UNIT 37: Making Movies

SOLIDCast allows you to create movies from the results of running simulations. These movies are in the form of WMV files. WMV files are standard animation files that can be viewed on any Windows system, as long as the Windows Media Player program is installed. Media Player is a Windows system accessory and is normally found in most Windows installations. A current version of the Media Player can be downloaded from the Microsoft web site, http://www.microsoft.com.

There are several types of movie files that can be created in SOLIDCast, as noted below. Each type is then described in a following section.

Iso-Surface Movies – Each frame of the movie is an iso-surface plot. The progression works best on time-related data. For example, an iso-surface movie of solidification time would show the progression of solidification in a casting. The surfaces would show what metal had not yet solidified at that point in time.

Cut-Plane Movies – Each frame of the movie is a cut-plane plot. A cut-plane plot is a 2D plot, taken from a slice of the 3D model. Again, this progression works best on time-related data. This movie shows only the 2D cut plane, and not the full 3D model.

CastPic Movies – Each frame shows a CastPic plot. This is 3D and in color, and can be cut into up to three directions to show internal features. The plot can be set to 'go gray' when outside the plot area, or to 'fade to blue'. This can be quite effective in showing a casting 'cool down' after solidification.

CastScan Movies – CastScan movies come in two flavors, Progressive and Rotating. Progressive CastScan movies plot a different range of data in each frame, to show, for example, the progression of solidification. Rotating movies, on the other hand, plot the same data in each frame, but the view is rotated in each frame, to give the impression that the casting is moving about, as on a rotating turntable. You can specify many colored layers, which give a very attractive visual effect.

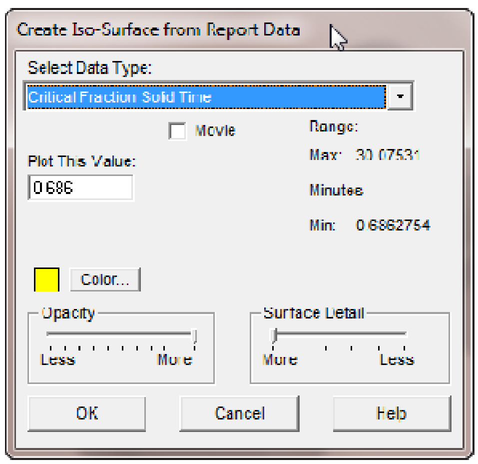
ISO-SURFACE MOVIES

Iso-surface movies are a series of iso-surfaces at progressive values in a casting model. Each iso-surface plotted becomes a single frame or picture in the movie. At the end of the movie creation function, all individual images are included in a single WMV file.

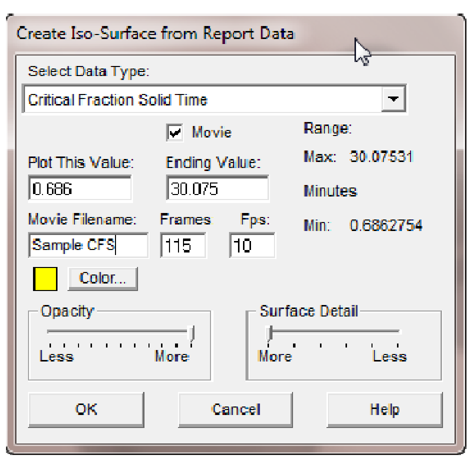
Creating an Iso-surface movie is similar to plotting a single Iso-surface plot (see the section on Iso-surface plotting). Load your simulation. Then rotate the casting to the angle want displayed for the movie. Now, double-click on the simulation icon on the Project Tree to display the Simulation Summary box, then close the Simulation Summary box and select Simulation… Plot Iso-Surface from the main menu. Select the data item that you want to plot.

For Iso-surface movies, the type of data that is plotted is usually either Critical Fraction Solid Time or Solidification Time, since both of these are time-based criteria and making a movie of progression of time makes the most intuitive sense.

If you select Critical Fraction Solid Time, for example, you will see a window like this:



Notice on this window that there is a small box labeled Movie. If you click this box, some additional fields appear in the window as shown here:



Now there are items labeled “Ending Value”, “Movie Filename”, “Frames” and “Fps”.

“Plot This Value” indicates the value of the iso-surface plot in the first frame of the movie.

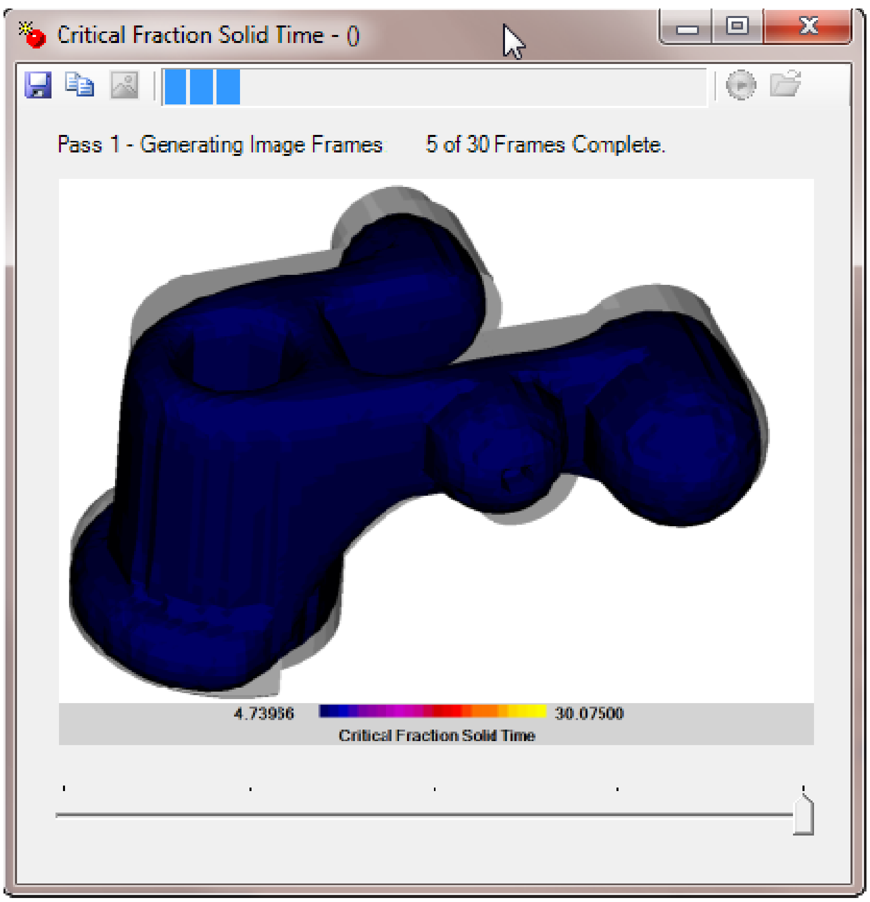
“Ending Value” indicates the value of the iso-surface to be plotted in the final frame of the movie. Typically you might want this to be slightly less than the Maximum value, so that the final area to solidify will appear with a definite shape in the last movie frame.

