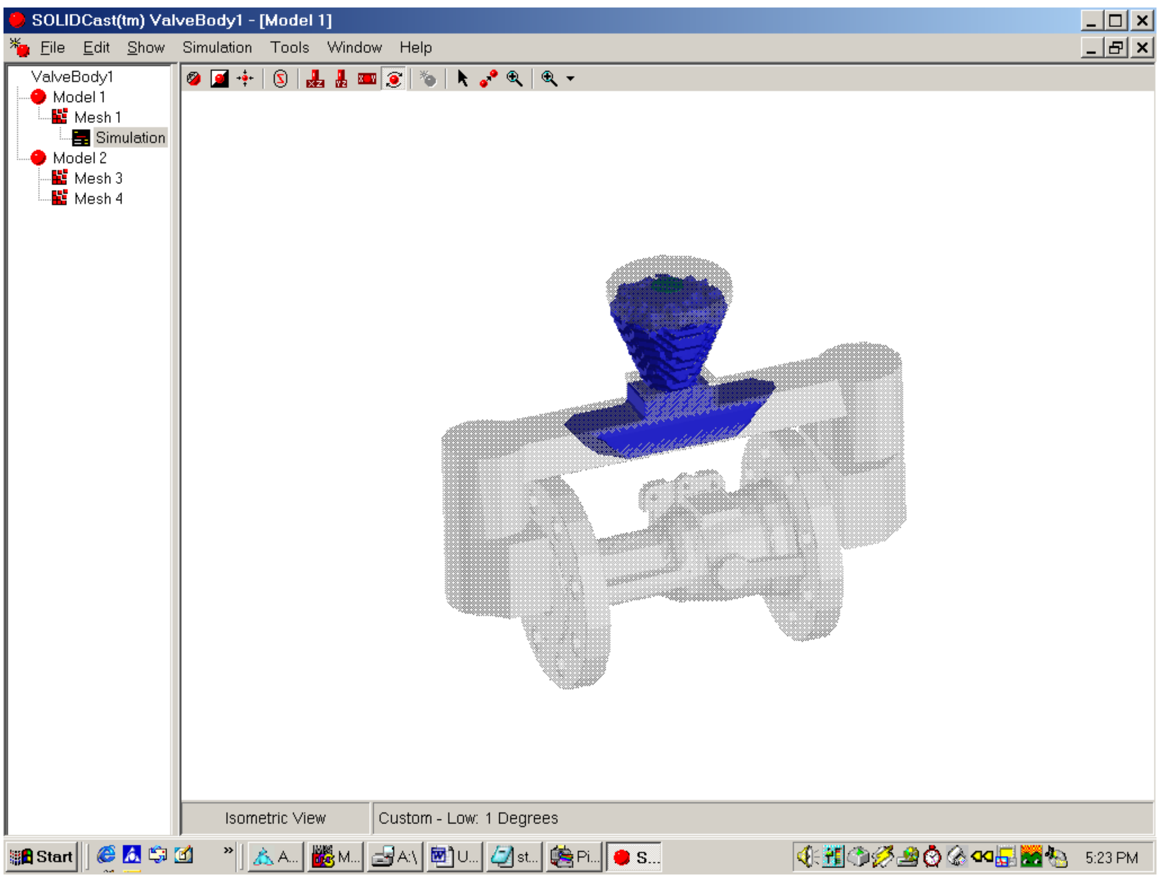
Now click the Open Simulation button. The following screen will appear:



Click the button in the center of the screen labeled Place Fill Times into Custom-Low Criterion. This will put the data into the Custom area so that it can be plotted in SOLIDCast using the Custom-Low function. When the process is complete you will see the following:



For example, the following shows an Iso-Surface plotted at 1 second after the start of fill, using Custom-Low:

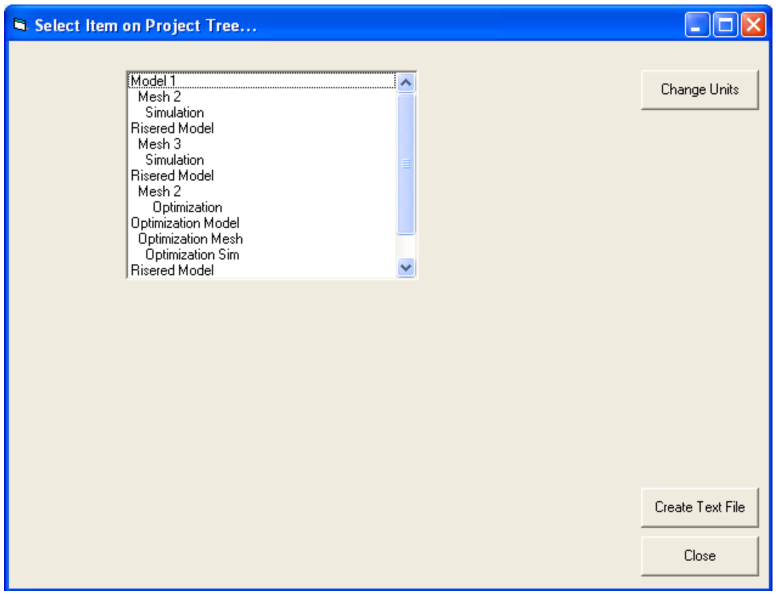


Note that the progressive temperatures during a fill are not saved. With this function you are plotting only FILL PROGRESSION and not temperature distribution. It is possible to use the movie option on any of the plots to create a filling animation sequence.

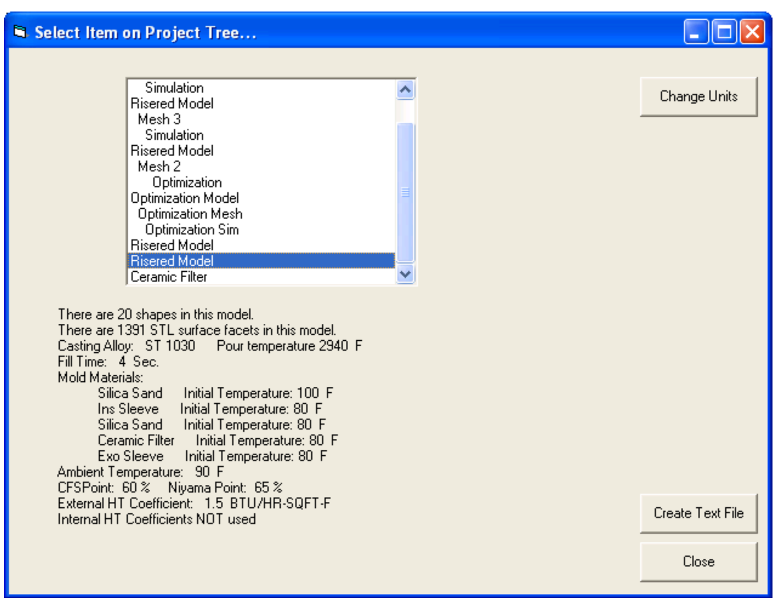
Also note that since the data is placed in the custom function, it will remain there until it is overwritten by another custom function, such as calculation of FCC Criterion, modulus or any other user-defined custom function.

UTILITY: Project Viewer

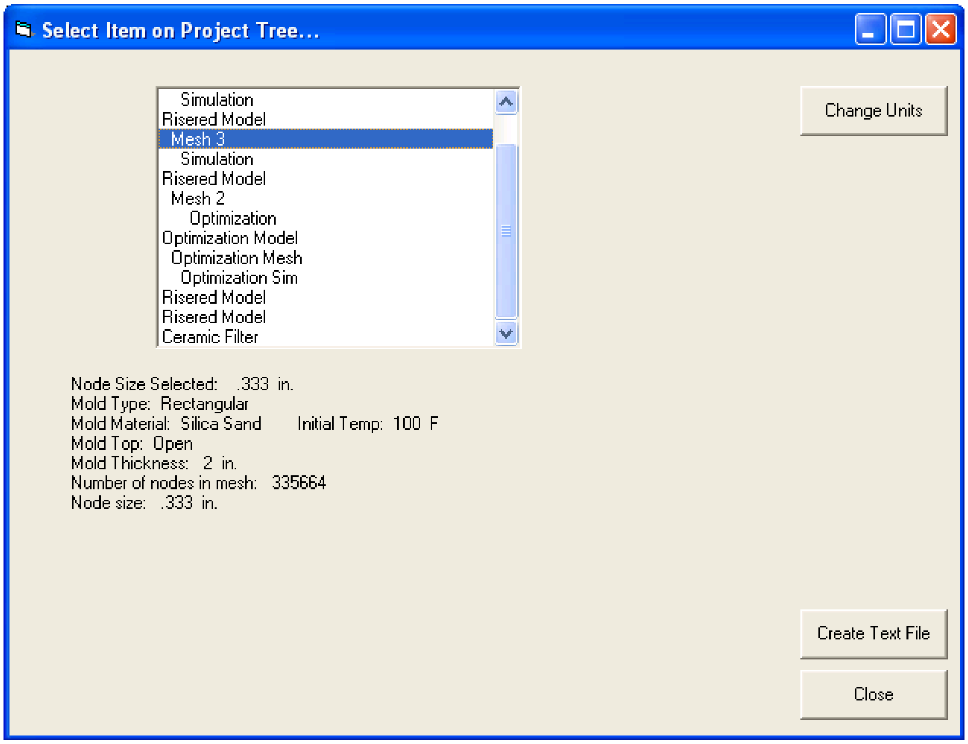
ProjectViewer is a handy tool to look back over models, meshes and simulations, to get a summary of their components. To start Project Viewer, open a model, then select Tools…Project Viewer. It will show a screen like this:



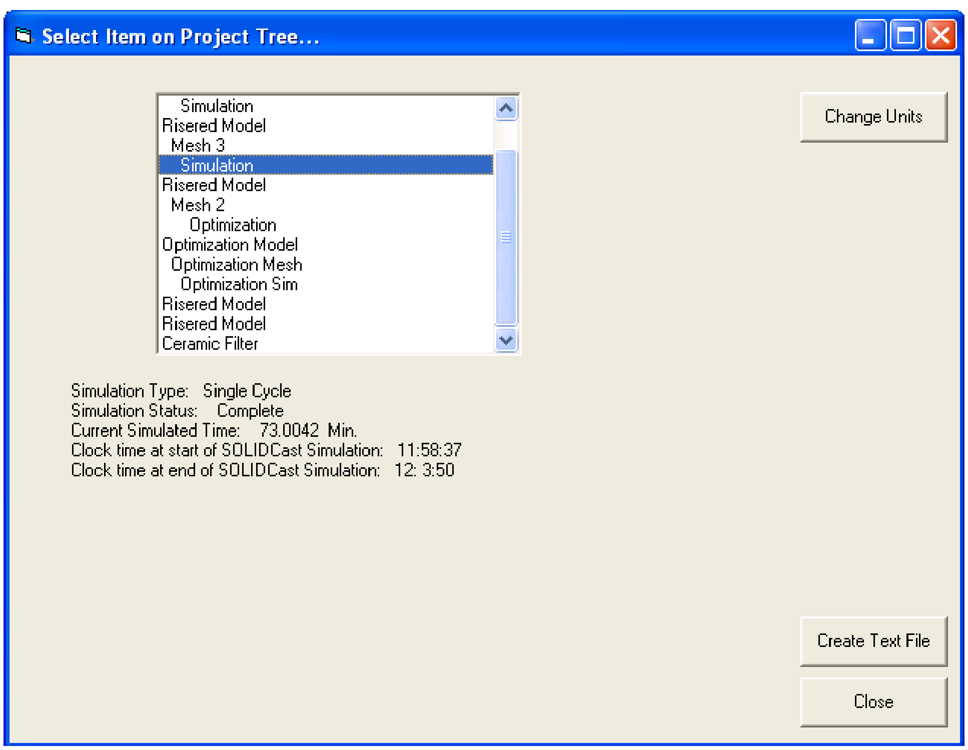
The project tree is displayed, and you can highlight a mesh, model or simulation, and the system will display a summary of that item. Sample summaries of a model, mesh and simulation are shown in the following figures:



Sample Model Summary



Sample Mesh Summary



Sample Simulation Summary

If you click on the box labeled Create Text File, the system will build a text file that contains summary information for each entry in the selected project tree. Once created, the system will display the following:

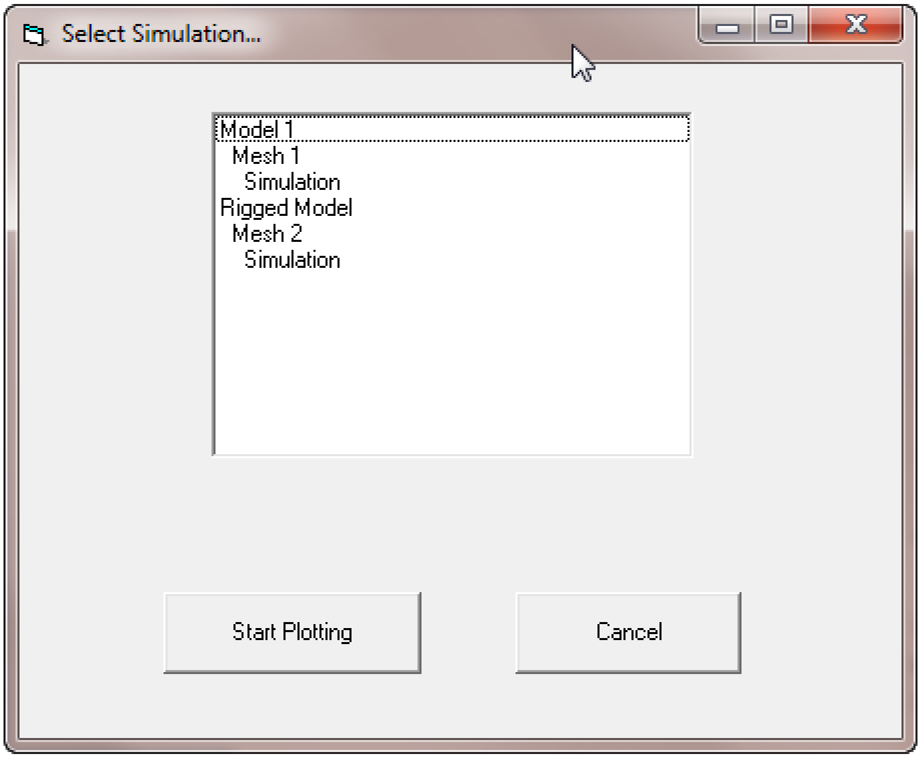
At present, the summary displays cover models, meshes and simulations only. Optimization data and Fluid Flow Simulation data will be added in a future release.

UTILITY: QuickPlot

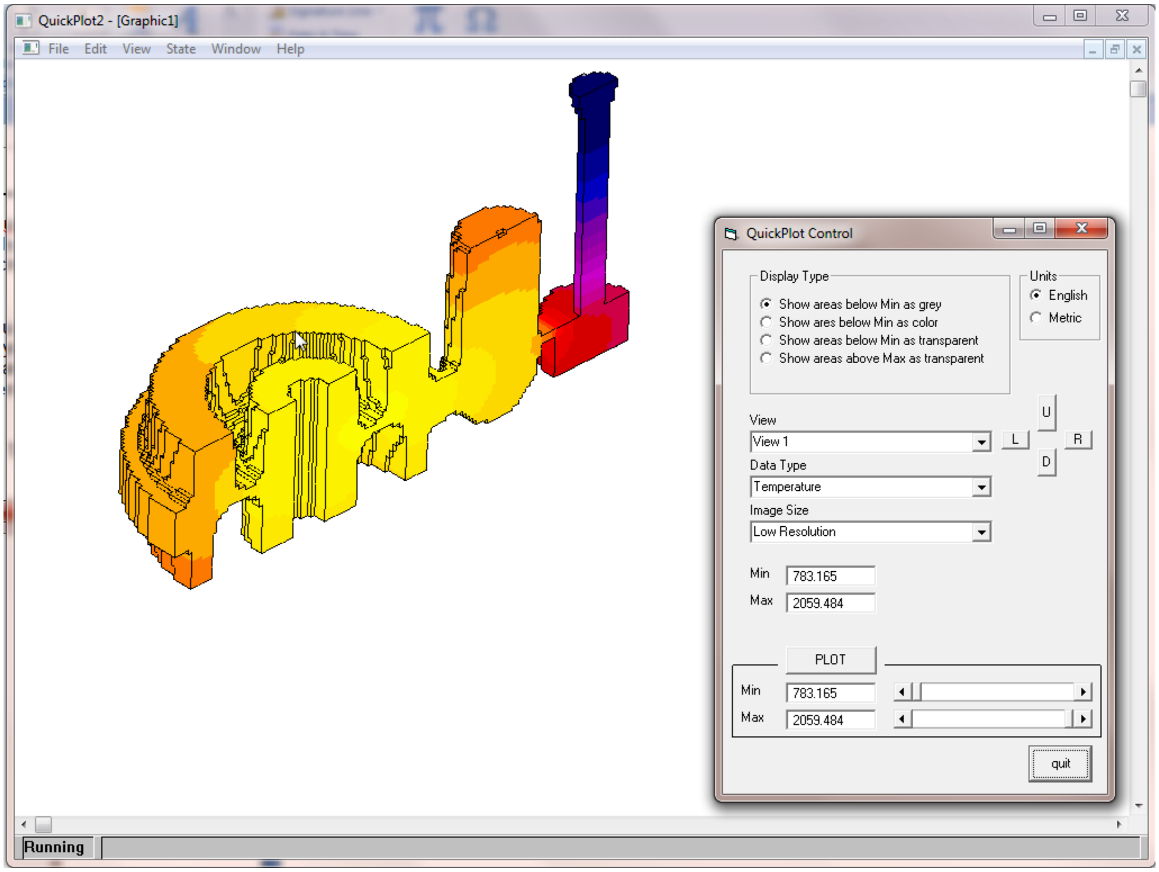
This utility is patterned after the Simulation Image Control utility. QuickPlot allows you to change resolution, plotting angle and data types very quickly. By clicking on slider bars, new plots are made instantaneously, and you can create live animations by holding down the slider bar.

QuickPlot greatly simplifies and speeds up multiple plots, since you don't have to select the plot type, data type and plot value prior to each plot. All you do is click the slider bar once and the new plot is displayed. Precise viewing angles are also included, making it easy to compare simulation results from different models by placing them in exactly the same orientation.

To use QuickPlot, first load a model, then go to Tools…QuickPlot. The system will bring up a copy of the Project Tree, something like this:



Highlight the simulation you want to plot from using the mouse, then click the Start Plotting button. The system will load a view of the model and display the QuickPlot Control. The default display is temperature. An example of the initial display is shown on the next page.



The default view is View 1, of Temperature, in Low Resolution. The Display Type is set to show areas below the minimum value as grey.

The PLOT box at the bottom of the QuickPlot Control lets you plot and re-plot almost instantaneously. For example, if you click the right arrow slider bar for the Minimum value, the system will re-plot with a new bottom value. This is a similar technique used in the various movie making functions in SOLIDCast. So, if you click and hold the mouse button down, the system will re-plot as quickly as it can, giving you an animation of the data. If you get to the end of the plot range, you can click on the left arrow of the slider bar to see the animation in reverse.

The various functions are as follows:

Display Type – This controls what happens to areas that fall outside the plot limits. As data falls below the minimum, it can turn grey, it can turn to a color(dark blue) or it can become transparent. If data is above the maximum, it will either be color(bright yellow) or transparent.

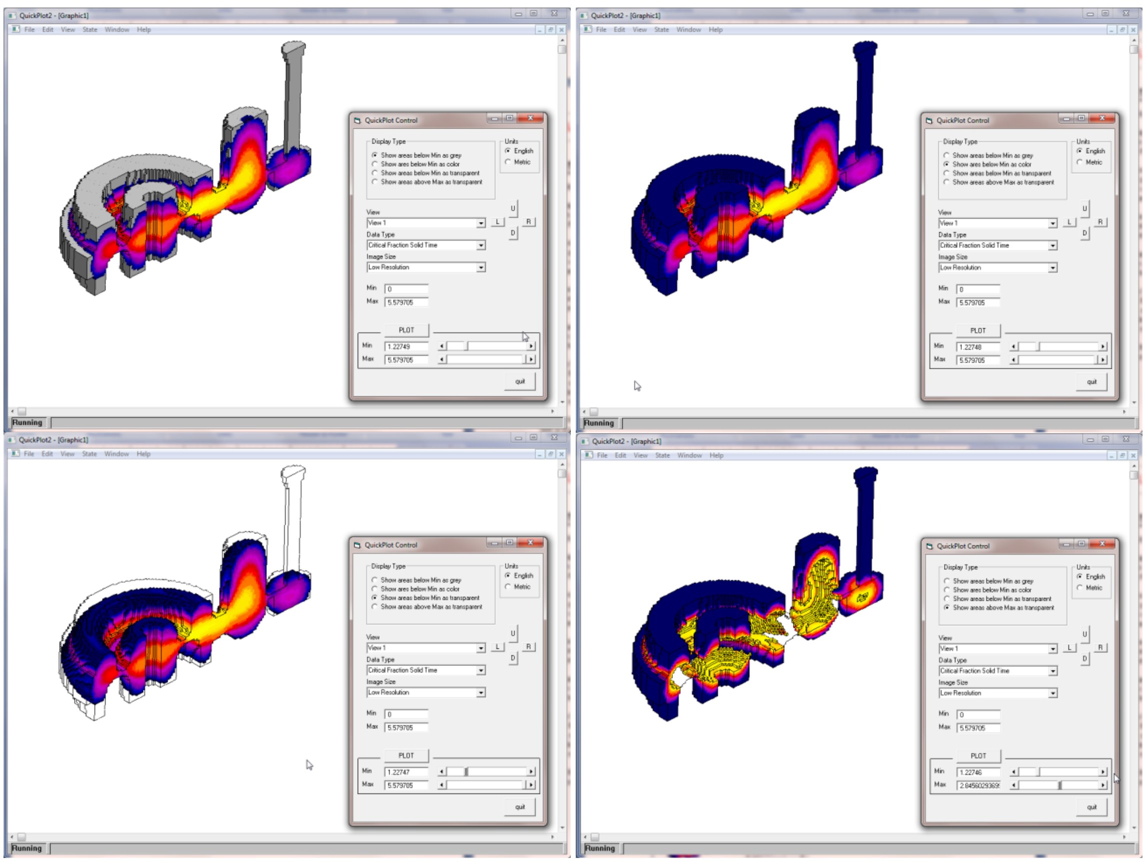
View – There are 7 pre-defined views; Angle Views 1-4, plus the XZ, YZ and XY Orthogonal Views. There are also 4 buttons; L, U, R and D, which, when you click on them will rotate the existing view by 5 degrees to the Left, Up, Right or Down, respectively. These tools allow you to view the part from precise, reproducable angles, so that you can make direct comparisons of various models by placing them in exactly the same orientation.