“Movie Filename” refers to the name of the WMV file that will contain this movie. In this example, the movie file will be called Sample CFS.WMV. This file will be placed into the current folder where the System Parameter for Import Files is set.

“Frames” refers to the number of individual pictures that will make up the movie. The more frames that are specified, the smaller will be the change from one surface to the next, however, with more frames it will take longer for the system to make the movie and the size of the WMV file will be larger.

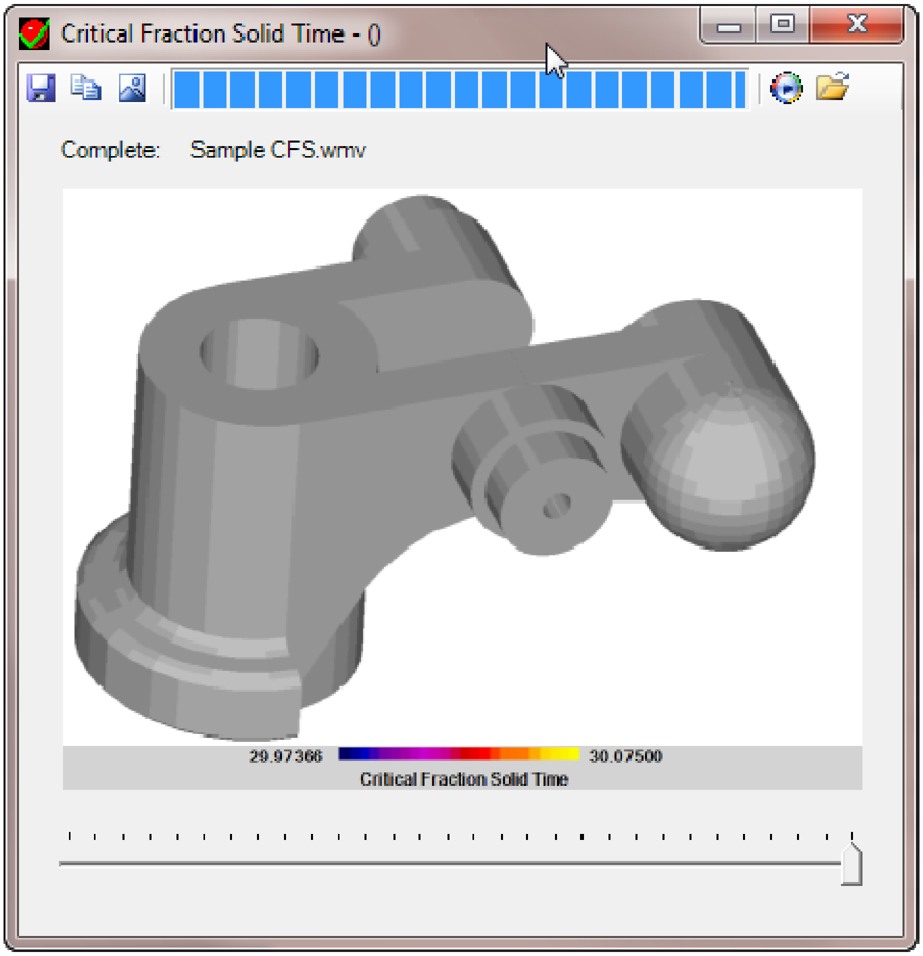
“Fps” stands for frames per second, and controls how fast the individual images are displayed.

After specifying the above information, click on OK. The system will display a window showing the progress of the movie creation. An example is shown here:



Note that for Iso-Surface movies, the only plot color available is a dark blue. Future versions of SOLIDCast will allow for other colors.

Once the movie is complete you should see a screen like this:



You can click on the icons to run the movie in the Windows Media Player or go to the movie folder. The normal procedure is to set your movie player to repeat, so you can see the movie several times without having to restart it manually each time. Once you have finished watching the movie, close the Media Player to return to SOLIDCast.

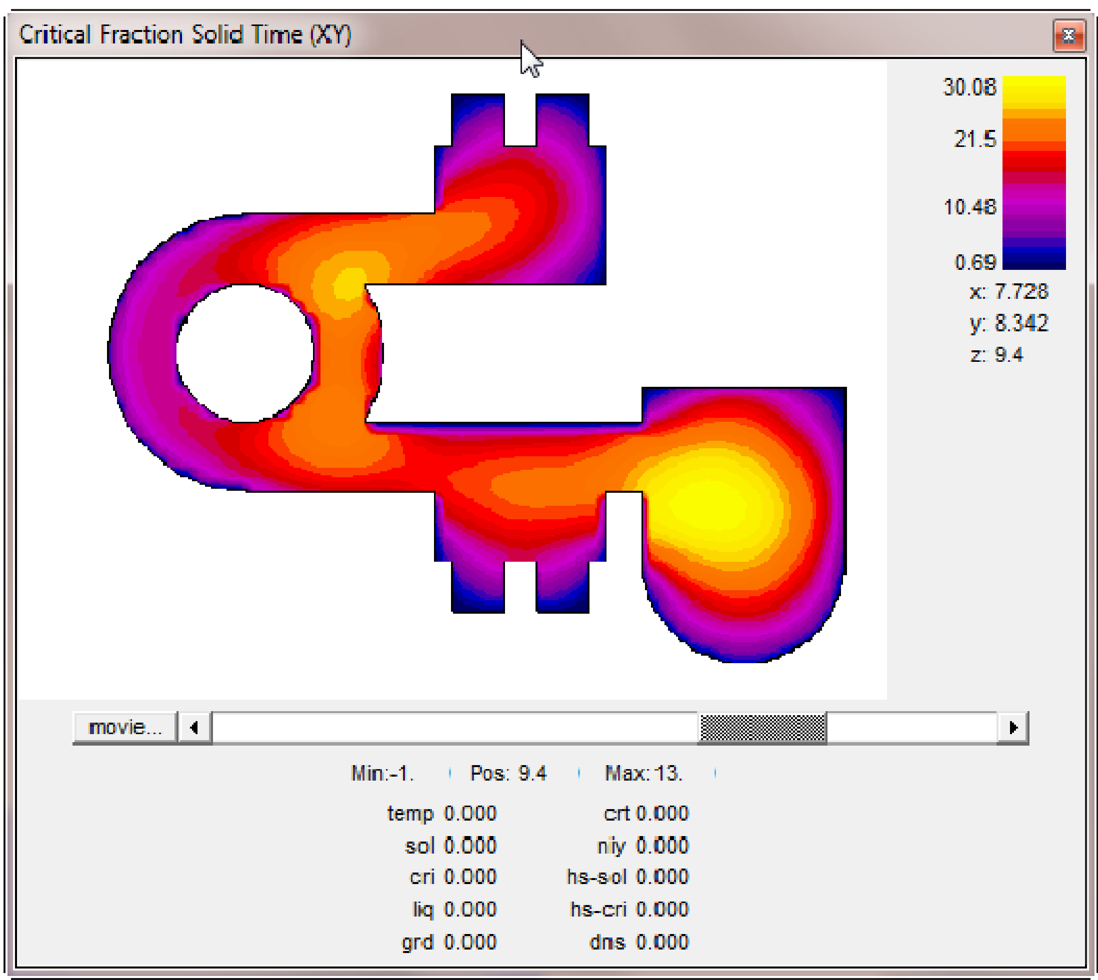
Note: While a movie is being created, you can still do other things, either in SOLIDCast or in any other software package. The movie creation process takes place in the background.

Any time that you want to later view the movie, or send it to another person to view, you just need to locate the WMV file that was created and double-click its icon to start viewing, or send the file as an email attachment to another person. When the other person receives the file, they can just double-click the icon to view the movie. It is not necessary to have SOLIDCast installed on a computer to view these movie files.

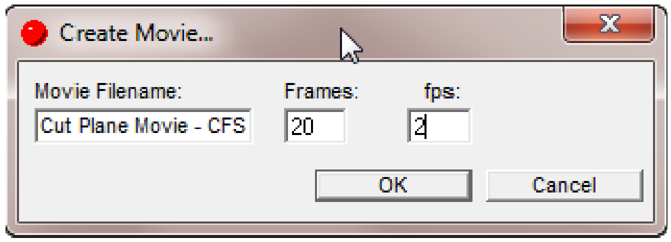
CUT-PLANE MOVIES

Cut-Plane movies show the progression of a selected type of data on a single 2D slice cut through a casting model. A series of images are created which show the progression of the chosen data item on the cut plane. At the end of the movie creation function, all individual images are included in a single WMV file.

Creating a Cut Plane movie involves first creating a Cut Plane plot on a chosen plane(see the section on Cut Plane plotting). A simulation must have been completed. Double-click on the simulation icon on the Project Tree to display the Simulation Summary box, then close the Simulation Summary box and select Simulation… Plot Cut Plane from the main menu. Select the data item that you want to plot, minimum and maximum plot values, and the direction of the cut plane. For Cut Plane movies, the type of data that is plotted is usually either Critical Fraction Solid Time or Solidification Time, since both of these are time-based criteria and making a movie of progression of time makes the most intuitive sense. Now click on OK to create a Cut Plane plot. You can now use the slider bar to position the Cut Plane where you want it to be in the model. As an example, you should see something like the following on the screen:



Notice on this window that there is a small box labeled Movie. If you click this box, a small window will appear as follows:



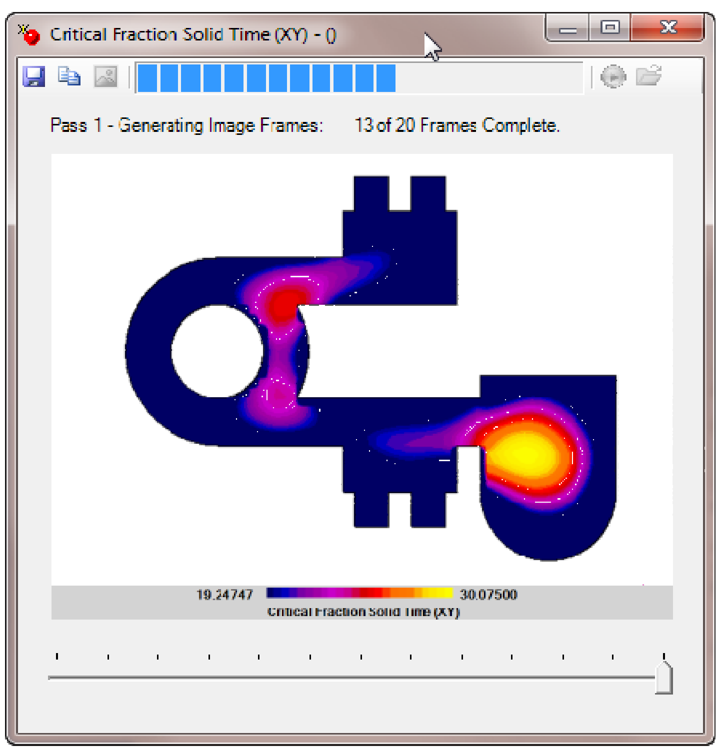
Notice there are items labeled “Movie Filename”, “Frames” and “fps”.

“Movie Filename” refers to the name of the WMV file that will contain this movie. In this example, the movie file will be called Cut Plane Movie - CFS.WMV. This file will be placed into the current folder where the System Parameter for Import Files is set.

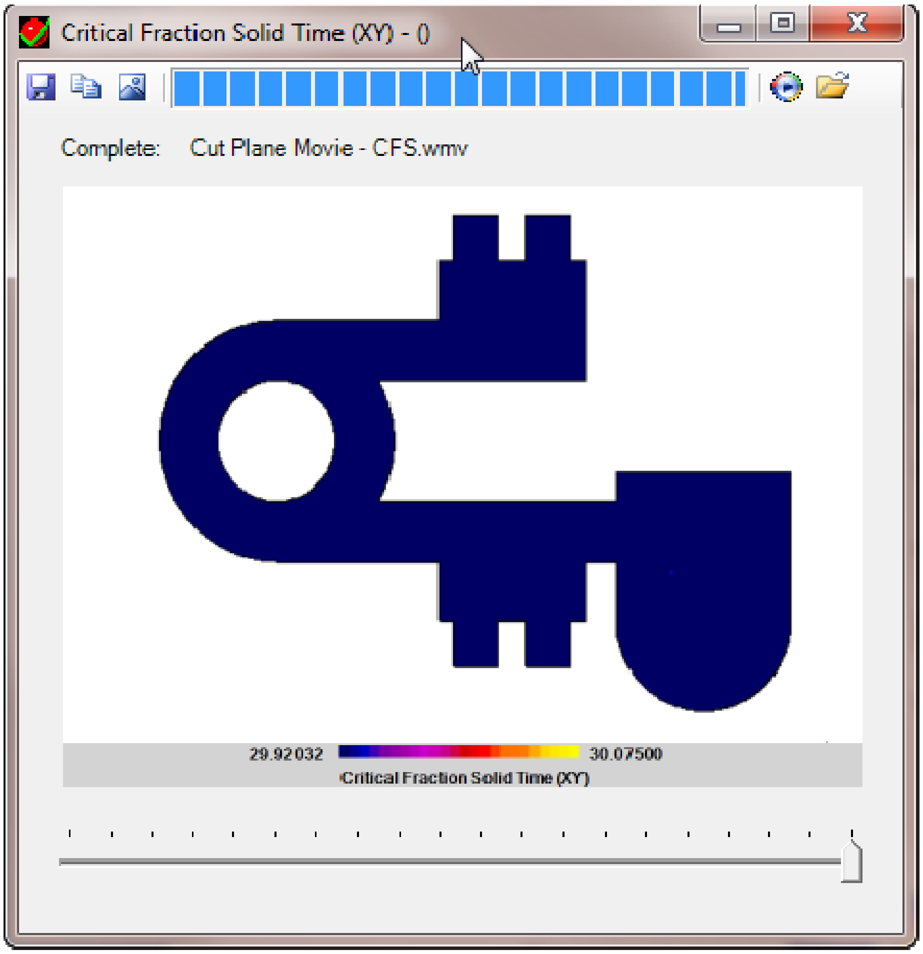
“Frames” refers to the number of individual pictures that will make up the movie. The more frames that are specified, the smaller will be the change from one surface to the next, however, with more frames it will take longer for the system to make the movie and the size of the WMV file will be larger.