Data Type – You can plot anything that SOLIDCast calculates, including Temperature, Solidification Time, Critical Fraction Solid Time, Temperature Gradient, Cooling Rate, Material Density, Liquidus Time, Niyama Criterion, Hot Spots and Custom Functions.

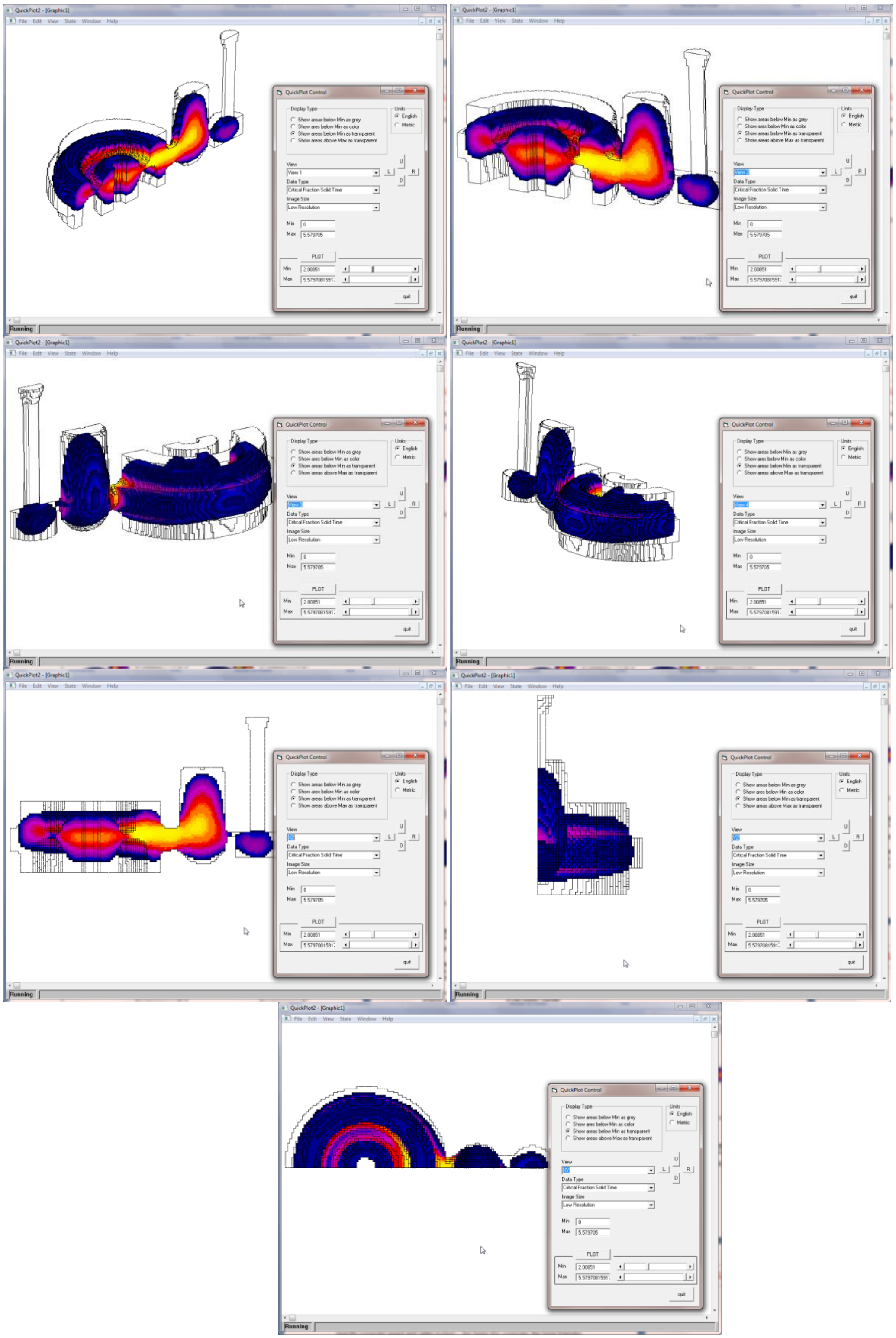
Image Size – The plots can be viewed in Low, Medium and High Resolutions.

The screen captures on the following pages have included the QuickPlot Control, so you can clearly see the effects of different selections.

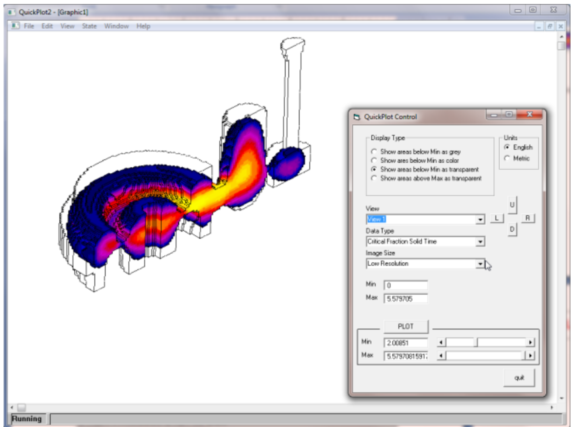
The screens below show the differences in Display Type:

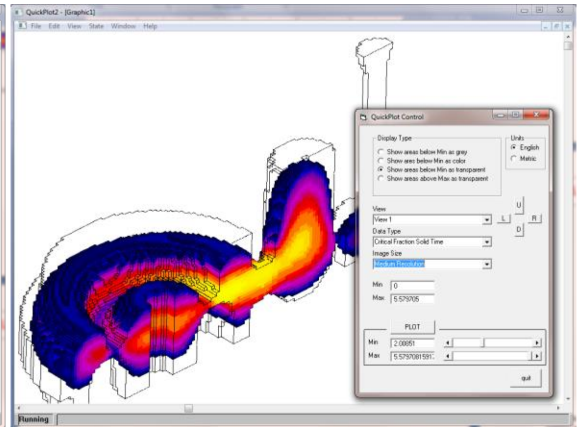


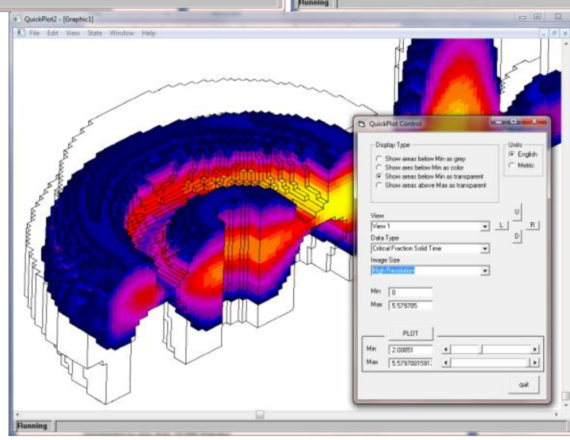
The screens below show the differences in model views:



These screen captures show the differences between Low, Medium and High Resolutions.







UTILITY: Reduce STL File Size

This utility is used to reduce the number of triangles in a binary STL file.

STL files are actually a “skin” made of small triangular surface facets. The number of triangles in a shape can be governed by the output settings of the 3D CAD system which creates the file.

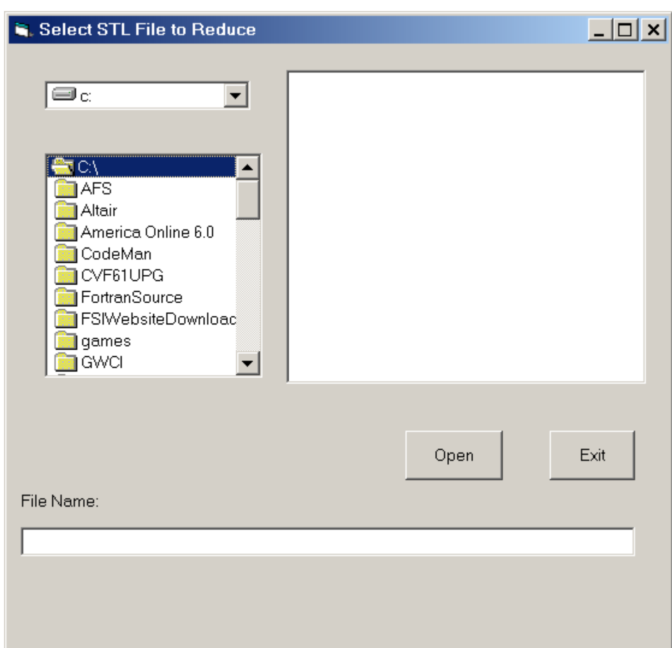
In general, when the number of triangles in a file exceeds about 60,000 or 70,000, the graphic performance of SOLIDCast begins to degrade. This depends, of course, on the specific computer speed and video system – the faster the computer, the more triangles it can easily handle.

We have found that usually about 30,000 to 40,000 triangles are sufficient for 3D representation of most commercial castings. In fact, many castings can be well represented by less than 10,000 triangles.

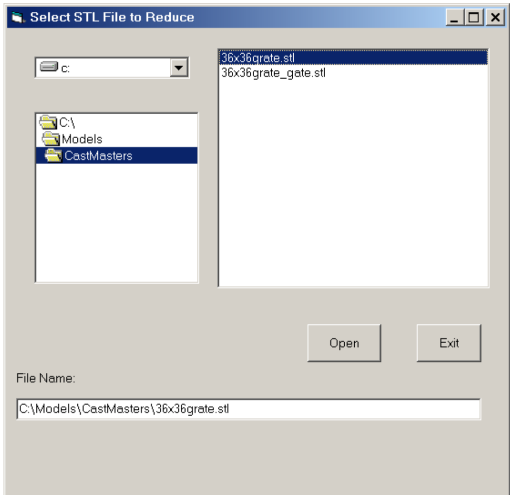
In a binary STL file, the number of triangles can be estimated from the file size. A file of 1 million bytes can hold about 20,000 triangles. Therefore, a file with a size of 3 million bytes (for example) will consist of about 60,000 triangles.

Reduce STL File Size… uses a proprietary algorithm to remove smaller triangles from an STL file while still maintaining a continuous surface. This is basically a compression routine, so the more compression that you use, the more possibility that the surface of the casting will begin to appear “warped”. However, in many cases a compression of 50% or more will still produce a usable and smaller STL file.

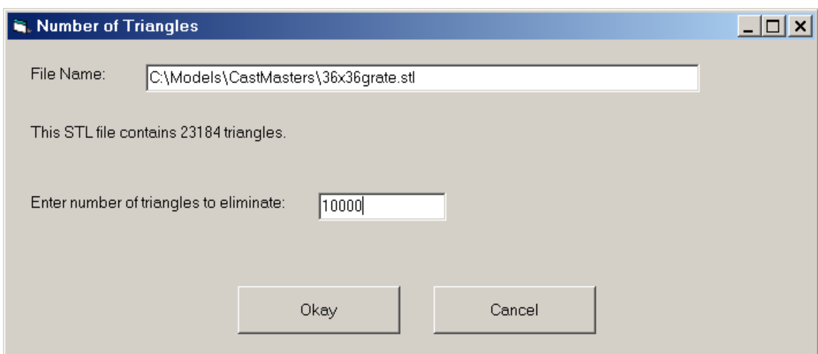
To use this utility, load a model, then select Tools…Reduce STL File Size. You will see a screen similar to the following:



Navigate to the folder containing the STL file that you want to reduce. As shown below, select this file and click the Open button:

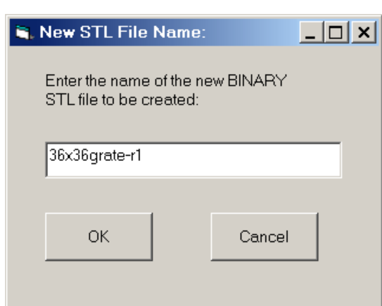


Next you will see a window which indicates the number of triangles in this STL file, as follows:

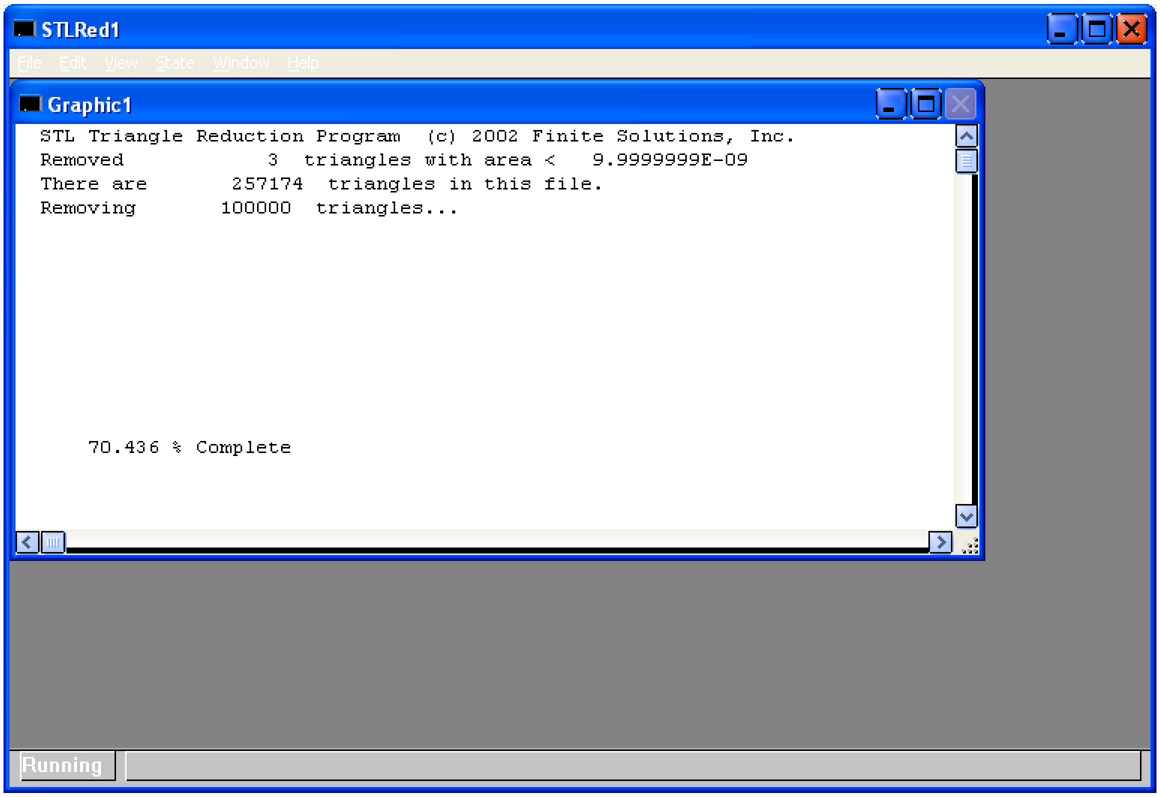


On this window is an entry field that allows you to enter the number of triangles to eliminate from the STL file. In the example shown, the original file contains 23184 triangles, and we are asking the utility to remove 10000 of those, leaving 13184 triangles.

After clicking Okay on this screen, you will see the following:

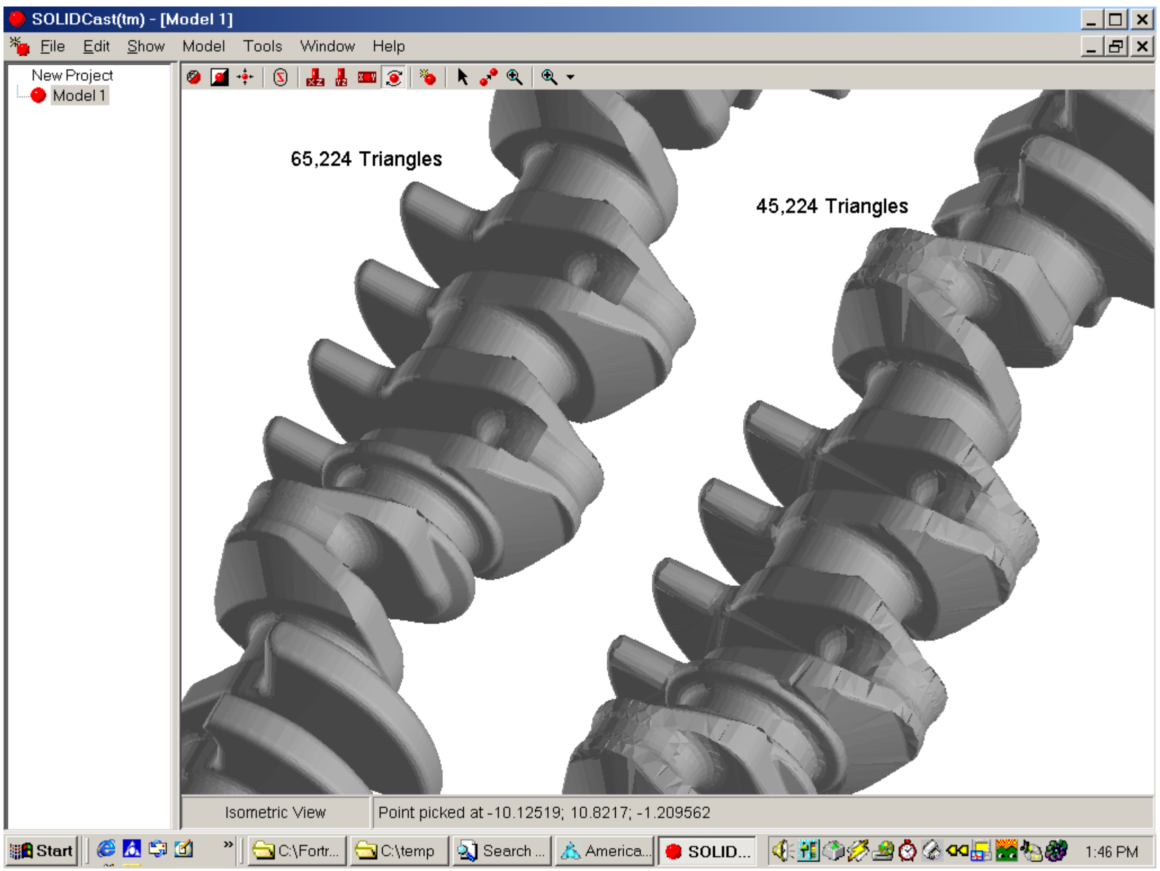


This screen asks you to enter the name of a NEW STL file to create. The original file will be maintained, and the new file will be created in the same folder as the original. Enter the name and press OK. You will then see a screen like the following:



This screen shows the progress of the reduction process as it occurs. When the % Complete indicates 100%, the utility will terminate. The new STL file should then be available to load as a shape into SOLIDCast.

The following example shows a crankshaft model that has been reduced in size from 65,224 triangles to 45,224 triangles. You will notice some small surface imperfections that appear in the reduced STL file which result from the compression operation. These imperfections are more noticeable when you zoom in closer to the shape as was done here. However, in this case these surface imperfections do not substantially affect the overall shape and mass of the crankshaft for the purpose of solidification modeling; the simulation results should be the same using either of these models. The only difference will be a slight difference in the graphic response time when manipulating these shapes on the screen. One major factor to consider is a situation where these shapes might be duplicated, for example, in a multi-cavity mold. Three of the original-size shapes would mean a total of 195,672 triangles in the total model, while three of the reduced shapes would equal 120,672 total triangles. This may have an impact on the graphic performance of SOLIDCast.