“fps” stands for frames per second, and controls how fast the individual images are displayed.

After specifying the above information, click on OK. The system will display a window showing the progress of the movie creation. An example is shown here:



Once the movie is complete you should see a screen like this:



You can click on the icons to run the movie in the Windows Media Player or go to the movie folder. The normal procedure is to set your movie player to repeat, so you can see the movie several times without having to restart it manually each time. Once you have finished watching the movie, close the Media Player to return to SOLIDCast.

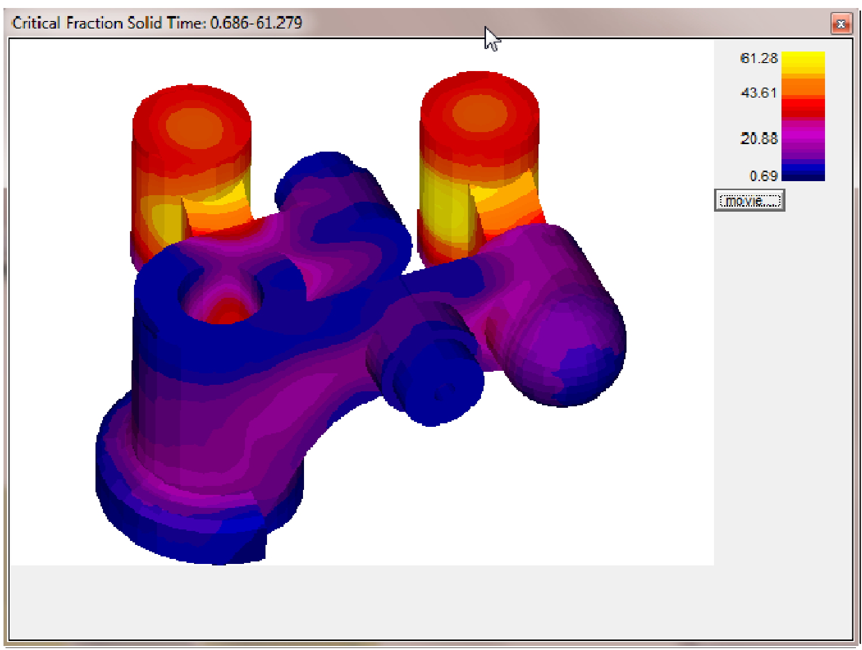
Note: While a movie is being created, you can still do other things, either in SOLIDCast or in any other software package. The movie creation process takes place in the background.

Any time that you want to later view the movie, or send it to another person to view, you just need to locate the WMV file that was created and double-click its icon to start viewing, or send the file as an email attachment to another person. When the other person receives the file, they can just double-click the icon to view the movie. It is not necessary to have SOLIDCast installed on a computer to view these movie files.

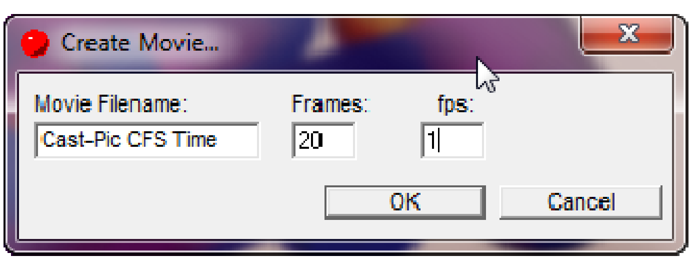
CAST PIC MOVIES

CastPic movies show the progression of a selected type of data in the form of CastPic images of a casting model. A series of CastPic images are created which show the progression of the chosen data item on the model surface. At the end of the movie creation function, all individual images are included in a single WMV file.

Creating a CastPic movie involves first creating a CastPic plot of a model at a given angle of rotation. (See the section on CastPic plotting). A simulation must have been completed. Double-click on the simulation icon on the Project Tree to display the Simulation Summary box, then close the Simulation Summary box and select Simulation… CASTPIC Plot from the main menu. Select the data item that you want to plot, minimum and maximum plot values, location of cut plane (if any), whether to plot data outside the range and the Plot Detail number. For CastPic movies, the type of data that is plotted is usually either Critical Fraction Solid Time or Solidification Time, since both of these are time-based criteria and making a movie of progression of time makes the most intuitive sense. Now click on OK to create a CastPic plot. As an example, you should see something like the following on the screen:



Notice on this window that there is a small box labeled Movie. If you click this box, a small window will appear as follows:



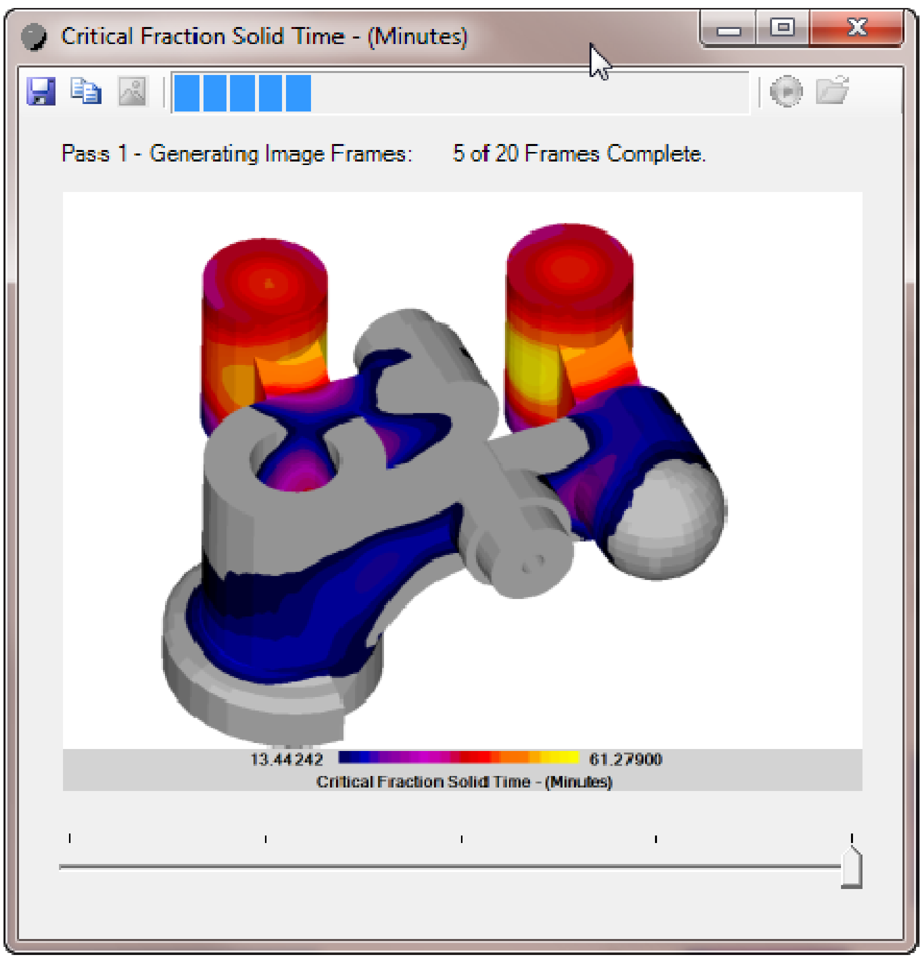
Notice there are items labeled “Movie Filename”, “Frames” and “fps”.

“Movie Filename” refers to the name of the WMV file that will contain this movie. In this example, the movie file will be called Cast-Pic CFS Time.WMV. This file will be placed into the current folder where the System Parameter for Import Files is set.

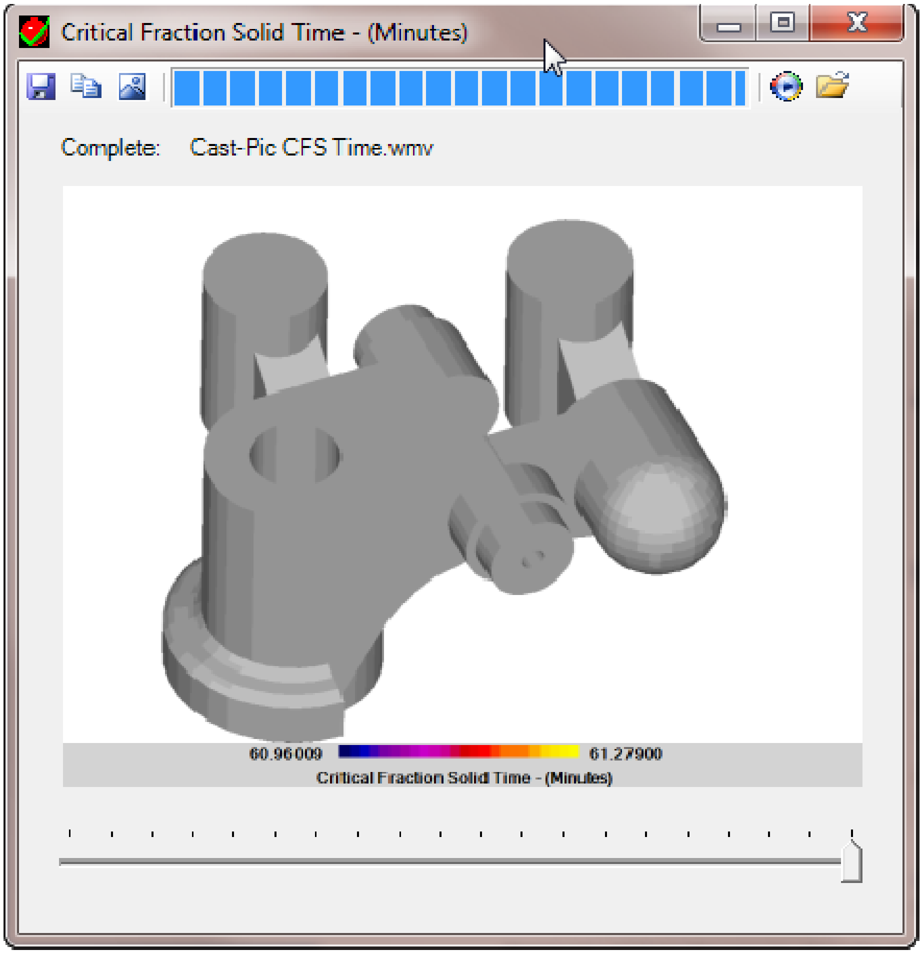
“Frames” refers to the number of individual pictures that will make up the movie. The more frames that are specified, the smaller will be the change from one surface to the next, however, with more frames it will take longer for the system to make the movie and the size of the WMV file will be larger.

“fps” refers to the number of frames per second that will be shown when the WMV file is displayed. This controls the speed of the movie.

After specifying the above information, click on OK. The system will display a window showing the progress of the movie creation. An example is shown here:



Once the movie is complete you should see a screen like this:



You can click on the icons to run the movie in the Windows Media Player or go to the movie folder. The normal procedure is to set your movie player to repeat, so you can see the movie several times without having to restart it manually each time. Once you have finished watching the movie, close the Media Player to return to SOLIDCast.

Note: While a movie is being created, you can still do other things, either in SOLIDCast or in any other software package. The movie creation process takes place in the background.