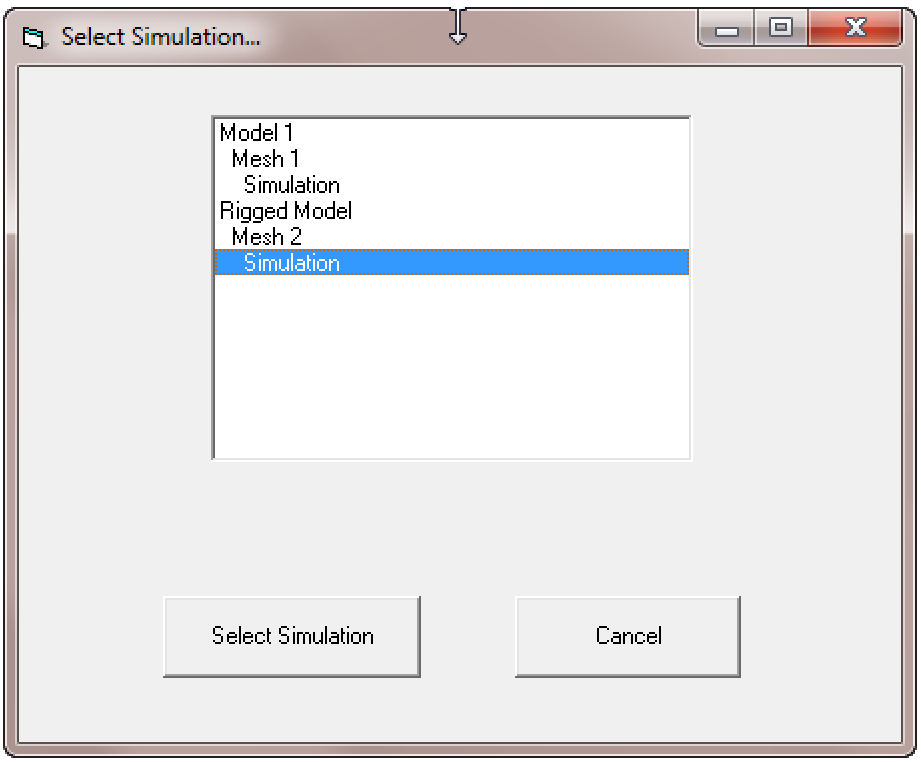


Sample crankshaft: Original and reduced-triangle-count STL files

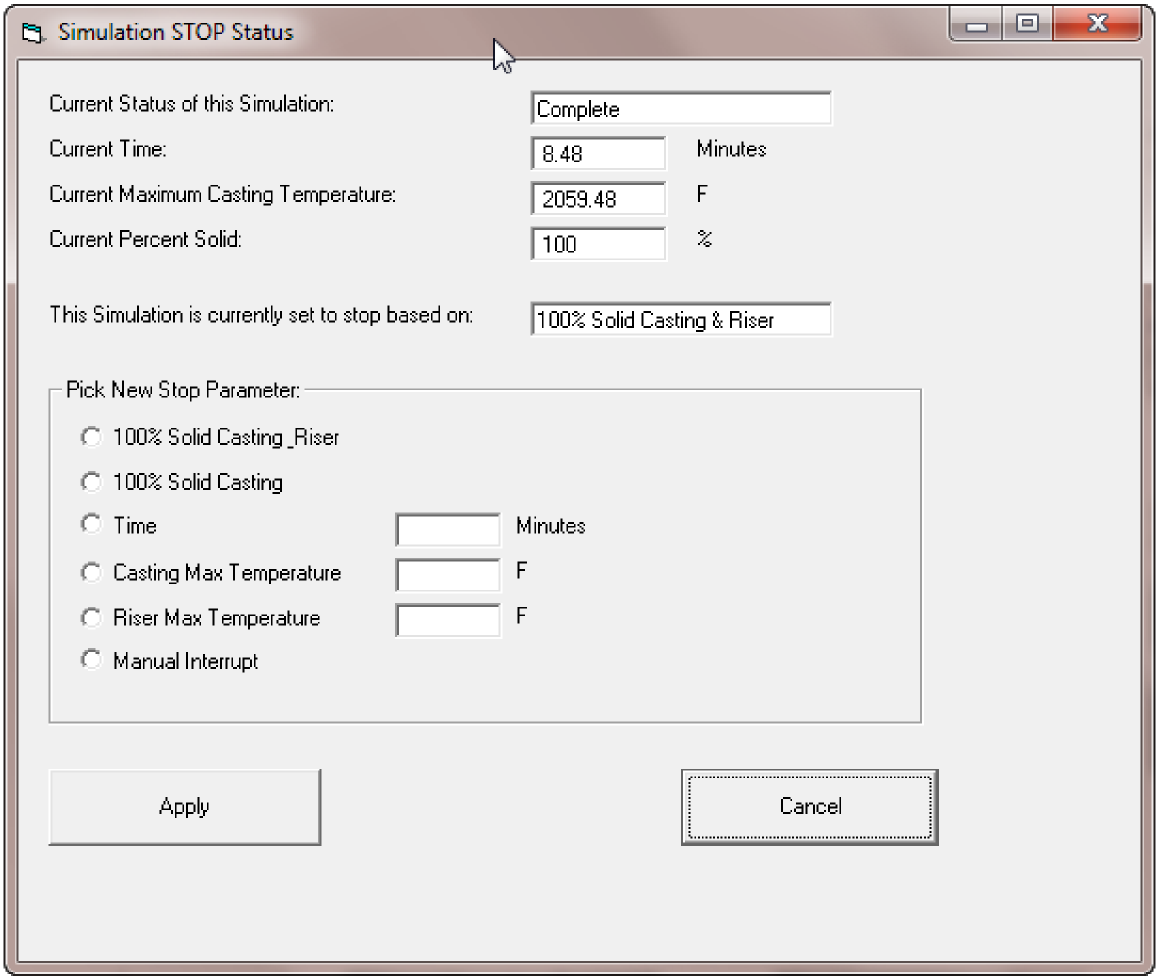
UTILITY: Reset Simulation Stop Point

This utility is used to change the point at which SOLIDCast assumes that the simulation is complete. When you first start a simulation, you can choose from a number of options to end the simulation. The default setting is when both the Casting and Riser materials are 100% solid. However, you may want to run to a certain time, or to a certain temperature in the casting, or even run until a manual stop. Then, after examining simulation results, you may want to change the stop point in order to continue simulating for a longer time. This utility allows you to make that change.

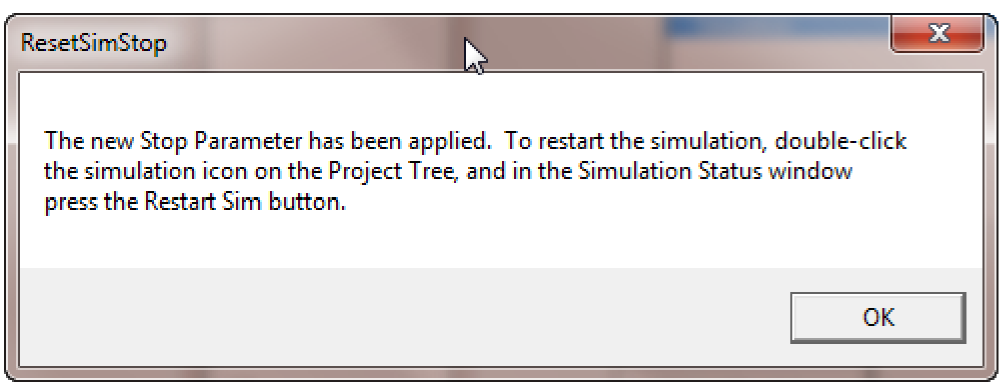
To use the program, first load a model, then go to Tools…Reset Simulation Stop Point. The system will display the current project tree and ask you to select the simulation that you want to adjust, as shown here:



Use the mouse to highlight the simulation name on the Project Tree, then click the Select Simulation button. The system will display a window as shown on the following page.



The top part of the windows shows the current state of the simulation. Pick New Stop Parameter box lists the options available. Click the button to the left of the option you want, and enter a stop time or maximum temperature if you have chosen one of those options. Then click Apply. The system will display the following:



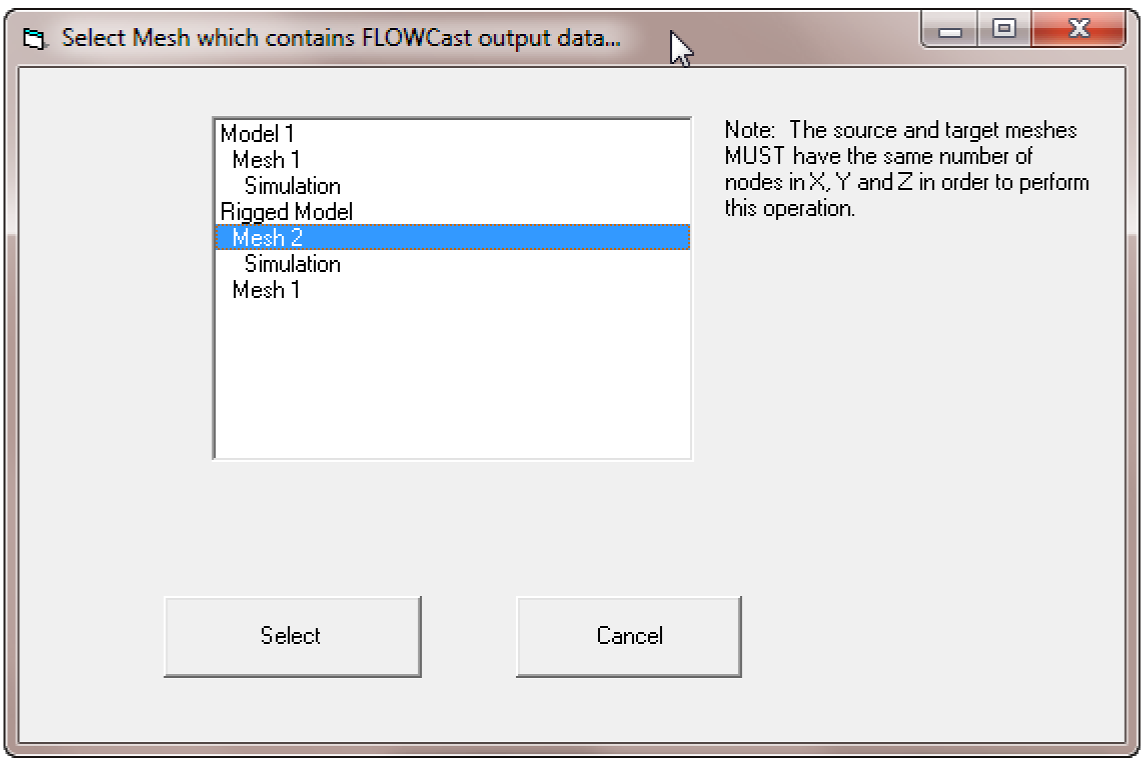
Click on OK to return to the Model Building window.

UTILITY: Reuse FLOWCast Results

This utility allows you to take the temperature distribution in a model calculated by a FLOWCast filling simulation, and place it into another mesh of the same size. Note that the X, Y and Z number of nodes must be exact from one mesh to the other in order to use this utility.

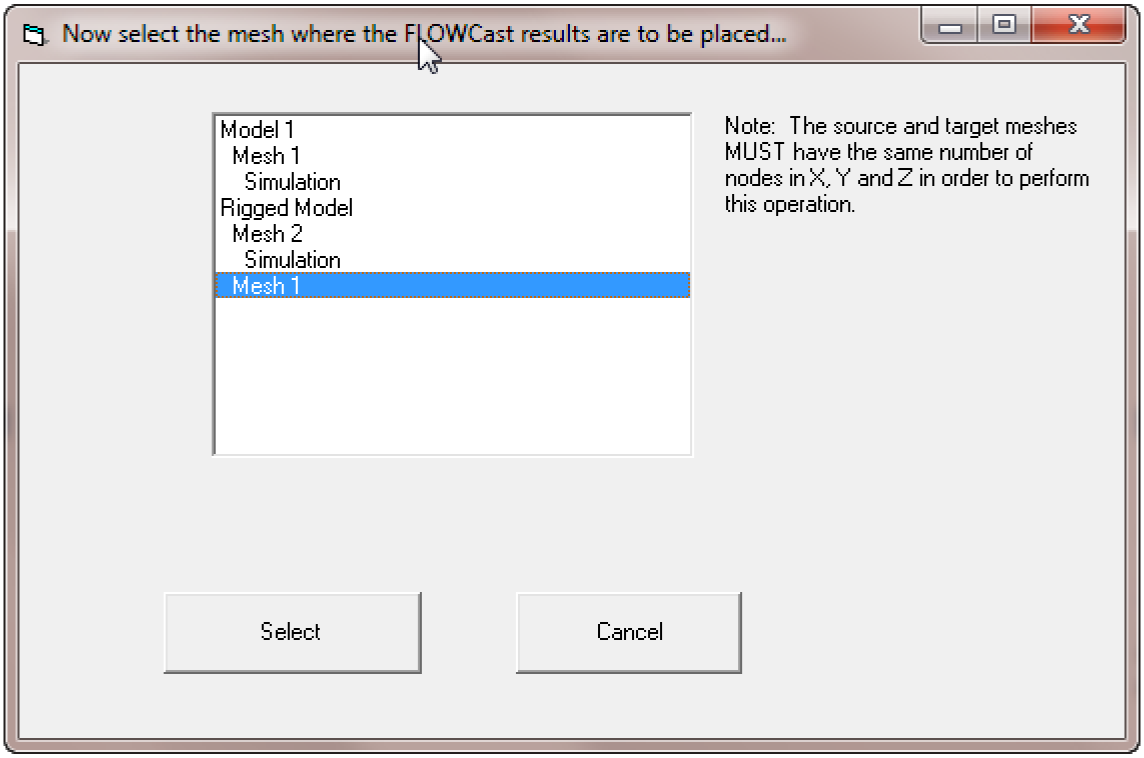
A possible use of this tool would be if you would be running a series of simulations where only minor geometry or process changes are made to a model. Using this utility can give you a good filling analysis without having to run the entire FLOWCast fill each time. It can also be used when simulating direct pour sleeves. An initial simulation of filling can be done with the filter in place at the base of the sleeve. A second simulation can be done without the filter, but using the filling results from the filter model as the starting point of the solidification analysis.

To use this utility, load a model, then click on Tools…Reuse FLOWCast Results. You should see a screen similar to this:

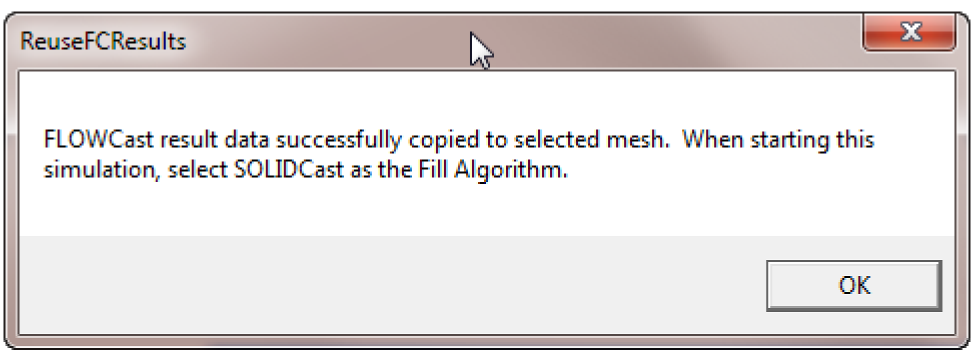


On the Project Tree highlight the name of the mesh that has already been run using FLOWCast. Then click Select.

The system will then display:



Highlight the name of the mesh where the previous FLOWCast results will be copied. Then click Select. Once the data has been copied over, you should see the following:



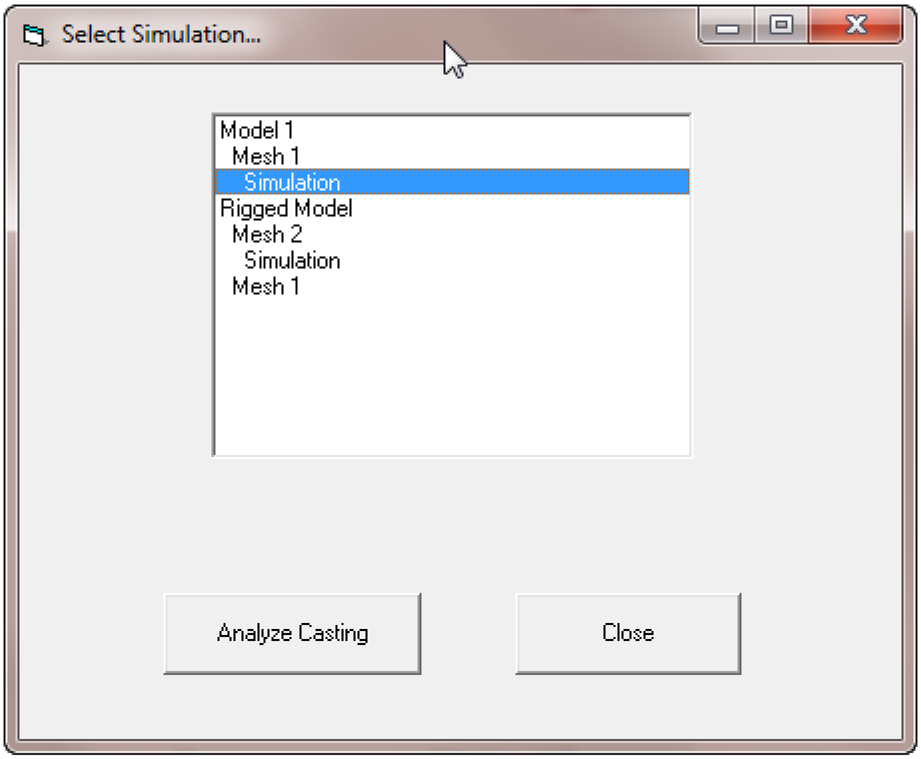
Clicking OK will return you to the Model Building screen.

Now, when you highlight the second mesh on the Project Tree and start a simulation, make sure you select SOLIDCast as the Filling Algorithm. The system will automatically detect the FLOWCast data, load the temperature distribution and start the solidification simulation using the FLOWCast data as a starting point.

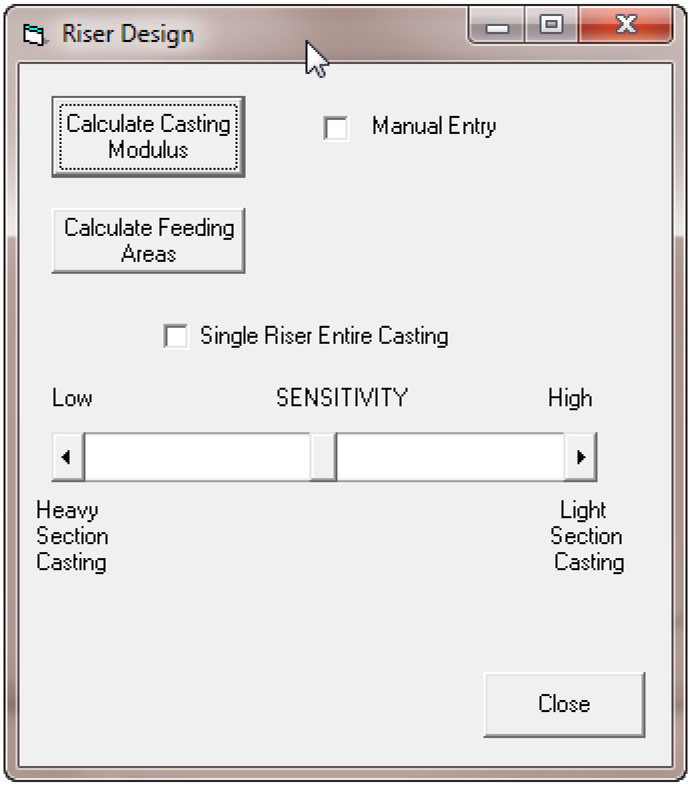
UTILITY: Riser Design

This utility gives 'direct access' to the riser design functions of SOLIDCast, without going through the riser design wizard. Sometimes, this can be more efficient, if you are just looking for a modulus calculation, or to check on a riser size.

To run the utility, first load a model, then click on Tools…Riser Design. You should see the following:



Highlight the name of the simulation that you want to do a riser design for, then click on Analyze Casting. You should then see the riser design screen, as shown on the next page.



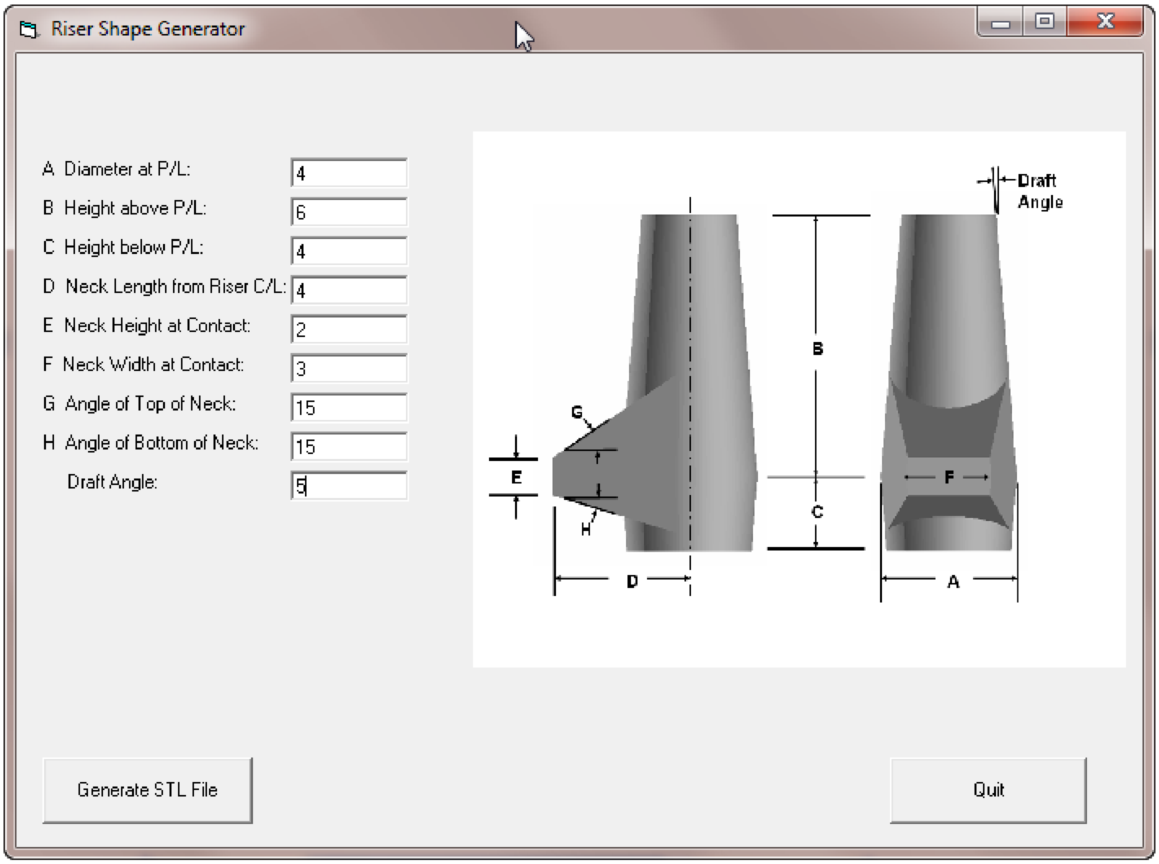
At this stage you will have the same options available to you as in the Riser Design Wizard. Full instructions on using the wizard are given in Unit 45.

If you entered this utility by mistake, click the Close button to be returned to the Model Building screen.

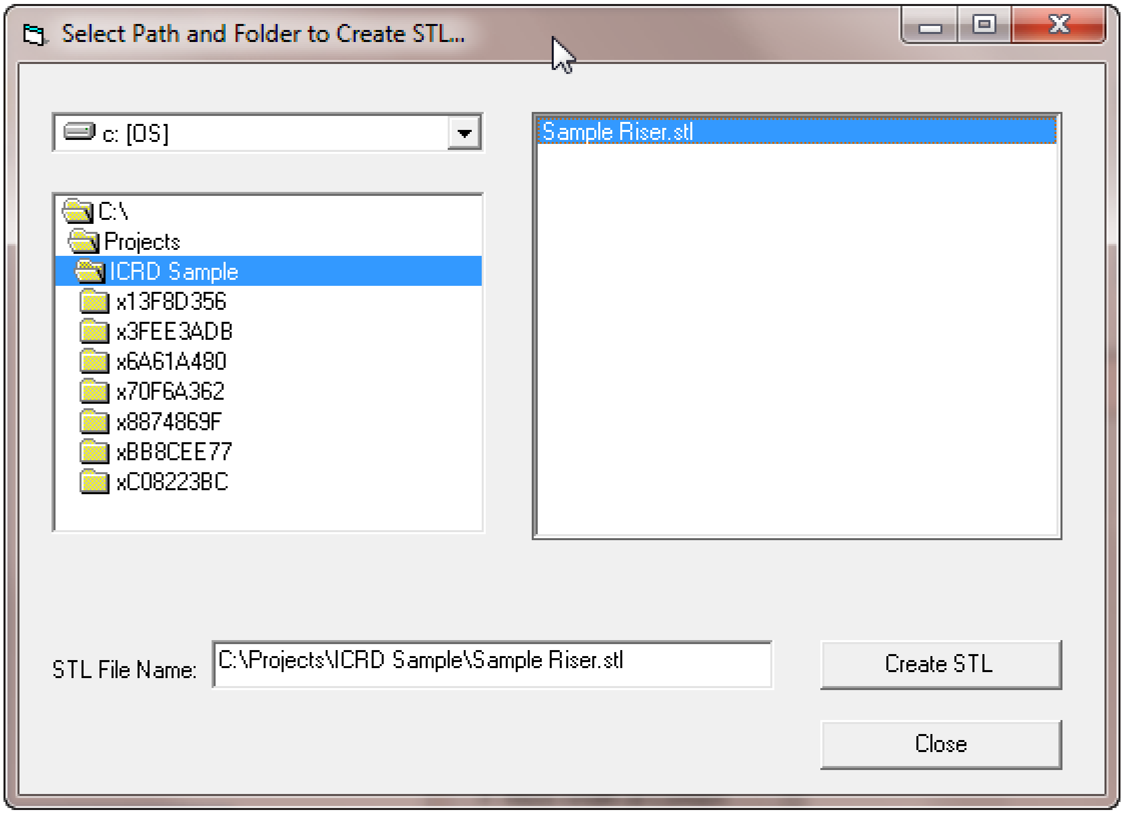
UTILITY: Riser STL Shape

This utility allows you to create a riser and contact by entering values for riser size, contact size and taper. The result is saved as an STL file, that can then be brought into a SOLIDCast casting model.

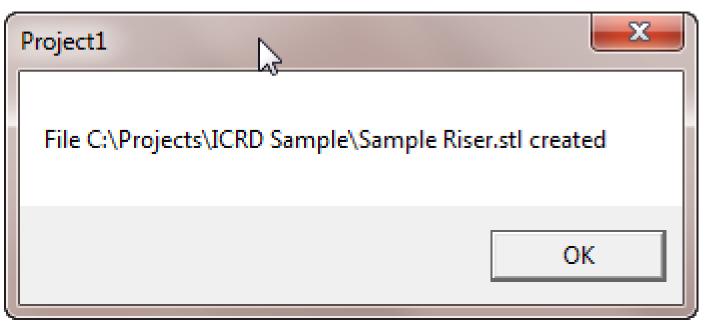
To run the utility, first load a model, then select Tools…Riser STL Shape. You should then see the following entry screen:



Once you have entered the data, click on Generate STL File to finish. You will be asked to name the STL file, as shown on the next page.

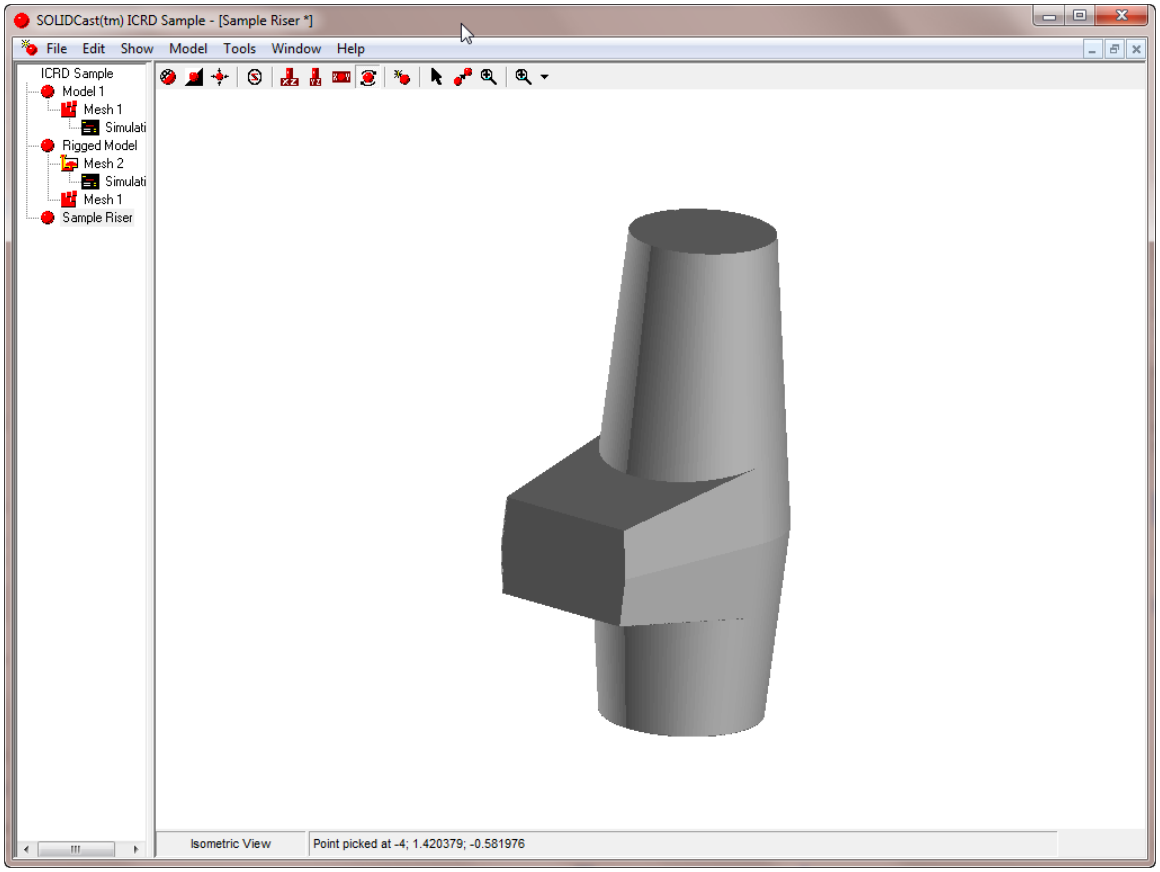


Once you enter or highlight a file name, click on Create STL. If all goes well, you should see a message similar to this:



Click OK to return to the Model Building window.

As an example, the riser created by the data entered in the screen on the previous page would look like the figure shown on the following page, once loaded into the Model Builder.



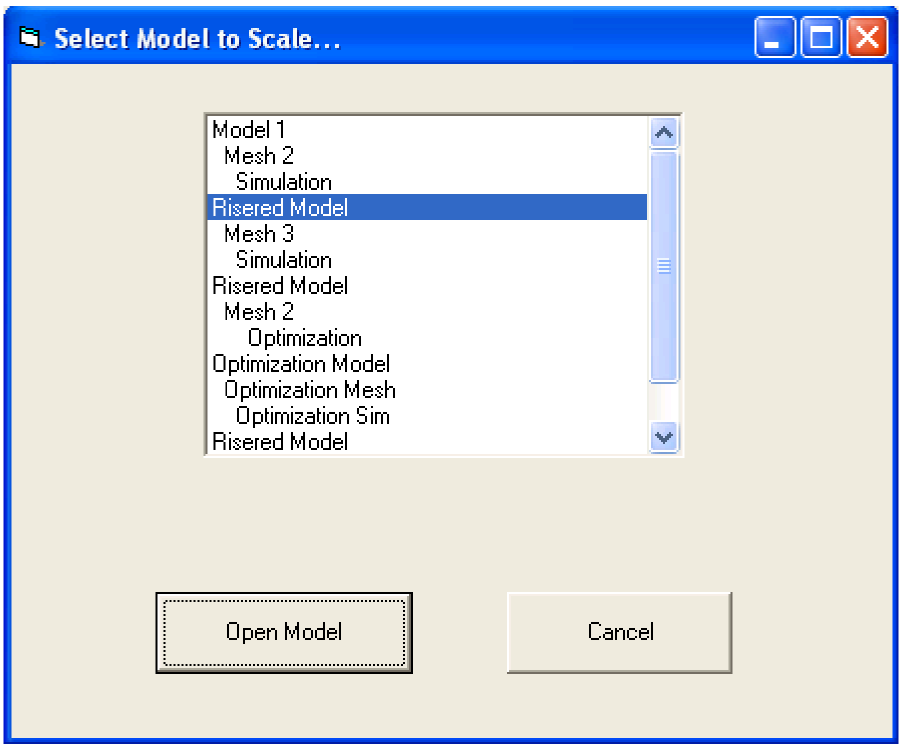
Riser STL file loaded into the SOLIDCast Model Builder.

UTILITY: Scale Model Size

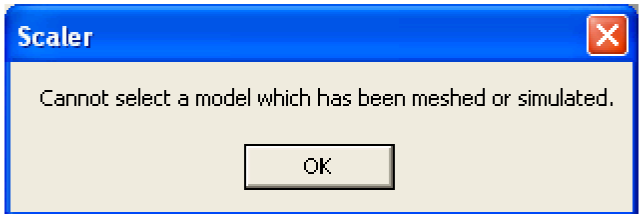
This utility allows you to scale a model in a SOLIDCast project. You can enter a single scale factor to make the model larger or smaller. This function may be useful in a "family of parts" situation, where parts are similar in size but just differ in scale.

This may also be useful to correct an error of interpretation. For example, suppose that your system is set to English Units and you import STL files which are in mm, but do not tell the system to convert them. SOLIDCast will then interpret the dimensions as inches. Your model will then be much larger than actual size. You could user Scaler.exe to scale the model down to the correct size (in the case, the scale factor would be 0.03937), without having to start over with the import operations.

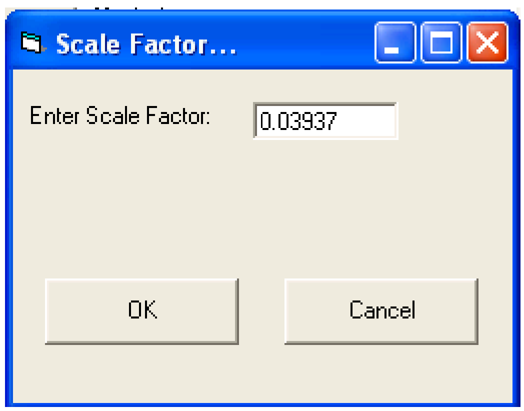
To run the utility, load a model then select Tools…Scale Model Size. You will see the Project Tree displayed. Select the Model entry on the Project Tree that contains the model that you want to scale:



Note that this utility WILL NOT SCALE A MODEL WHICH HAS BEEN MESHED AND SIMULATED. If there is a mesh or simulation entry under the model, then Scaler will not allow that model to be selected, as shown here:

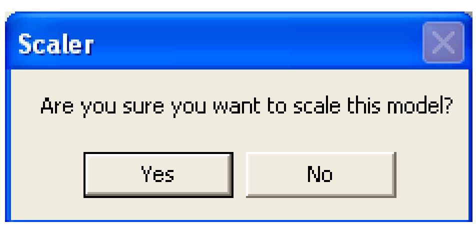


Once you have selected a proper model, click the Open Model button. You will see the following:



Enter a scale factor to scale the model size, and click OK. A scale factor of 2 will double the dimensions of the model. A scale factor of 0.5 will produce a model which is one-half the original size.

The system will verify one last time that you want to scale this model, as follows:



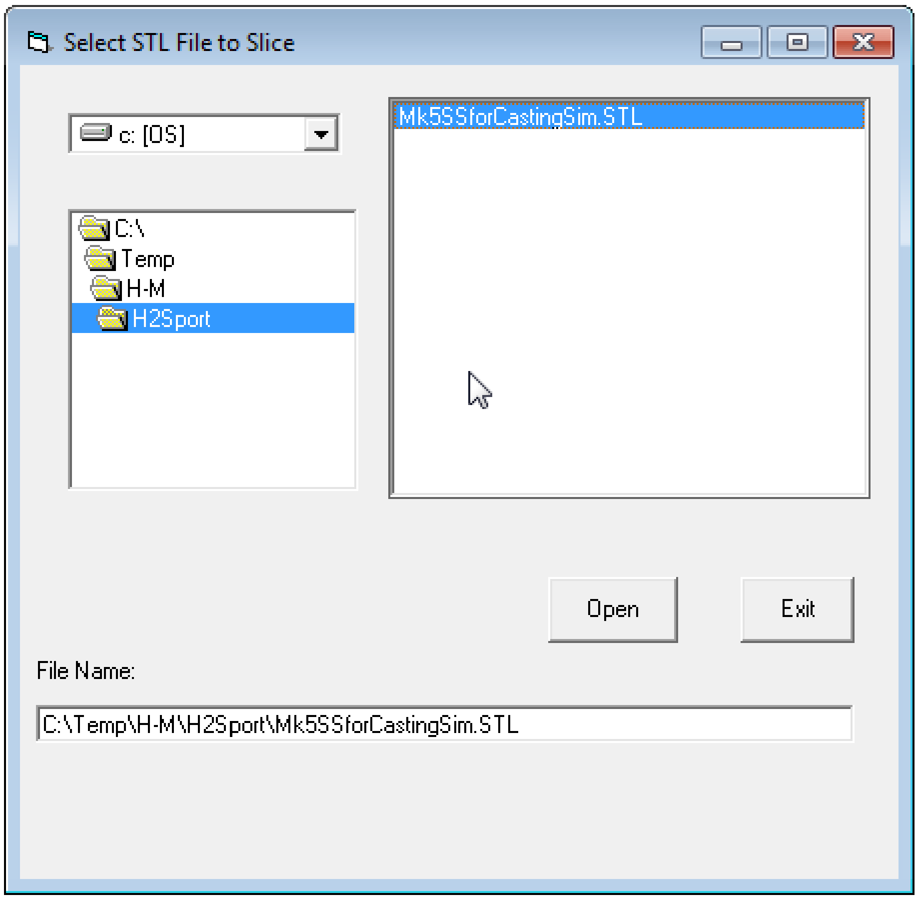
Click Yes to proceed.

The entire model will be scaled larger or smaller, according to the scale factor entered. You can then load the model to view the change and proceed with simulation of the scaled model.

UTILITY: Scale STL File

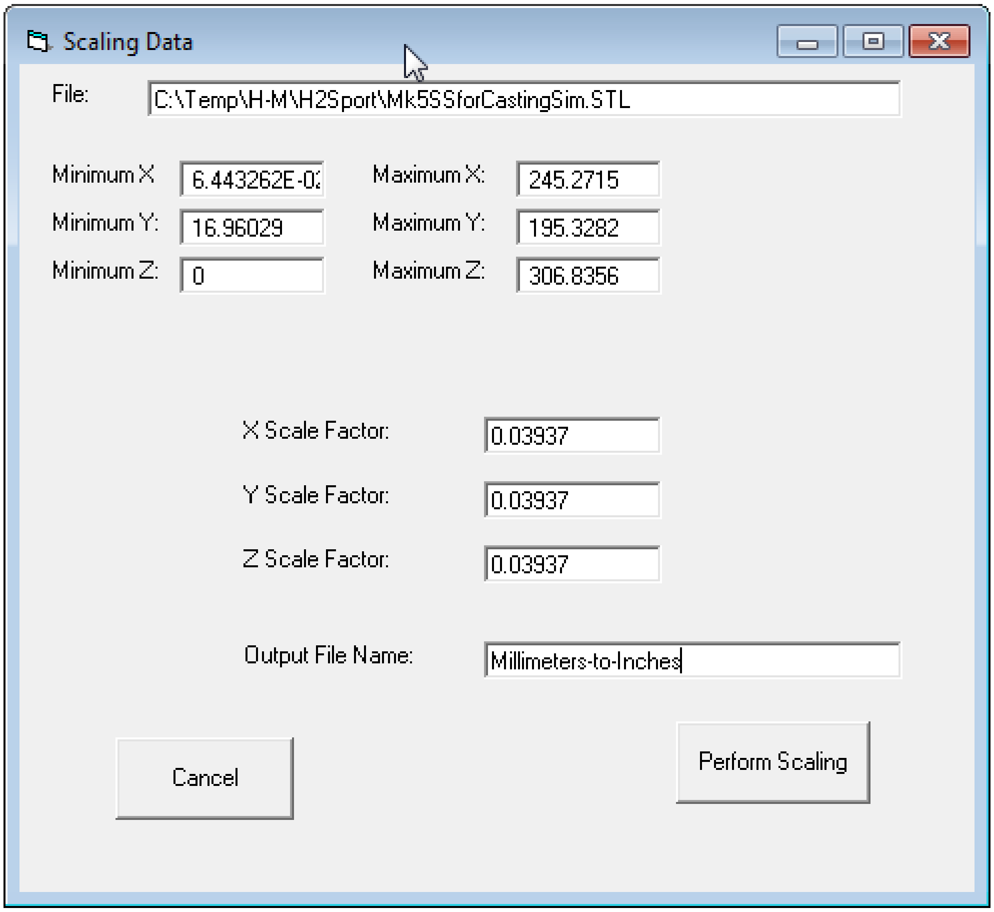
Sometimes an STL will be created using the wrong units for use in SOLIDCast. For example, the file may be save in centimeters, when SOLIDCast expects either inches or millimeters. The Scale STL File Utility can expand or contract the size of an STL file in the X, Y and Z directions.

When you select the Scale STL File utility, you should see the following screen:



Navigate to the folder where the STL file you need to scale is located. Highlight the name of the STL file in the window on the right, then click Open.

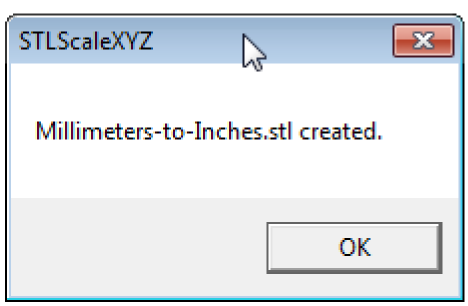
The file will be analyzed, and you should see a window similar to that shown on the next page.



The window will display the size of the model in the X, Y and Z directions. You can then enter X, Y and Z scale factors into the blanks in the middle of the window, along with a name for the scaled file. The example above shows how to convert a file in millimeters to one in inches, by entering scale factors of 1/25.4=0.03937.

Once you have entered the scale factors, click the Perform Scaling button.

Once the scaling operation has finished and the file has been created, you should receive a message similar to this:

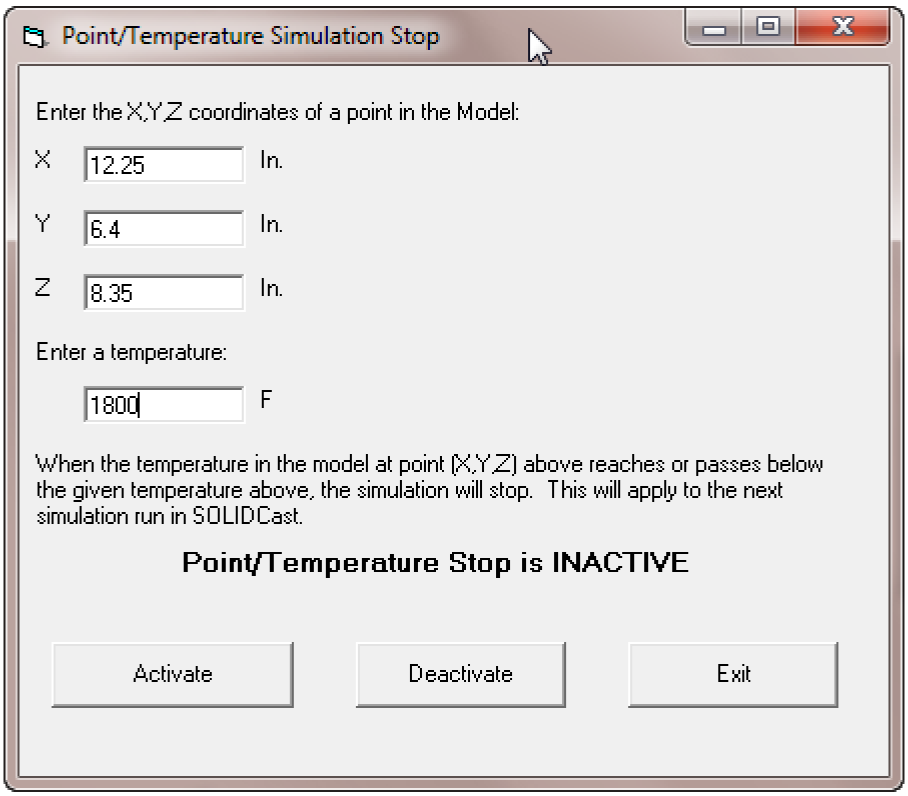


Click OK to return to the Model Builder.