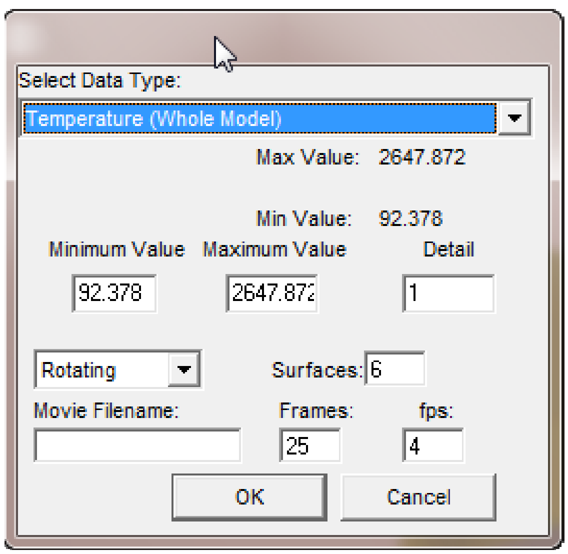
Any time that you want to later view the movie, or send it to another person to view, you just need to locate the WMV file that was created and double-click its icon to start viewing, or send the file as an email attachment to another person. When the other person receives the file, they can just double-click the icon to view the movie. It is not necessary to have SOLIDCast installed on a computer to view these movie files.

CAST SCAN MOVIES - Progressive

A Cast Scan movie combines elements of both a CastPic and Iso-Surface plots. In a Cast Scan movie, the casting background is created as a solid rendered and shaded casting picture as it is in a CastPic plot. The data inside the casting is created as one or more Iso-Surfaces. If there are multiple Iso-Surfaces specified, then the Iso-Surfaces are variably colored with the brighter colors representing the more critical data areas.

Cast Scan Movies can be created showing either a progression of data with the casting image in one orientation, or with the data plot limits fixed and the casting rotating in space. We will first discuss the Progressive type of plot, and the next section will deal with the Rotating type of plot.

To create a Progressive Cast Scan, first orient the model at the angle at which you want it to appear in the movie. The casting will remain at this angle in all frames of the movie. You must have completed a simulation prior to creating the Cast Scan movie. Double-click on the simulation icon on the Project Tree to display the Simulation Summary box, then close the Simulation Summary box and select Simulation… CASTSCAN Movie. A window similar to the following will appear:

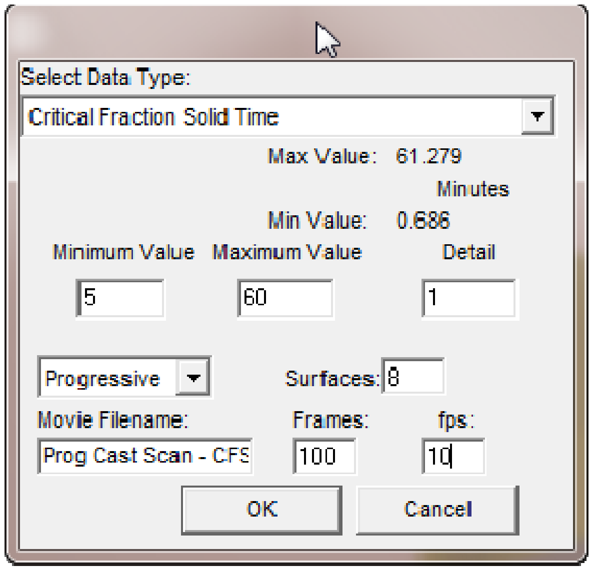


Note that when this window first appears, Temperature is the default data item. For a Progressive Cast Scan movie, the type of data that is plotted is usually either Critical Fraction Solid Time or Solidification Time, since both of these are time-based criteria and making a movie of progression of time makes the most intuitive sense. You can select a data item by clicking on the selection arrow next to Temperature (Whole Model) and then selecting either Critical Fraction Solid Time or Solidification Time.

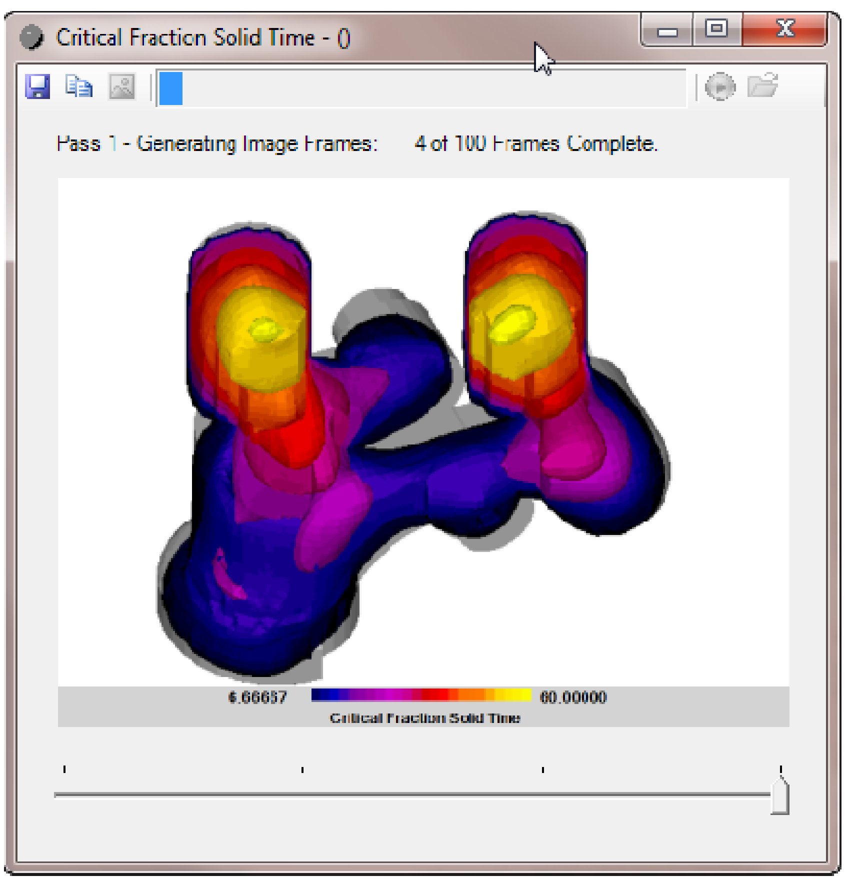
Note also that "Rotating" appears in a window. You can change this to Progressive by clicking on the selection arrow next to Rotating.

The number of Surfaces specifies how many Iso-Surfaces will be plotted within each image. This can vary from 1 to 24.