UTILITY: Set Up Point/Temperature Stop

This utility allows you to specify a given X, Y, Z location in a model to be used to monitor temperature. Once the temperature at that spot reaches or drops below the specified set point, the simulation will stop. This differs from the default temperature stop point in that it is for a specific location in the model, not the maximum temperature in the whole casting (or riser).

To run this utility, load a model, then click on Tools…Set Up Point/Temperature Stop. You should see a screen similar to this:



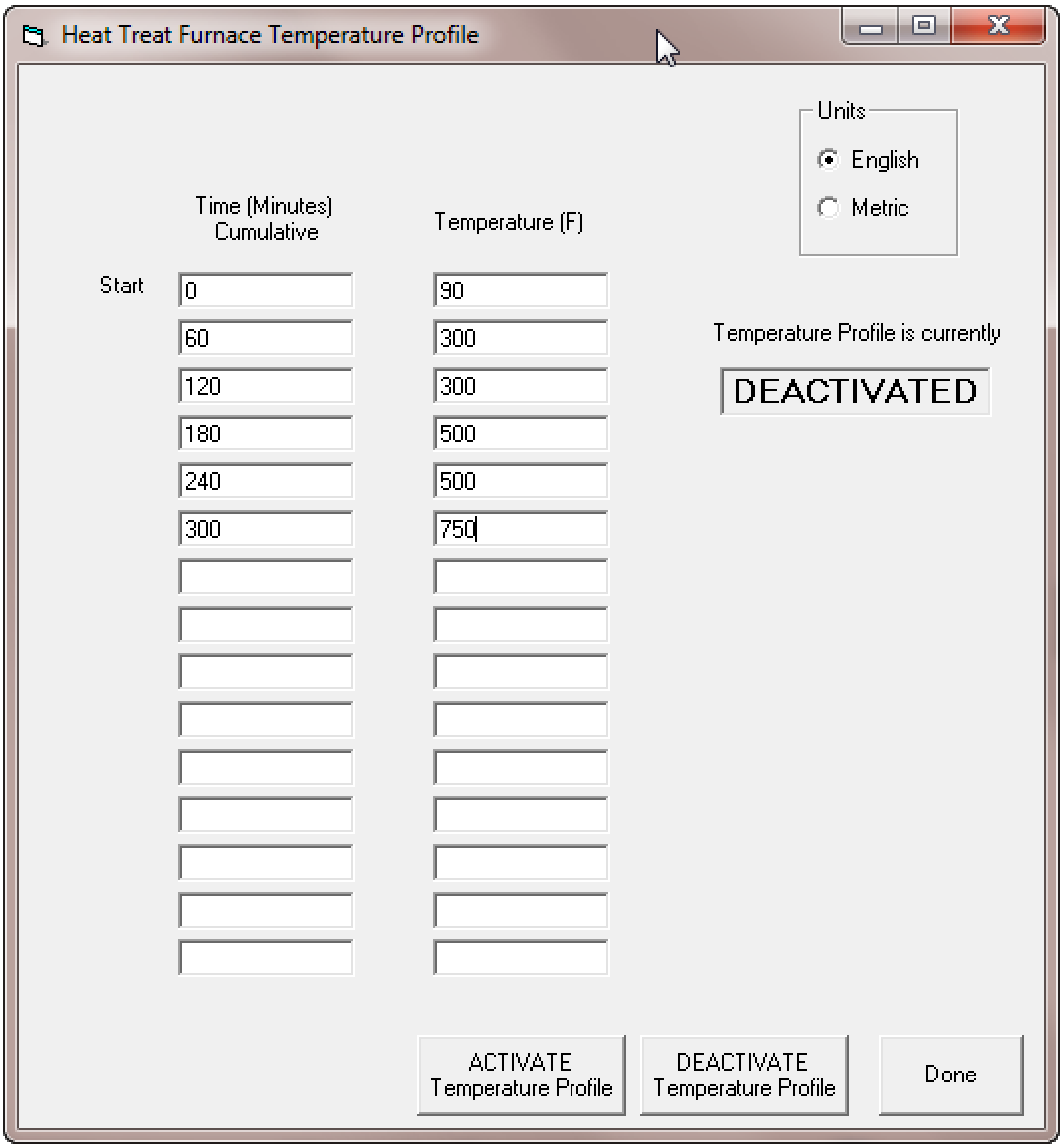
Simply type in the X, Y, Z location of the point in the model that you want to trigger the simulation stop, as well as the temperature at which you want the simulation to end. One you have made these entries, click on Activate. Finally, click on Exit to go back to the Model Builder screen.

Once you have activated this function, simulation stops will be governed by this. So, once your simulation is complete, run this utility again, and click Deactivate to turn the function off.

UTILITY: Set Up Variable Furnace Temp

This utility helps you to set up a furnace temperature ‘ramp-up’ schedule for a heat treat simulation. You can set up to 15 time-temperature set points, and you can activate or deactivate the schedule for subsequent simulations.

To use the utility, load a model, then click on Tools…Set Up Variable Furnace Temp. The system should display:



You can have the system ramp up a temperature over a given time, or you can hold for a period by entering the same temperature at each of two time periods.

Once you have your temperatures entered, click on ACTIVATE Temperature Profile. The system will store the temperature profile, and the listing will change from DEACTIVATED to ACTIVATED.

Once you have activated the profile, click on Done to return to the Model Builder screen.

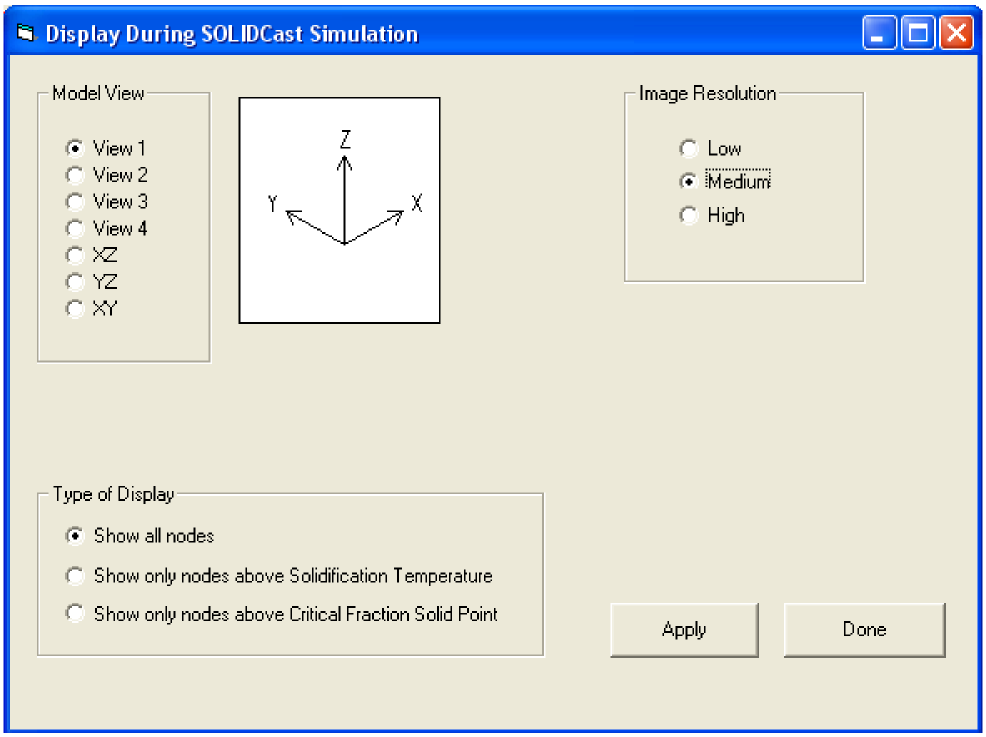
Now, when you run a simulation, you will be simulating the heat treat process with a temperature ramp up cycle, as defined by this utility. For more details on running a heat treat simulation, see Unit 51 of this workbook.

Note that when you are finished with the simulation, you should run this utility once more and click on the DEACTIVATE Temperature Profile button, so that it will not affect other simulations.

UTILITY: Simulation Image Control

This utility allows you to adjust how the simulation graphics appear while a simulation is running. You can vary the picture resolution, the viewing angle and whether the solidifying metal turns gray or transparent. This utility can be run prior to a simulation, or while the simulation is actually running.

To run the utility, load a model then select Tools…Simulation Image Control. You will see a window that allows you to control the Model View, Image Resolution and the Type of Display:



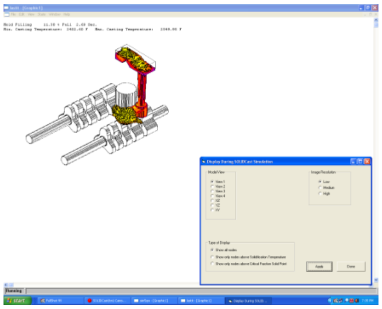
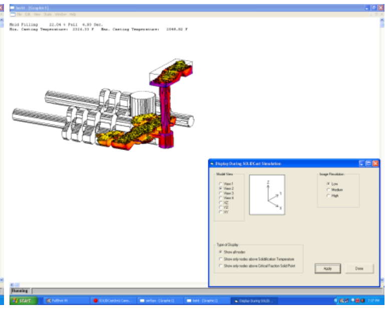
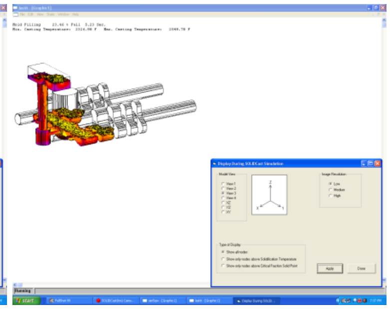
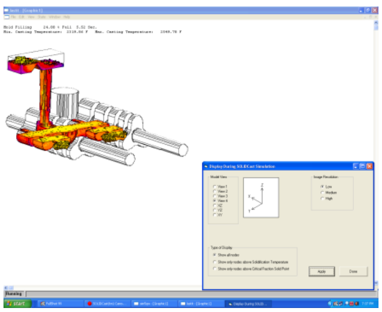
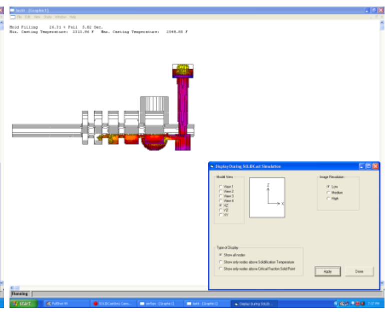
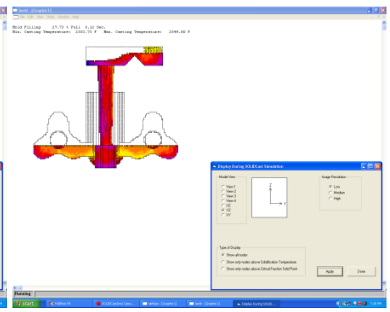
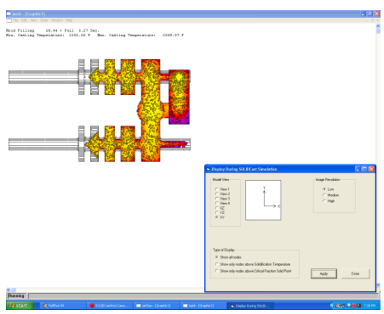
The system default is View 1, Medium Resolution and Show all nodes, as shown above.

To adjust the viewing parameters, click the button for the desired view, resolution or display type, then click the Apply button. When you have the parameters set the way you want, click Done.

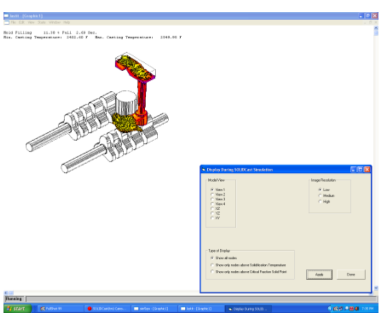
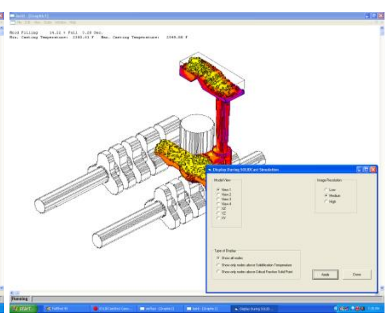
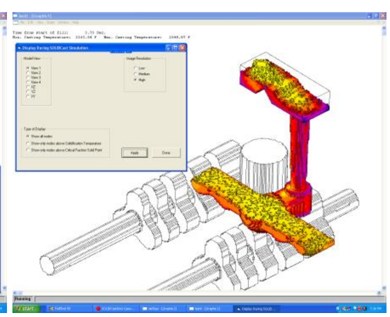
If the parameters have been adjusted prior to a simulation run, they will be applied when the simulation is started. If the utility is activated during a simulation, the screen will be updated as soon as you click Apply. In fact, you can make several adjustments, click Apply, and view the results. If you are not satisfied, you can change them, then click Apply once again. This can be done as often as you wish. Once you have parameters you like, click Done to remove the selection window.

The following figures show a view of a camshaft model during filling and simulation. The screens were captured with the selection window visible. This allows you to see clearly the effects of different selections.

The screens below show the differences in model views:

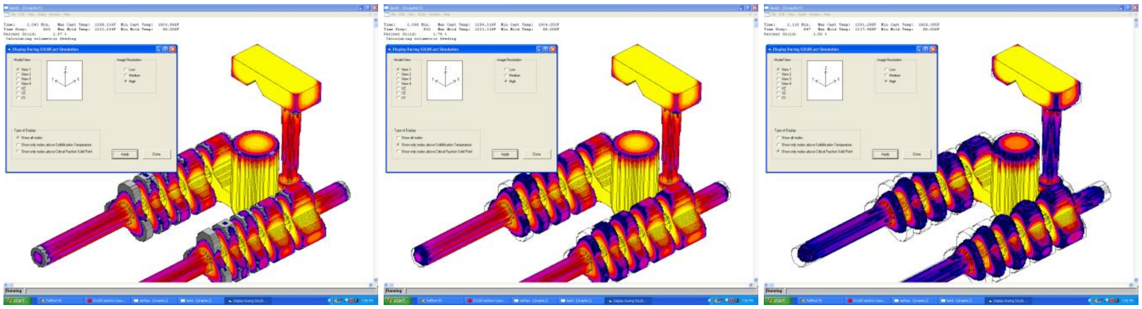








The screens below show the differences between low, medium and high image resolution:

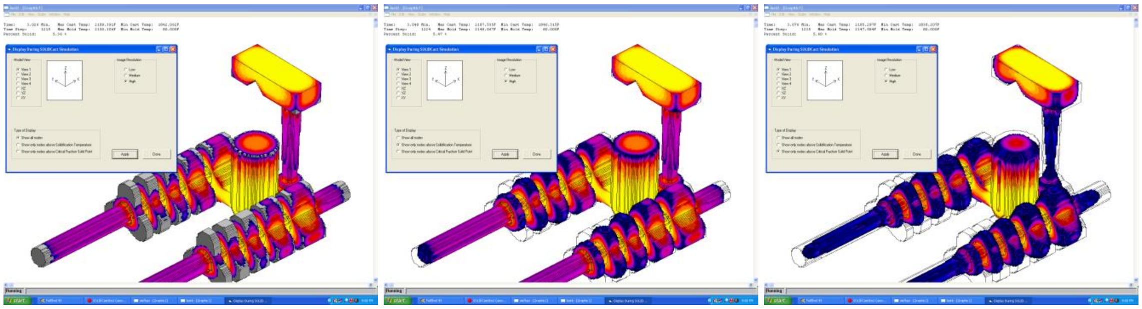




These screens show the type of display selection affects the view of the part. Two sets of screen captures are show, to show the effect at different stages of solidification:

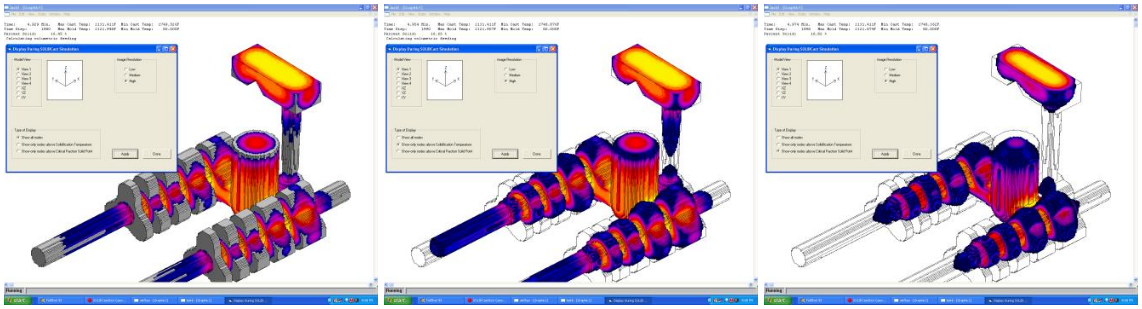
Early on



Midway



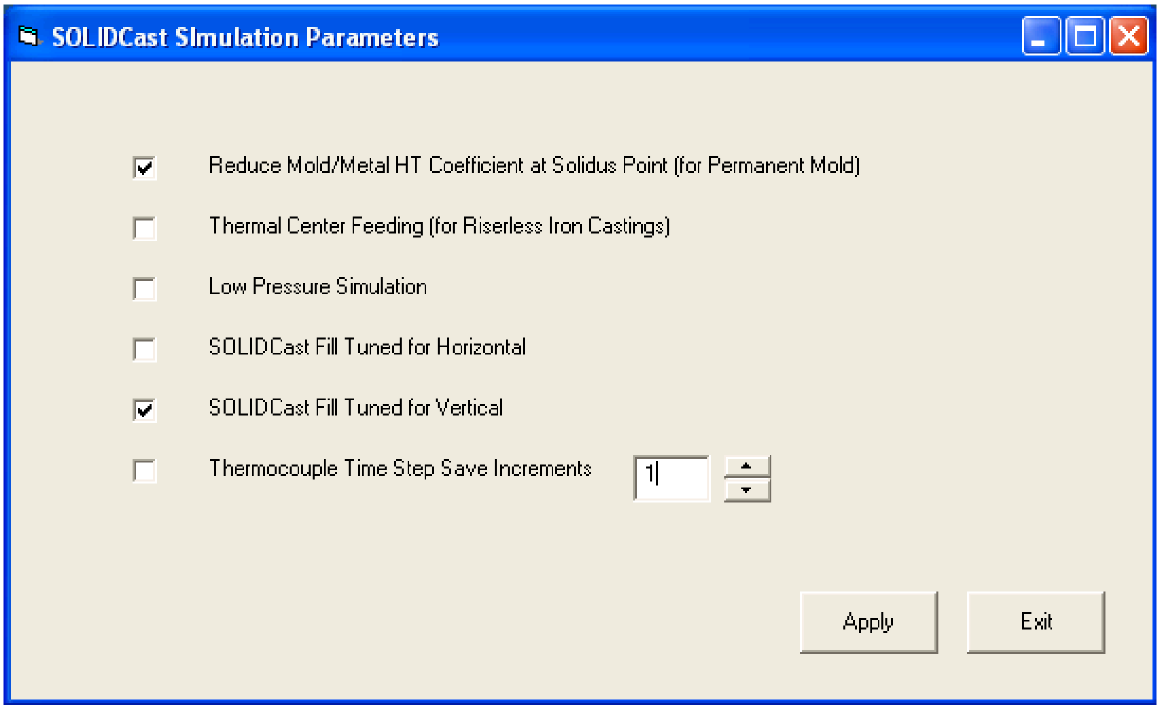
Still Later



UTILITY: Simulation Parameters

This utility is used to set several simulation parameters that in previous versions of SOLIDCast had to be set by creating and editing text files in the installation folder. This utility simplifies their use. Detailed information on each setup file and its function are given below.

To run the utility, load a model, then select Tools…Simulation Parameters. You should see the following screen:



Use the cursor to check or uncheck a box. Note that you cannot select both SOLIDCast Fill Tuned for Horizontal AND SOLIDCast Fill Tuned for Vertical. Selecting the second one will de-select the first one. If neither SOLIDCast Fill box is checked, the system will default to Horizontal tuning.

Reduce Mold/Metal HT Coefficient at Solidus Point (for Permanent Mold)

Research has suggested that the Heat Transfer Coefficient at the surface between the mold and the casting decreases as a step function at the solidus point in permanent mold processes. The best fit to this data indicates that the HTC decreases to 30% of its starting value. This effect can be simulated by checking this box and clicking the Apply button prior to running the simulation. If this box is not checked, SOLIDCast will assume no reduction in HTC.

Make sure that this box is NOT checked if you are running any simulation other than permanent mold, or you will get strange results.

Thermal Center Feeding (for Riserless Iron Castings)

This controls how feeding occurs. If the box is cleared, feeding is based on gravity feeding from risers. If checked, feeding occurs outward from thermal centers in the casting.

Low Pressure Simulation

If the box is checked, then feeding will occur as a Low Pressure Permanent Mold simulation. This means that the direction of gravity will be reversed 180 degrees when the filling is complete, so that feeding of liquid metal will be done by the pressure from below rather than the force of gravity. This affects the indications as given by the Material Density Function.

Once you are finished running the simulation, run this utility again and clear the box, and click Apply. Otherwise all subsequent simulations will use Low Pressure Feeding, which will cause unusual results.

SOLIDCast Fill Tuned for Horizontal
SOLIDCast Fill Tuned for Vertical

If you are doing a SOLIDCast mold filling simulation, the behavior of the filling metal can be controlled to some extent by adjusting this parameter. By checking one box or the other, you can make the behavior of the filling more realistic for primarily horizontal flow or primarily vertical flow. You cannot check both boxes at the same time. Checking one box will clear the other. If neither box is checked, horizontal filling is assumed.

Thermocouple Time Step Save Increments

If you are using thermocouples in your model, the default action of the system is to record time/temperature data at each time step.

By checking this box, you can tell the system to record this data less often. Use the up and down arrows to tell the system how many time steps to increment before recording a time/temperature data point for any thermocouples.

For example, if this is set to 10, then data will be recorded every 10 time steps. This means that the file of time/temperature data will be 1/10th as large as recording every time step.

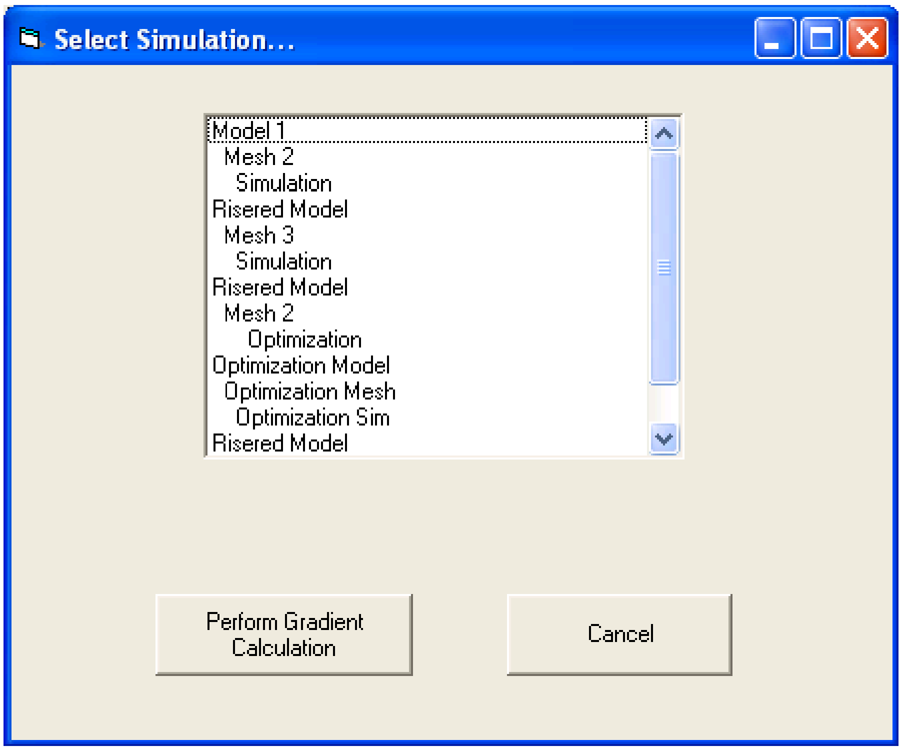
If this box is cleared, SOLIDCast will assume every time step.

Once you have made your selections, click on the Apply button to activate your changes. Then click on Exit to return to the Model Builder screen.

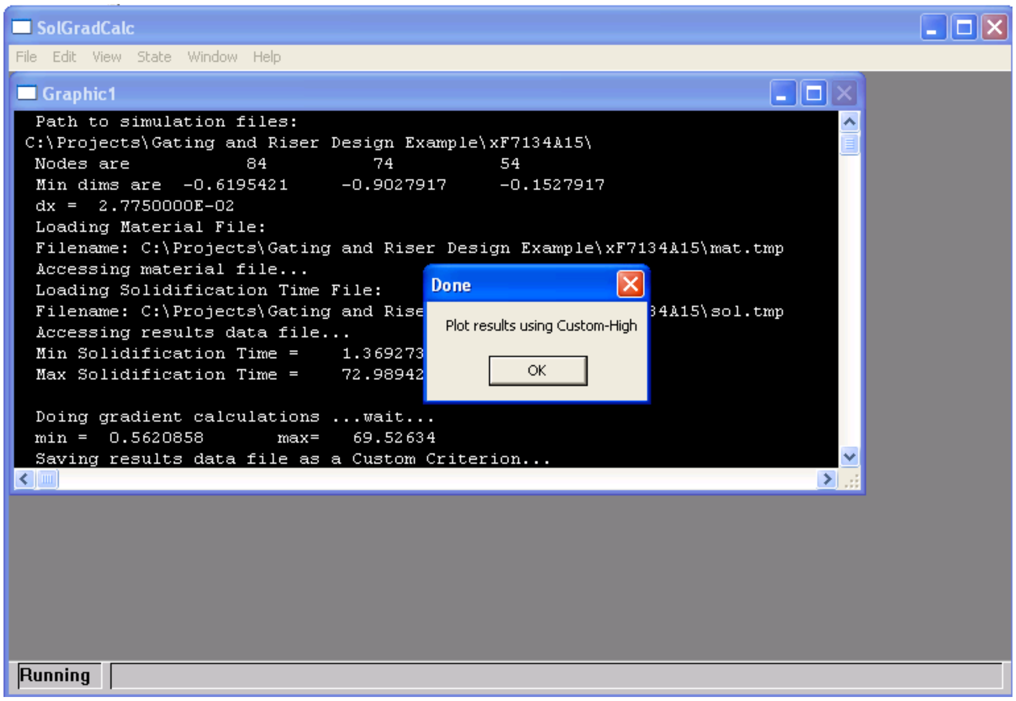
UTILITY: Solidification Time Gradient Calc

This utility calculates and places solidification time gradient calculations from a simulation into the Custom function so that they can be plotted. The Solidification Time Gradient has shown promise as a predictor of areas in a casting prone to hot tearing.

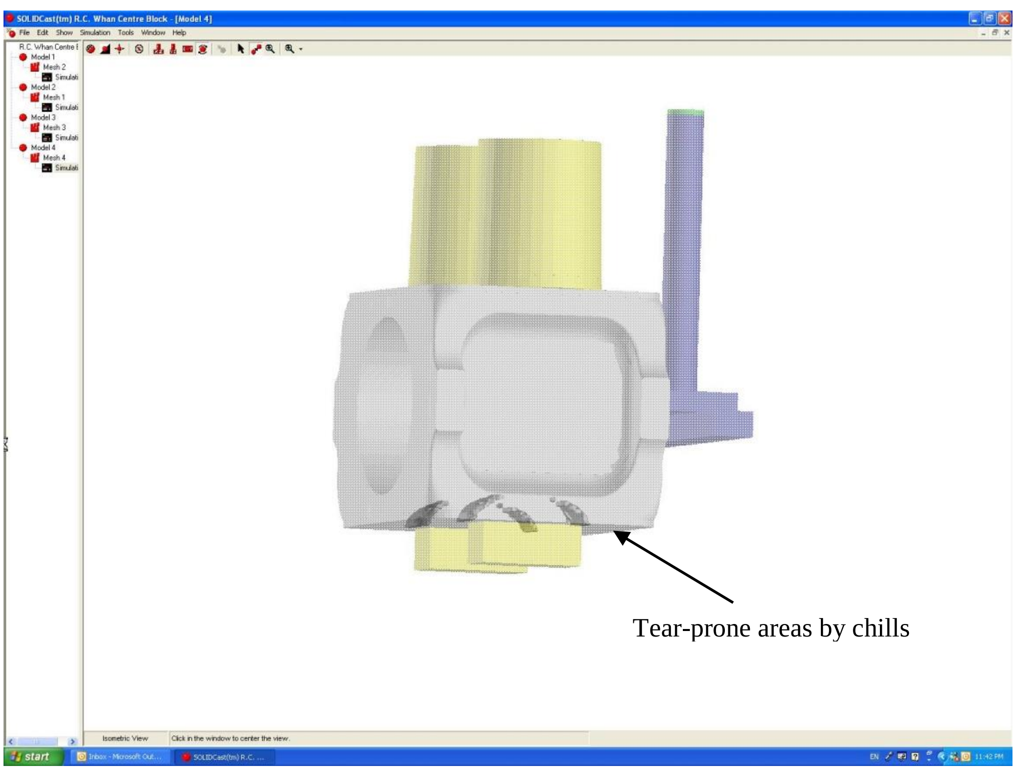
To run the utility, load a model, then select Tools…Solidification Time Gradient Calc. The system will display a window similar to this:



Use the cursor to highlight the desired simulation. Once you have done this, click on the Perform Gradient Calculation button. The system will perform its calculations as shown on the next page.



Once the calculation is done, click OK. You can now go into the SOLIDCast plotting routines and see the results by plotting the Custom-High function. During initial testing, we have found that making an iso-surface plot at about 75% of the maximum value gives good results. An example plot is shown here:



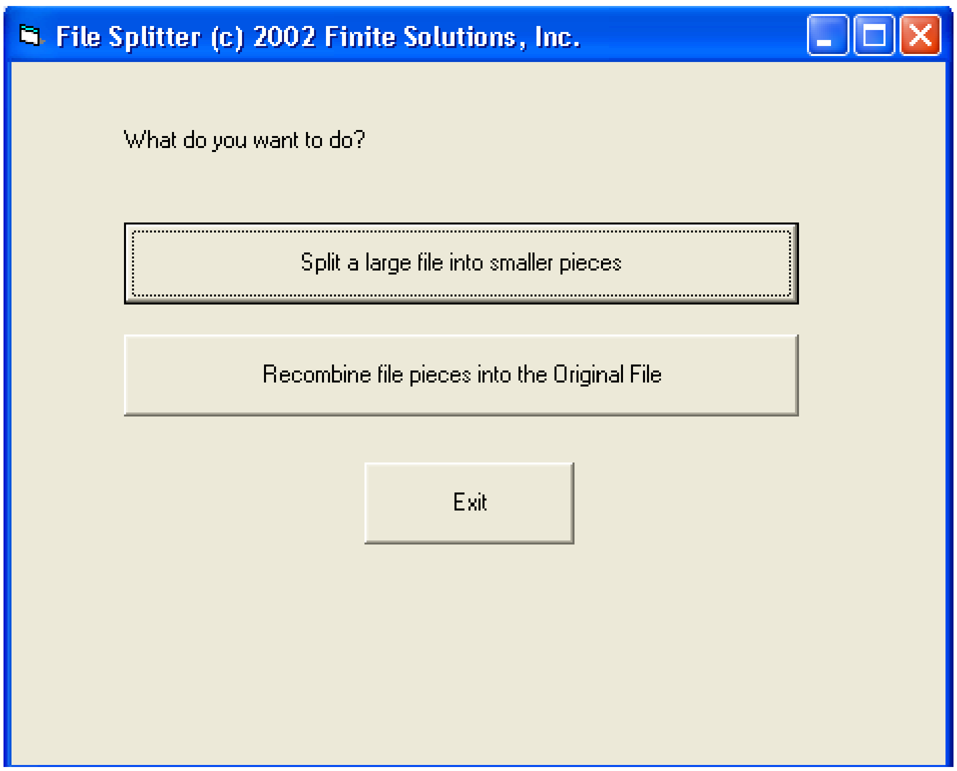
UTILITY: Split Files for Emailing

This utility is used to split large files into multiple smaller files and later recombine the pieces to recreate the file. This can be very useful if you need to email large files, but the receiver cannot accept such a large file.

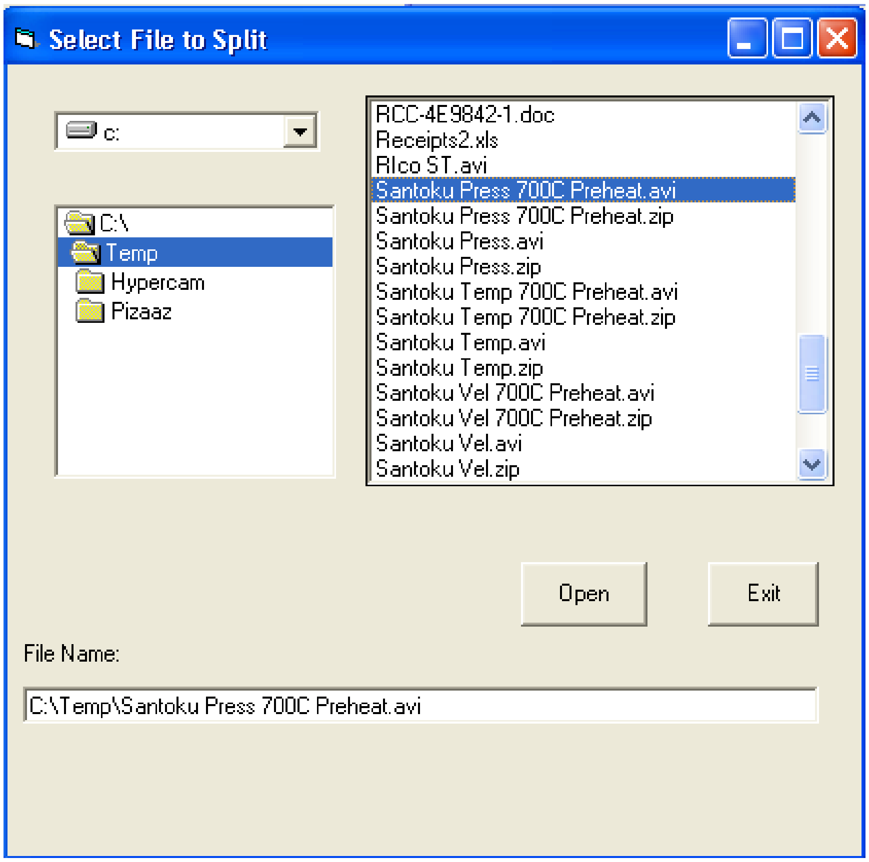
This can often happen if you are making a movie file in FLOWCast, and you have saved the data at small time steps. In these cases you can split the file into pieces, then email them individually. Then send along a copy of FileSplitter.exe.

When the email recipient receives all the pieces, they can run the program to recombine the pieces into the entire file.

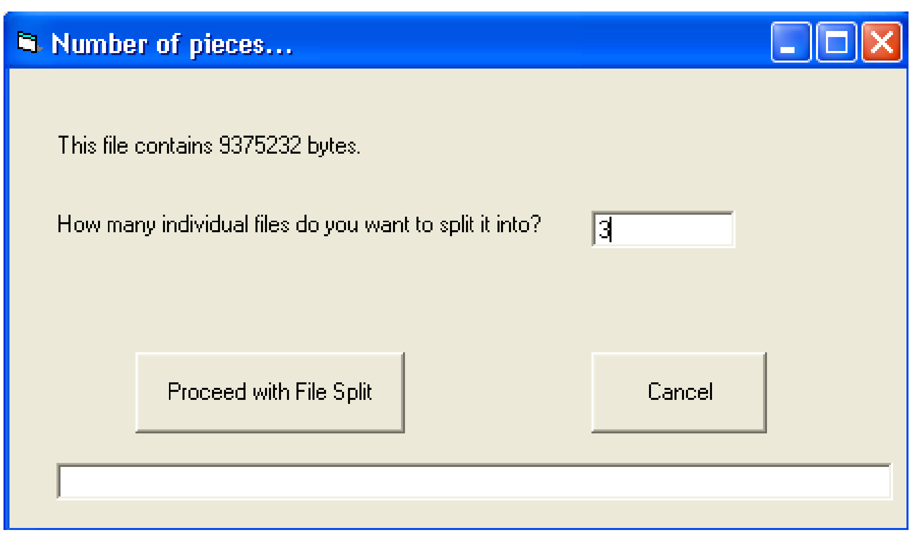
To run the utility, load a model, then select Tools…Split Files for Emailing. The system displays a screen similar to this:



If you will be sending the files, click on Split a large file into smaller pieces. You should then see a screen similar to that shown on the next page:



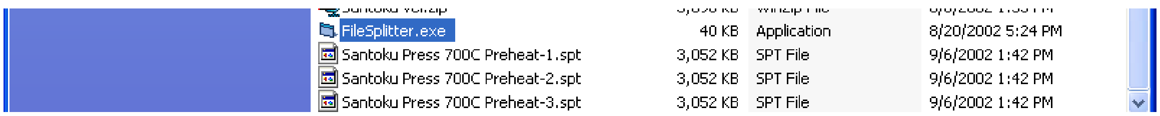
In this example we have chosen an AVI movie file created by FLOWCast that is almost 10 Mb in size. Many email systems cannot handle such a large file, so we can split it using FileSplitter. Highlight the file name with the mouse and click on Open.



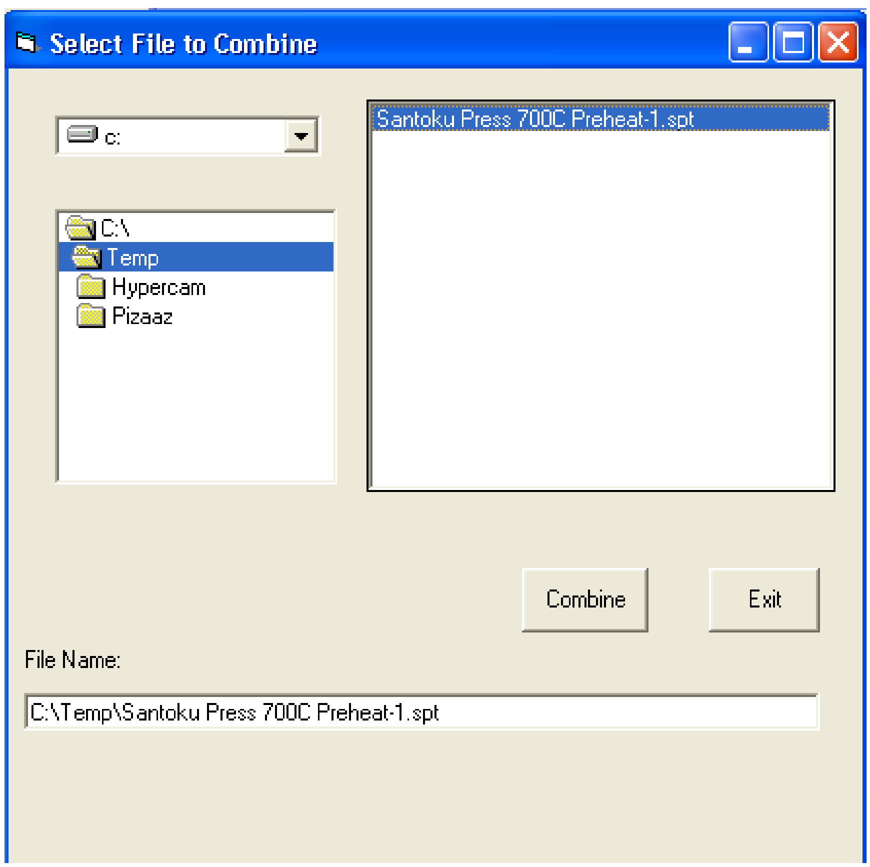
The system will ask how many files you want to break the large file into. Type the desired number into the box and click on Proceed with File Split. The system will then create the desired number of sub-files.

If the process is successful, you should see a response similar to this:

This section of a directory listing shows the three new files that have been created. They will have the name of the original file, but will have **–X.spt** added to each file name, where X stands for the sub-file number.

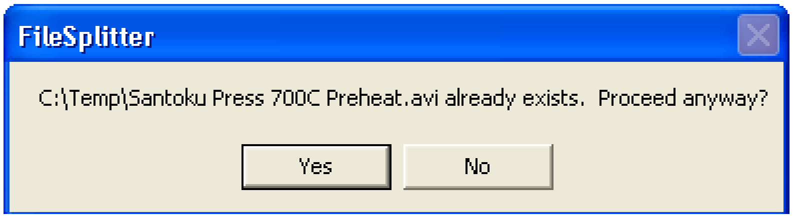


Once the recipient has received all the files, they can recombine the files into one by running FileSplitter at their end. This time they would select Recombine file pieces into the Original File from the choices. They should see a screen similar to this:



The file listing will only show the first file of each set. If there is more than one file, highlight the desired file and click on the Combine button.

If the full file already exists, you will see a message similar to this:



If you don't want to overwrite the existing file, choose No. You will then need to rename the original file before proceeding.

Once the recombination process is complete, you should see the following:

When you click on ok, you will be returned to SOLIDCast.

This utility can be used for any type of file that you need to make smaller for transport, either via email, a network or even floppy disks! Feel free to use it for purposes other than solidification modeling.

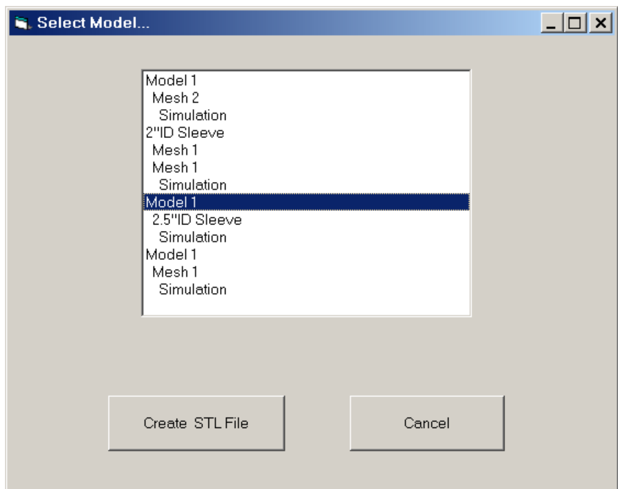
Note that the utility program, FileSplitter.exe, can be run as a stand-alone program. If you send split files to someone, also send them a copy of FileSplitter.exe. They can run the program by double-clicking on the file name, or they can use the Windows Run command and enter FileSplitter.exe in the blank. This means you can send split files to anyone, not just those who are running SOLIDCast.