A typical setting for this window would be as follows:



After specifying the above information, click on OK. The system will display a window showing the progress of the movie creation. An example is shown here:



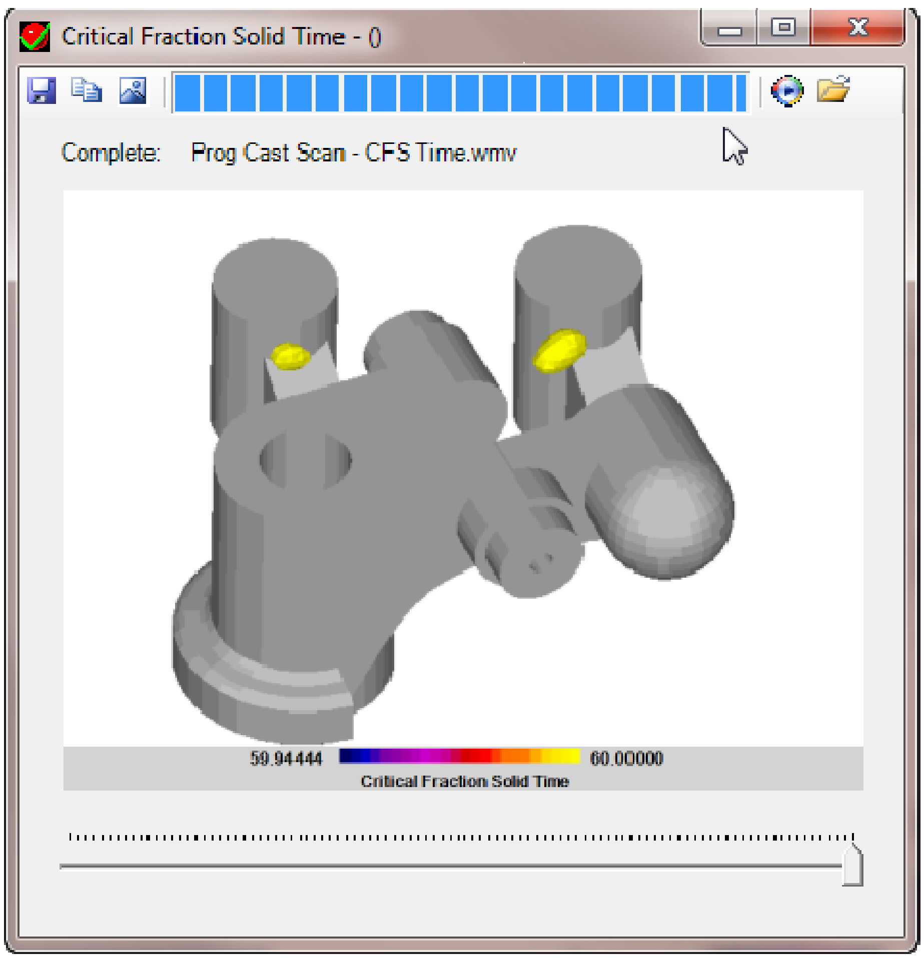
Note that when making Cast Scan movies, the time to create a movie can vary widely. Some of the items that affect the time to make a movie are as follows:

Number of Frames: The greater the number of frames, the longer the time to make the movie.

Surfaces: The more surfaces, the longer the time.

Detail: The higher the detail, the longer the time. (Note that higher detail produces a larger movie when using the default size in the Media Player).

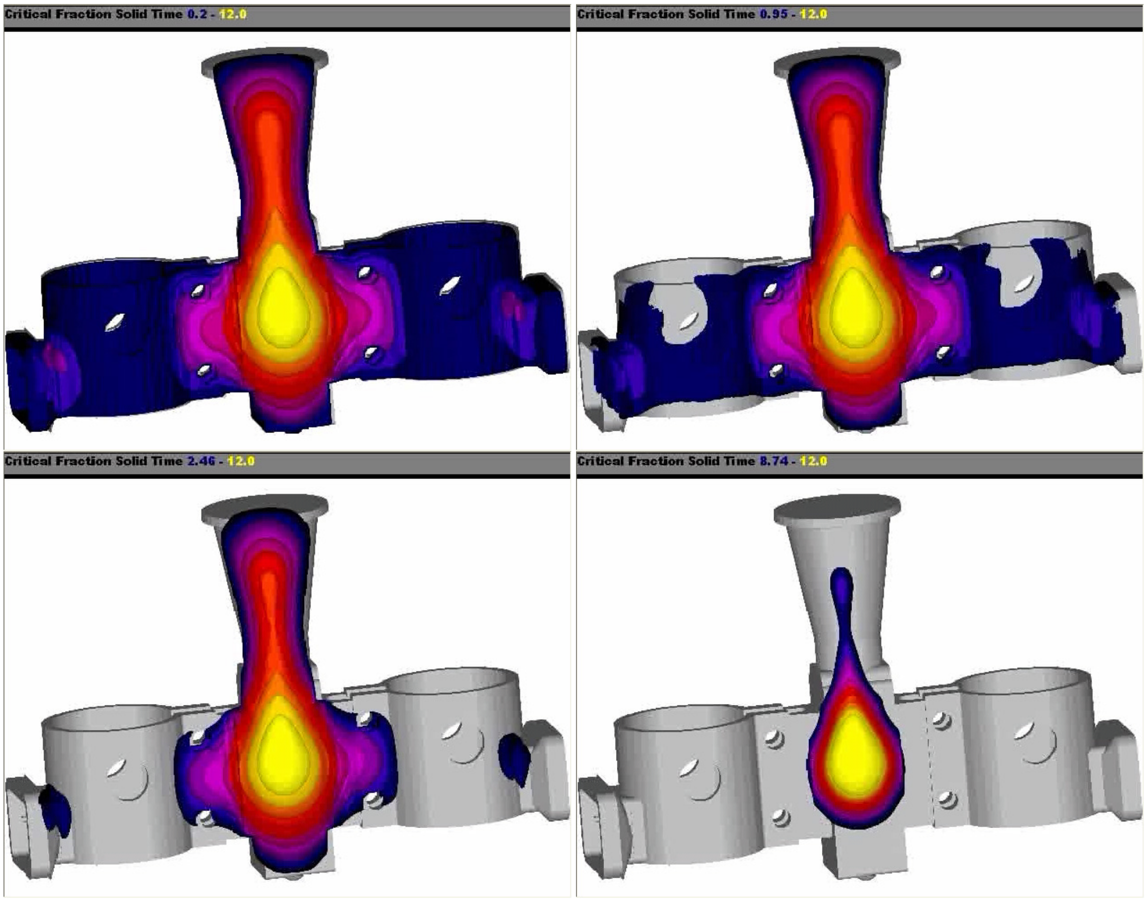
Once the movie is complete you should see a screen like this:



You can click on the icons to run the movie in the Windows Media Player or go to the movie folder. The normal procedure is to set your movie player to repeat, so you can see the movie several times without having to restart it manually each time. Once you have finished watching the movie, close the Media Player to return to SOLIDCast.

Note: While a movie is being created, you can still do other things, either in SOLIDCast or in any other software package. The movie creation process takes place in the background.