UTILITY: STL From Model

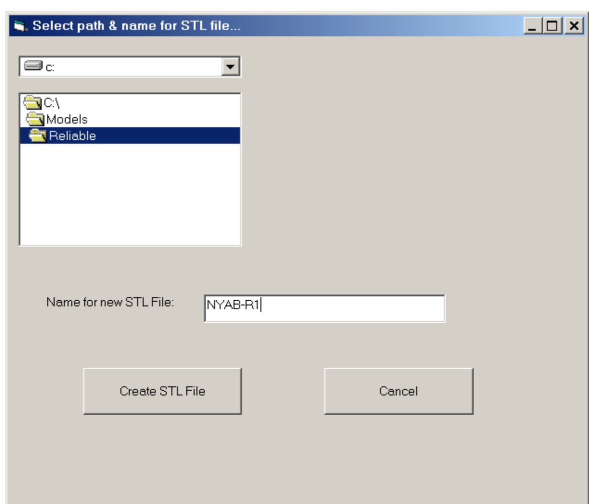
This utility creates a SINGLE STL FILE from a SOLIDCast model. The STL file includes ALL of the shapes in the model.

This feature was requested as an export feature from SOLIDCast, so that shapes which are built in SOLIDCast (for example, gating and risers) could be exported to an external CAD system for adding dimensions for a pattern shop.

To run the utility, load a model, then click Tools…STL From Model. You will see the Project Tree displayed. Use the cursor to highlight the model that you want to export in STL format:



Now press the Create STL File button. The system will then ask you to enter a name for the new STL file as follows:



You can now select a folder where the new file should go, and a name for the new file. Enter this information and click on the button labeled Create STL File. The new file will be created.

NOTES:

1. This will create an STL file INCLUDING ALL SHAPES IN A MODEL. This means that if you have sleeves or chills in the model, they will also show up in the STL file.

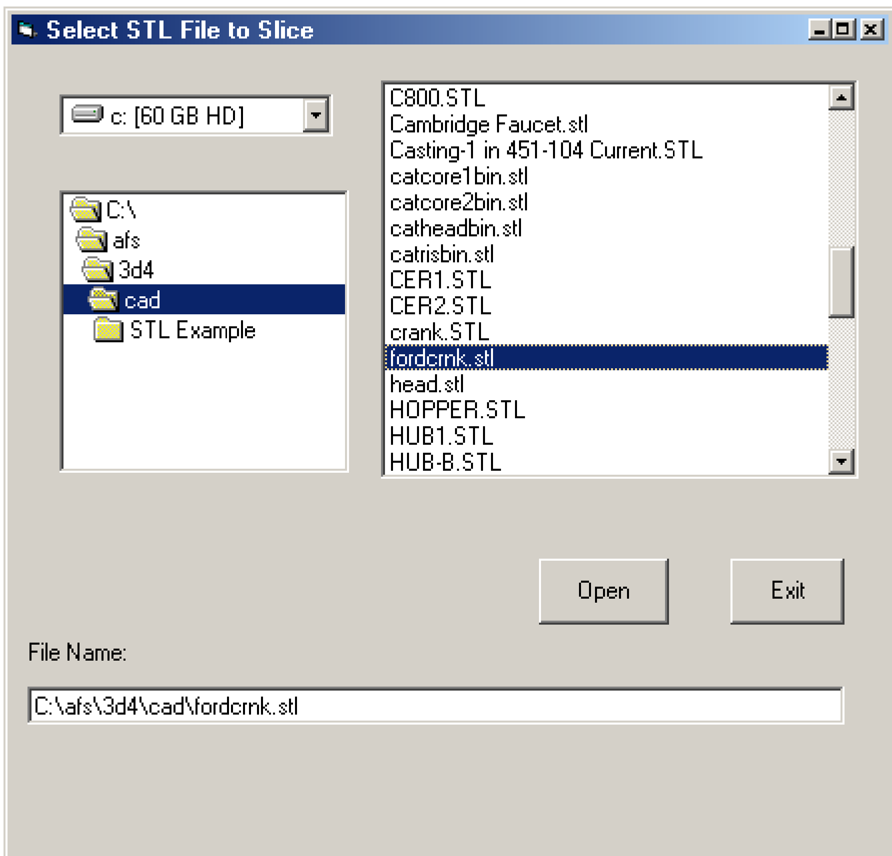
2. THE STL FILE THAT RESULTS FROM THIS OPERATION IS NOT NECESSARILY ONE THAT CAN BE USED AS A SHAPE IN SOLIDCAST FOR MODELING. IF ANY SHAPES IN A MODEL OVERLAP, THE SURFACES IN THE OVERLAP REGION WILL EXIST IN THE STL FILE. IF YOU IMPORT THIS STL FILE INTO SOLIDCAST, THESE INTERNAL REGIONS WILL BE INTERPRETED AS HOLES (VOID AREAS). THIS PROGRAM DOES NOT TRIM THE SURFACES IN THE STL FILE.

3. One popular use of this utility is to create an STL file of the casting and rigging of an investment casting, for use with the Create Shell Around STL Shape utility.

UTILITY: STL Slice

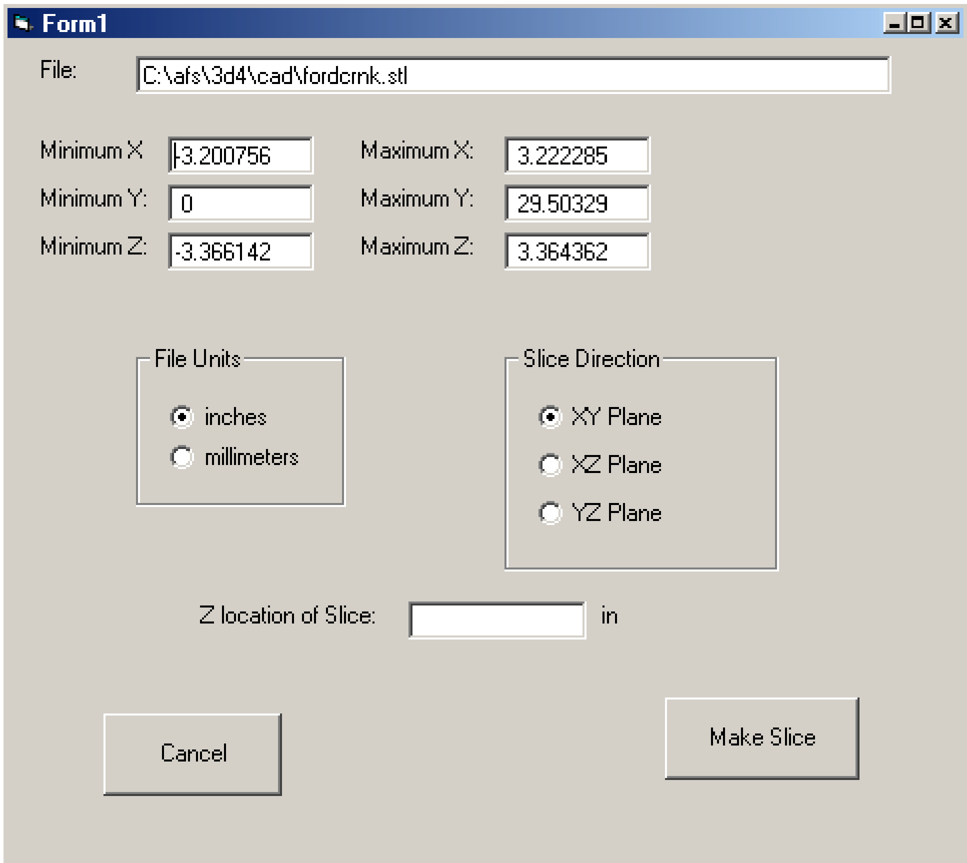
This utility is used to cut a slice through an STL file, then save that slice either in AFSCad or DXF format. This slice can then be imported into SOLIDCast for rotation or extrusion, or brought into AFSCad or another 2D CAD package. A typical use for this utility would be to take a slice through a file at the gate level. The section can then be viewed to find out just how big the ingate needs to be to make full contact. At Finite Solutions Inc, we often use this utility to troubleshoot problems with STL files, such as gaps or overlapping surfaces. By looking at 2-D slices, gaps and overlaps become very obvious.

To use the utility, load a model, then select Tools…STL Slice. The system will display the following:

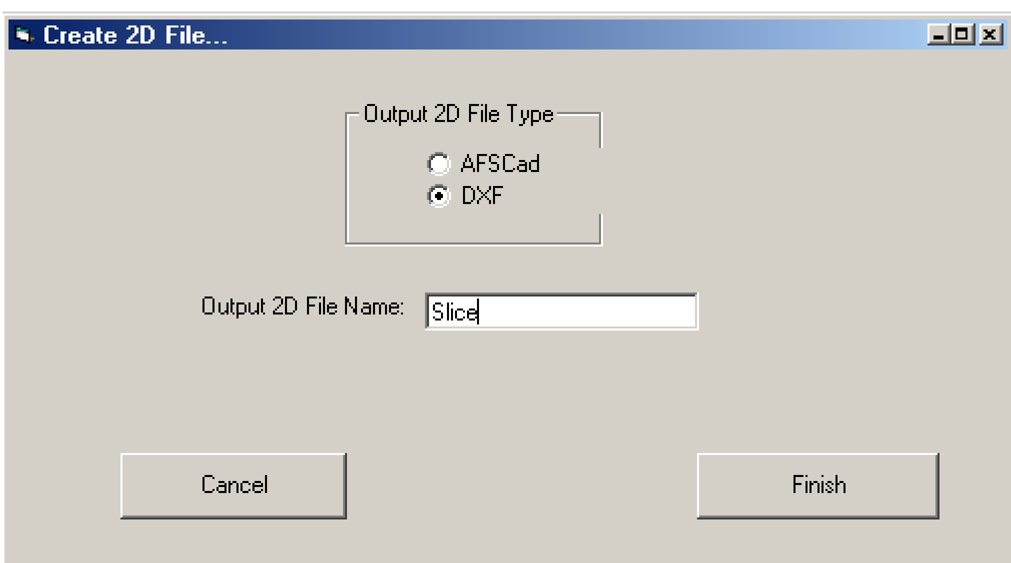


You can browse your hard drive to find the file. Once the file is located, highlight the file name and click on the open button. A window similar to that shown on the next page will be displayed.

This window displays the properties of the STL file, including size and position in the X, Y and Z directions. You can select units, direction that the slice be taken, and the slice position in that direction.



Once you enter the location of the slice, click on the Make Slice button. The utility will do its calculations and then let you enter the name for the file that will be created. Note that you can output the slice in either AFSCad or DXF formats.



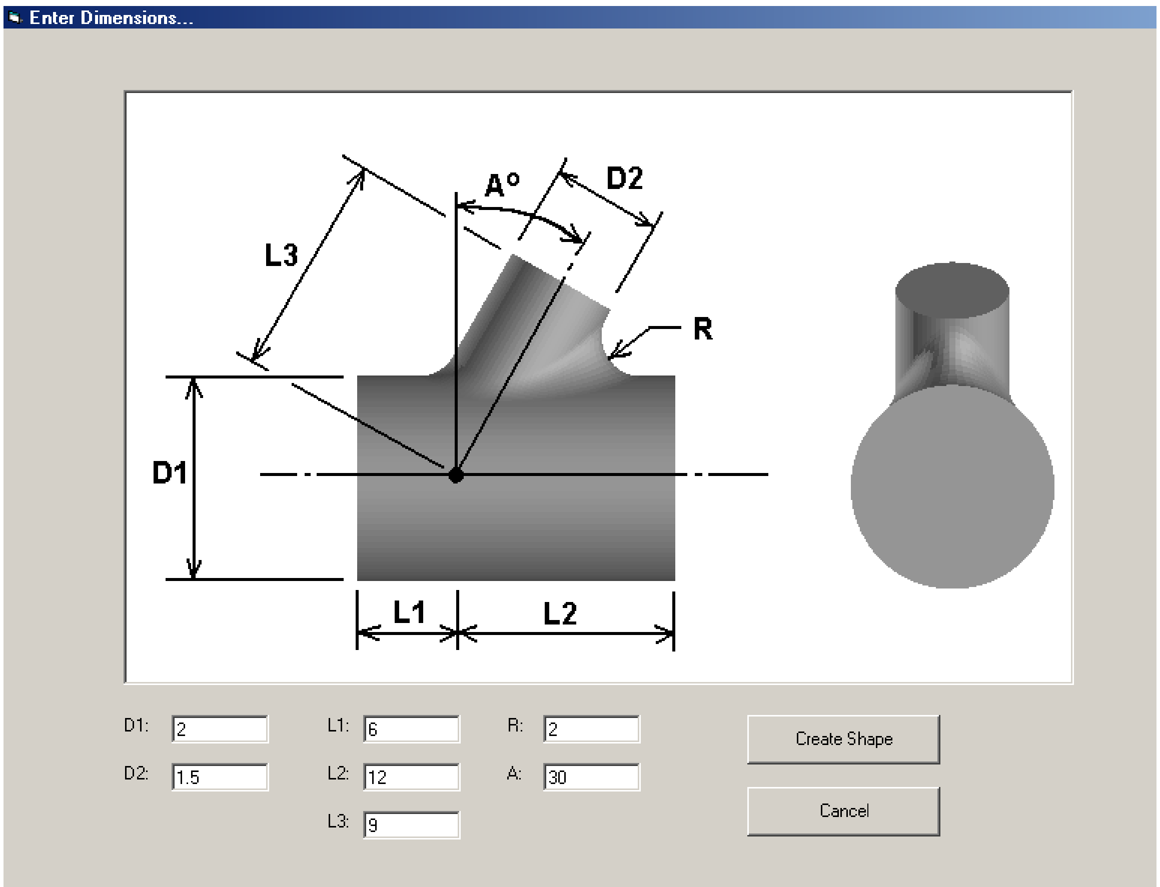
Once you have entered the file name, click on the Finish button. After the slice file is created you should see the following message:

The slice file will be created in the same folder as the original STL file. Click OK to return to the Model Building screen.

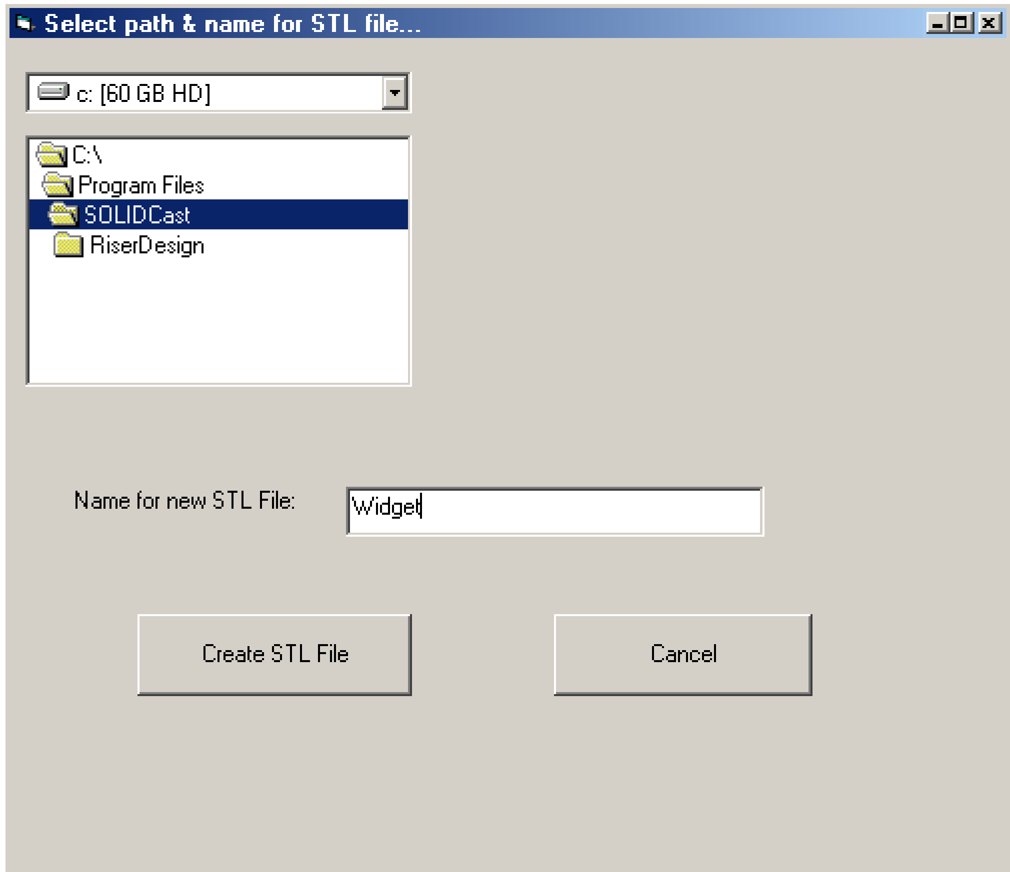
UTILITY: STL of Intersecting Cylinders

This utility is used create a blended shape composed of intersecting cylinders. The new shape is saved in Binary STL format and can be imported directly into SOLIDCast. The utility automatically creates a 3D fillet between the cylinders, based on your input parameters.

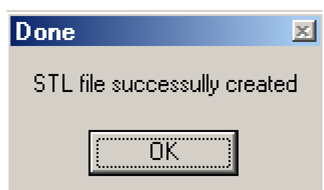
To start the utility, load a model, then select Tools…STL of intersecting cylinders. The system displays a screen similar to this:



The required dimensions are displayed in the diagram, so it is a simple task to fill in the blanks below the diagram, then click on the Create Shape button. A File Window will be displayed, allowing you to select a file name to save the new shape under. This file will be a Binary STL file. A sample window is shown on the next page.

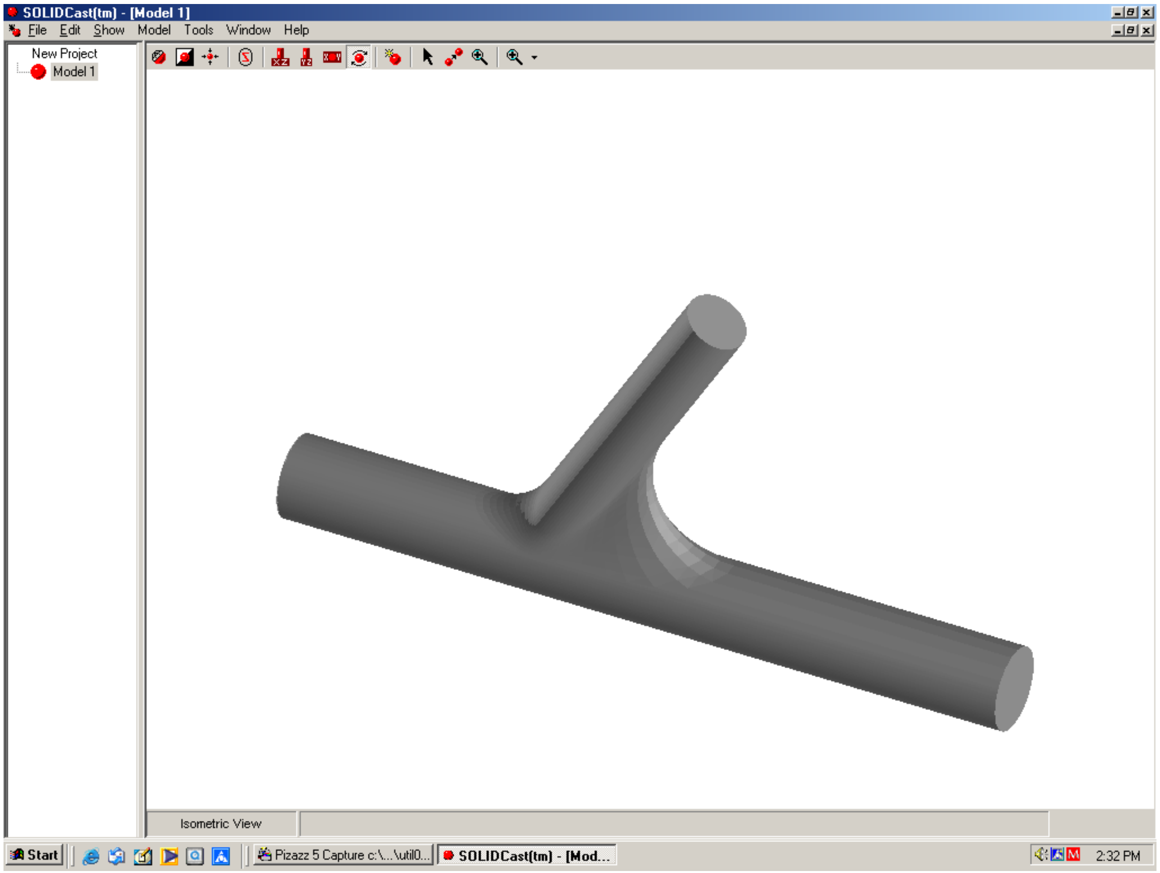


Type the file name into the line, then click on the Create STL File button. If all goes well you should receive the following message:



This STL file can then be imported into SOLIDCast in the same way as any other STL file. Click OK to return to the Model Builder screen.

An example STL file created with this utility and loaded into the Model Builder is shown on the next page.

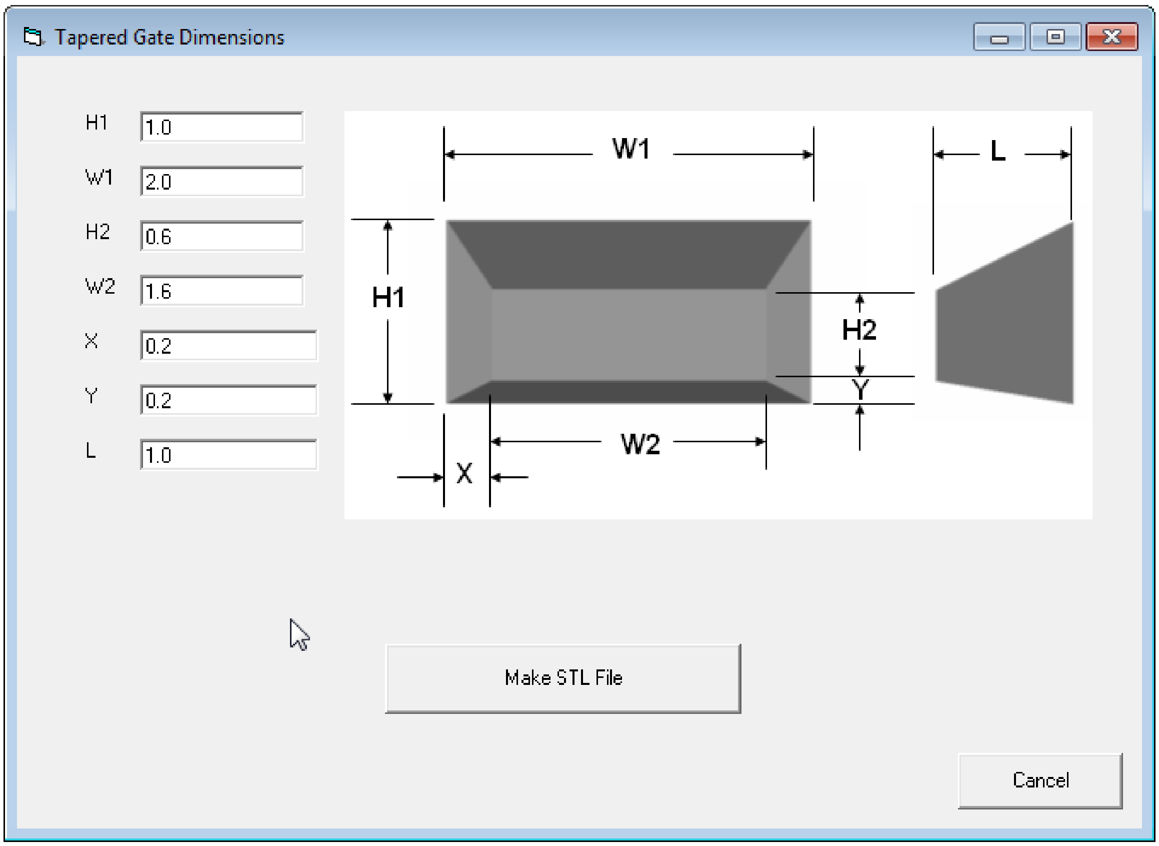


This utility can be used to help construct pieces of manifolds, etc. that have blended or filleted connections. To create a cored section, simply run the utility twice, adjusting for inside and outside dimensions. The core section should then be imported into SOLIDCast with a smaller priority number(closer to 1) so that when meshing occurs, the core removes metal.

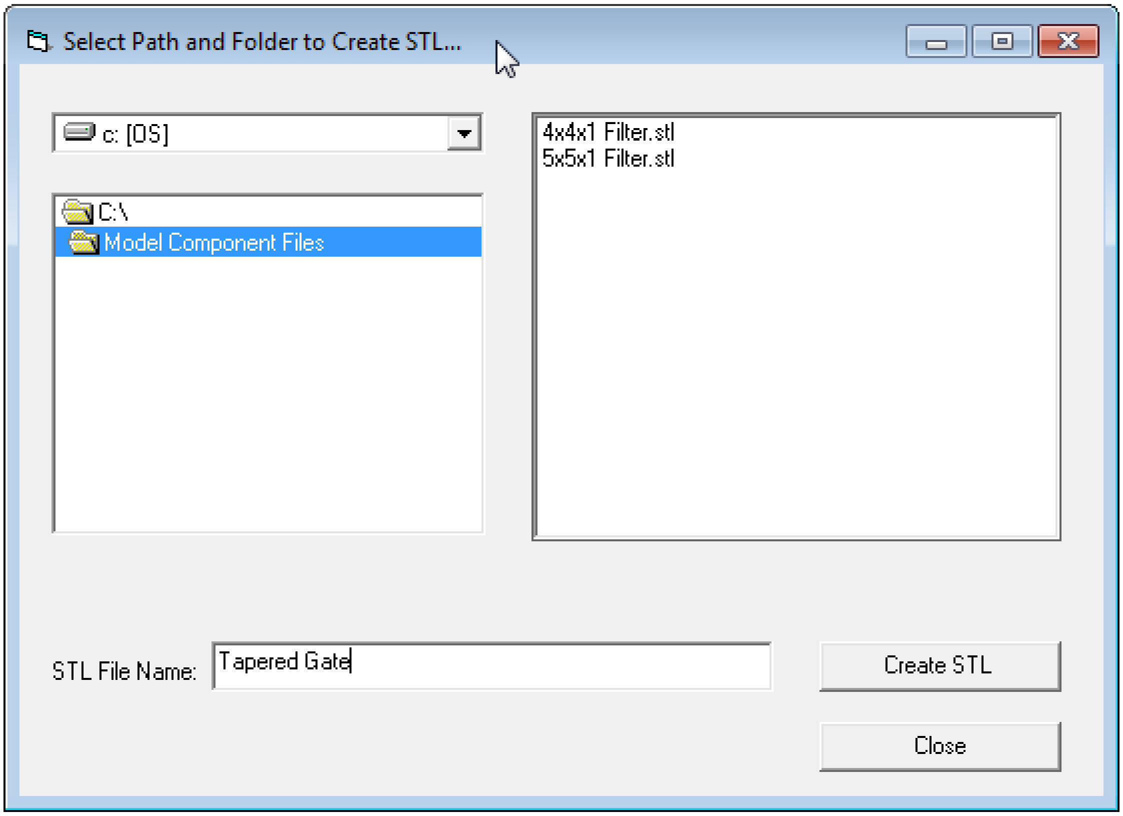
UTILITY: STL of Tapered Gate

In the investment casting process, the use of tapered gates is often done, to create a connection from the casting to the tree that increases in size, promoting directional solidification from the casting, through the gate and into the main body of the tree. The STL of Tapered Gate Utility can be used to create a 3D shape of a tapered gate in STL format, which can be added into a SOLIDCast model.

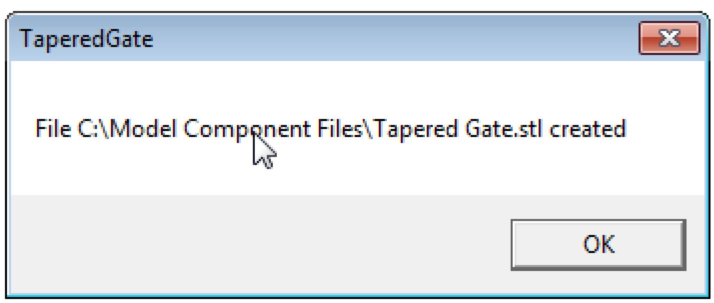
To start the utility, you must first have a model loaded. Then go to Tools…STL of Tapered Gate. You will see the following form, giving you a schematic view of the gate shape, and an empty spaces where you can enter appropriate dimensions for your gate:



Once you have entered all values, click on Make STL File. You will see a window like that on the next page, allowing you to name the new file.



Once you have named the file, click on Create STL to make the gate file. Once the file has been created you will see a message similar to this:



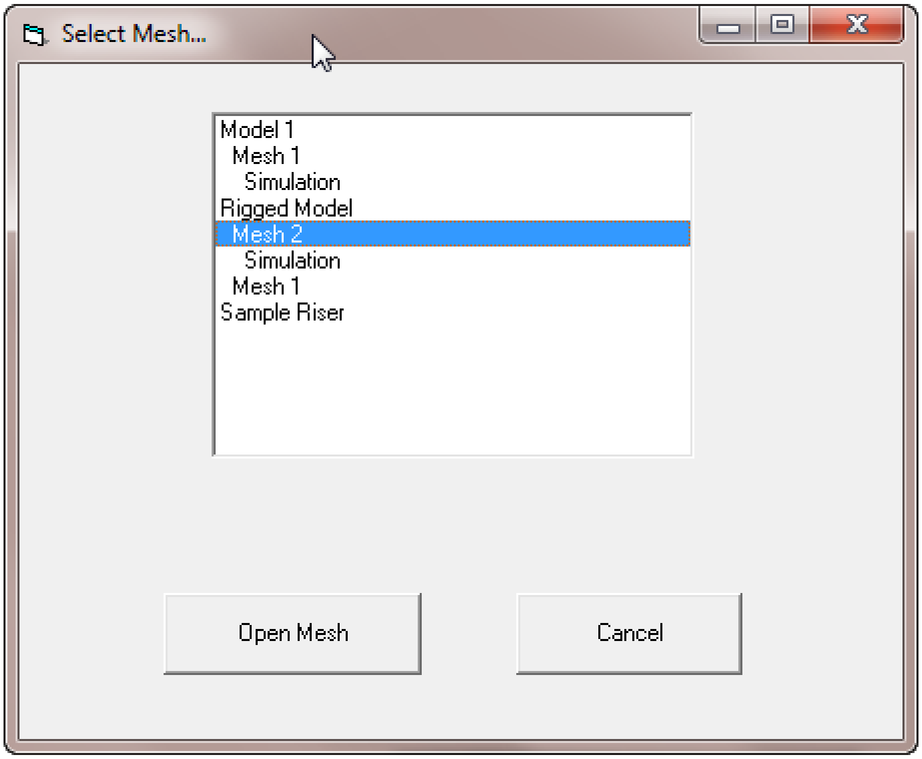
Click OK to return to the Model Builder. Here is what the gate created in this example would look like in the Model Builder:

UTILITY: Two Stage Pour

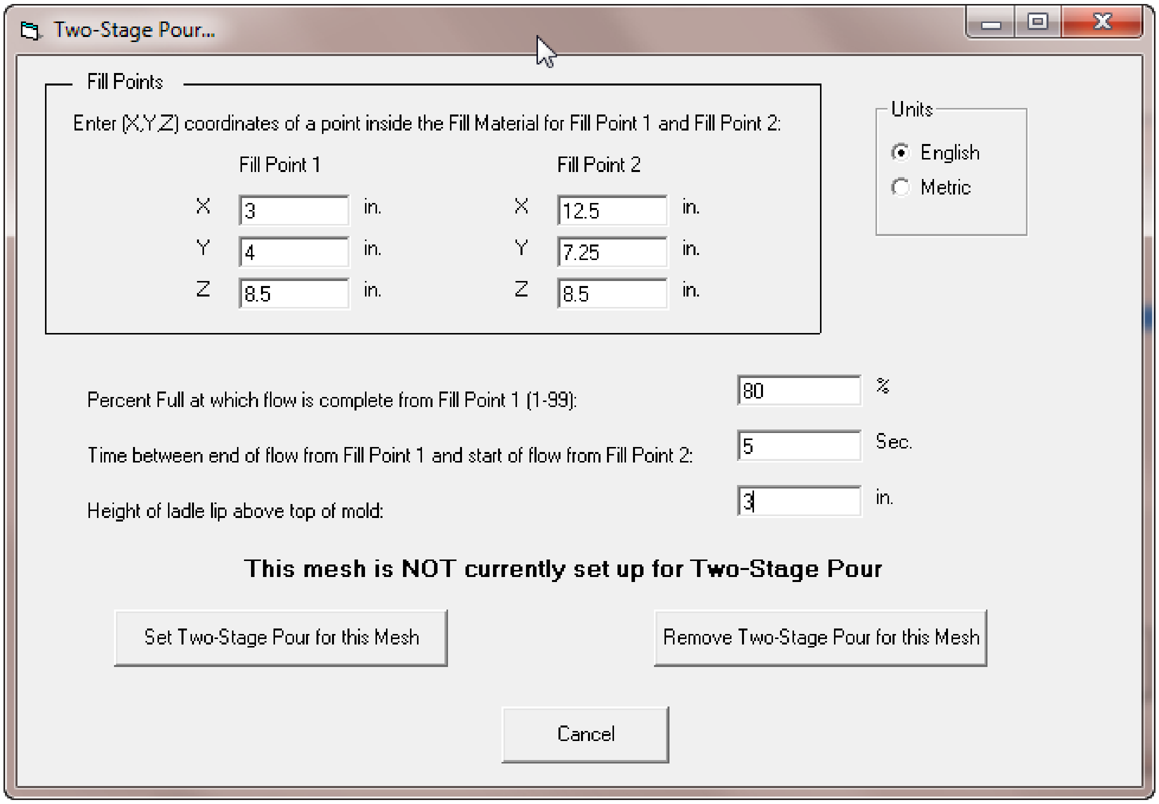
In this utility, the pour starts at Fill Point 1 and continues until a given % full is reached. Then flow stops from Fill Point 1. There is a pause of a given number of seconds, after which pouring starts from Fill Point 2 and continues until filling is complete. This utility can only be used with the FLOWCast filling algorithm.

This would most often be used to simulate a 'hot topping', where you would initially pour a mold through the sprue, then move the ladle over the top of an open riser and top up the riser with hot metal.

To use this utility, load a model, then click Tools…Two Stage Pour. You should see a reproduction of the Project Tree, something like this:



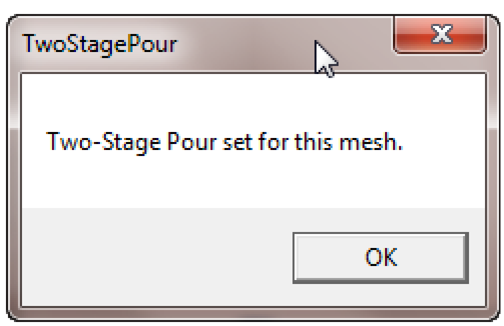
Highlight the name of the mesh you want to work with, then click on Open Mesh. The system will analyze the mesh, then present you with a setup screen, similar to that shown on the next page.



You need to enter the X, Y, Z coordinates of the first and second filling points. These points must lie inside of pieces of Fill Material in your model.

Next enter the percentage of the full pour at which you stop filling from the first fill point. Then enter the time lag, in seconds, between the end of filling at the first point and the beginning of filling at the second point. Also enter how high the lip of the ladle is above the top of the mold.

Once these entries have been made, click the button that says Set Two-Stage Pour for this Mesh. This will add the necessary information to run the two stage pour simulation to the mesh file.

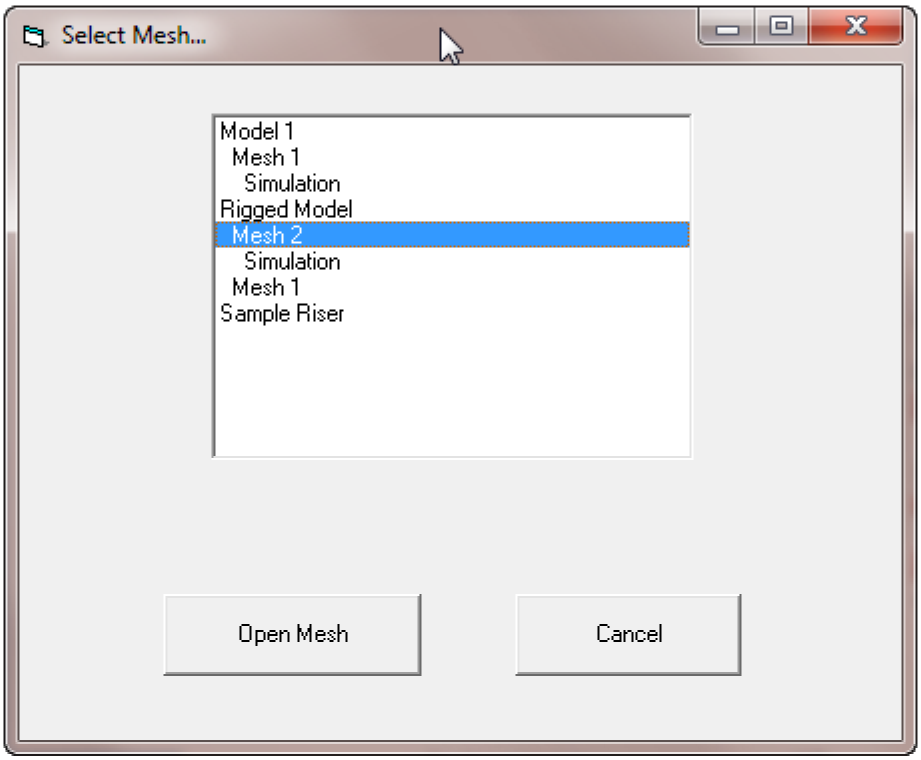


Click OK to return to the Model Building screen. Now, when you run the FLOWCast filling simulation, the system will automatically apply the filling from two points at two times.

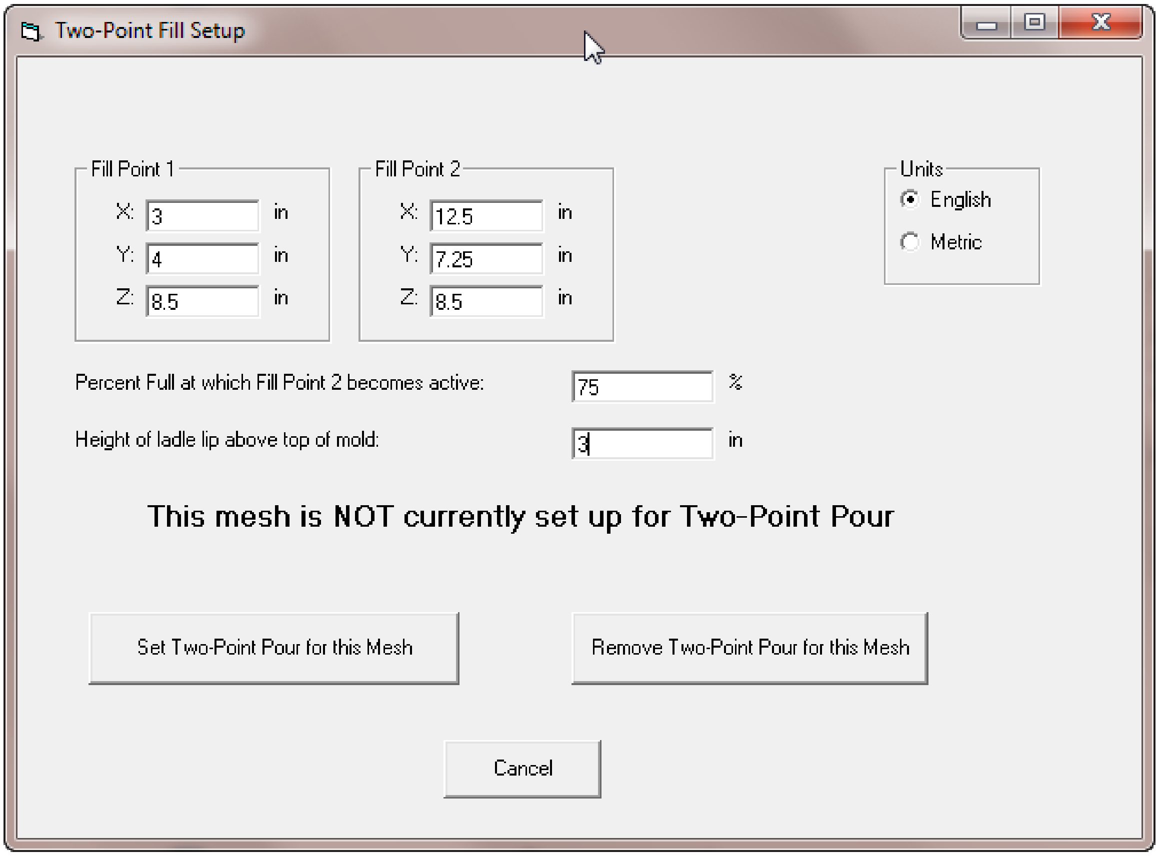
UTILITY: Two Point Pour

In this utility, pouring starts at Fill Point 1 until a given % full is reached. At that point, flow also starts from Fill Point 2. Flow continues from both Fill Point 1 and Fill Point 2 until the end of filling. This utility can only be used with the FLOWCast filling algorithm.

To use this utility, load a model, then go to Tools…Two Point Pour. The system will display a Project Tree, similar to this:



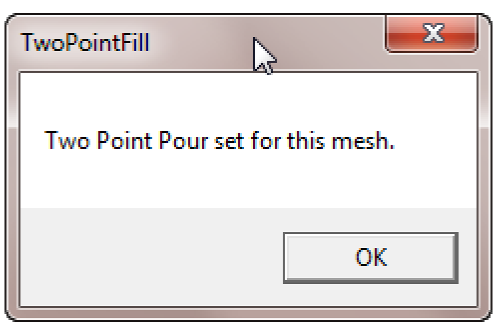
Highlight the name of the mesh you want to work with, then click Open Mesh. The system will analyze the mesh, then present you with an entry screen, similar to that shown on the next page.



You need to enter the X, Y, Z coordinates of the first and second filling points. These points must lie inside of pieces of Fill Material in your model.

Next enter the percentage of the full pour at which you beginning filling at the second point in addition to the first point. Also enter how high the lip of the ladle is above the top of the mold.

Once these entries have been made, click the button that says Set Two-Point Pour for this Mesh. This will add the necessary information to run the two point pour simulation to the mesh file.



Click OK to return to the Model Building screen. Now, when you run the FLOWCast filling simulation, the system will automatically apply the filling scenario.