The following image shows four frames from a typical example of a Progressive Cast Scan movie, showing a progression of solidification on an investment cast tree. Two castings are attached to the central downsprue/runner/feeder:

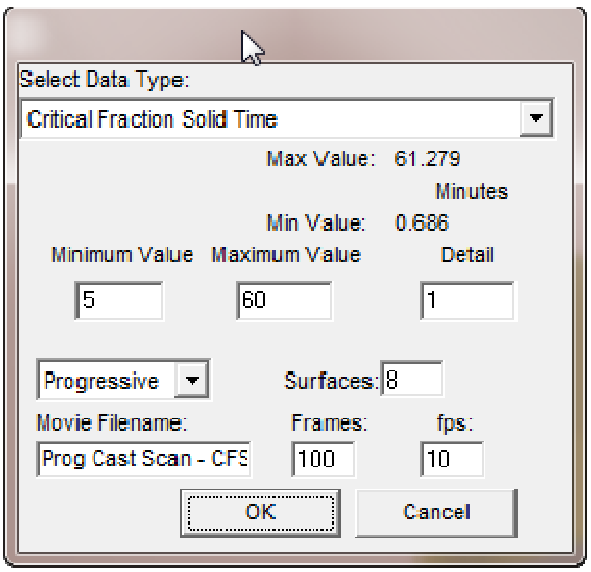


Any time that you want to later view the movie, or send it to another person to view, you just need to locate the WMV file that was created and double-click its icon to start viewing, or send the file as an email attachment to another person. When the other person receives the file, they can just double-click the icon to view the movie. It is not necessary to have SOLIDCast installed on a computer to view these movie files.

CAST SCAN MOVIES - Rotating

A Rotating Cast Scan movie is an alternative to the Progressive Cast Scan type of movie. In a Rotating movie, the range of data that is plotted stays constant, and the casting model rotates in space about a vertical axis. This allows the data that is plotted inside the casting to be viewed from all angles, and can sometimes give a viewer a more "intuitive" feel for where various indications inside the casting model are actually located.

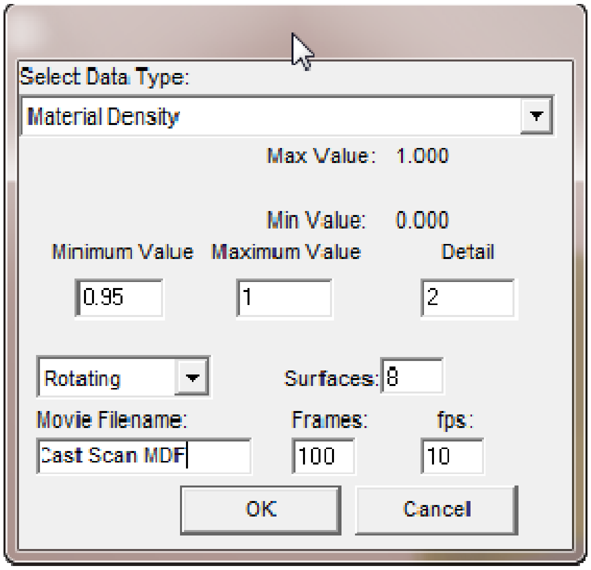
To create a Rotating Cast Scan, first orient the model at the angle at which you want it to appear initially in the movie, keeping in mind that the casting will rotate horizontally about its vertical axis in the movie. You must have completed a simulation prior to creating the Cast Scan movie. . Double-click on the simulation icon on the Project Tree to display the Simulation Summary box, then close the Simulation Summary box and select Simulation… CASTSCAN Movie. A window similar to the following will appear:



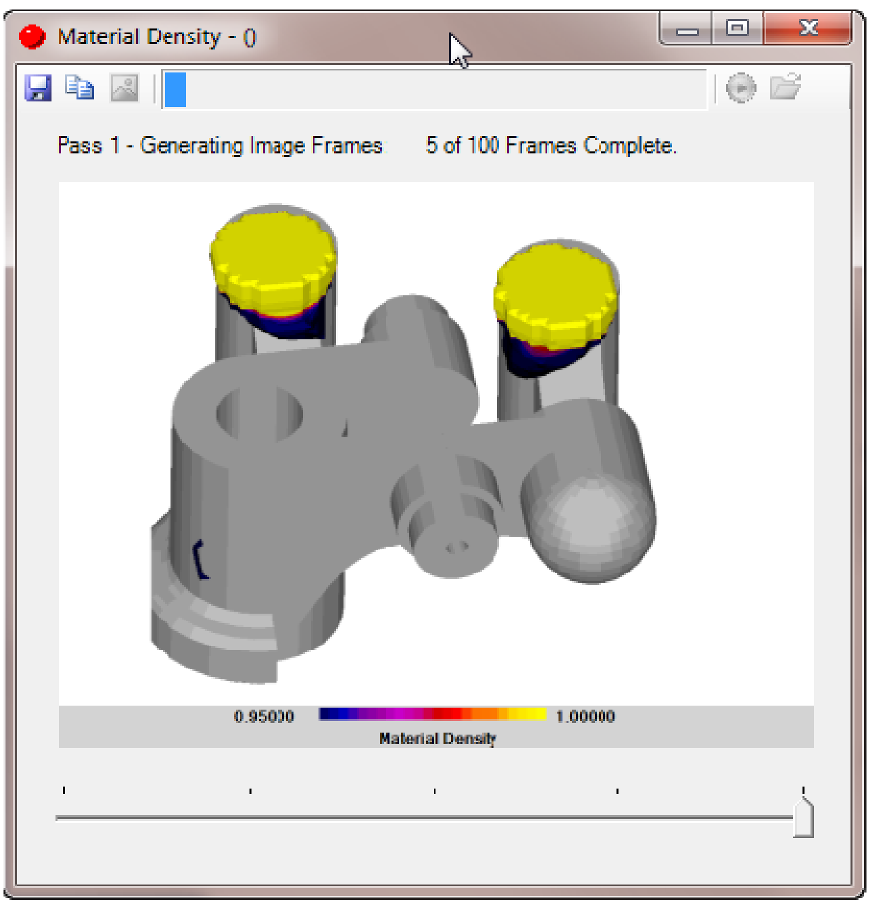
Now select the type of data you want to plot. In a Rotating Cast Scan movie, any data item may make sense to plot. Our example will show Material Density.

Fill in the Movie Filename, Number of Frames and fps (Frames per Second). In a Rotating Cast Scan movie, the model will perform one full rotation. Therefore, the number of frames determines the degrees of rotation from one frame to the next, If number of frames is specified to be 36, then the model will rotate 10 degrees in each frame. If the number is 72, the rotation from one frame to the next will be 5 degrees.

After filling out the items in the window, it should look something like the following:



After specifying the above information, click on OK. The system will display a window showing the progress of the movie creation. An example is shown here:



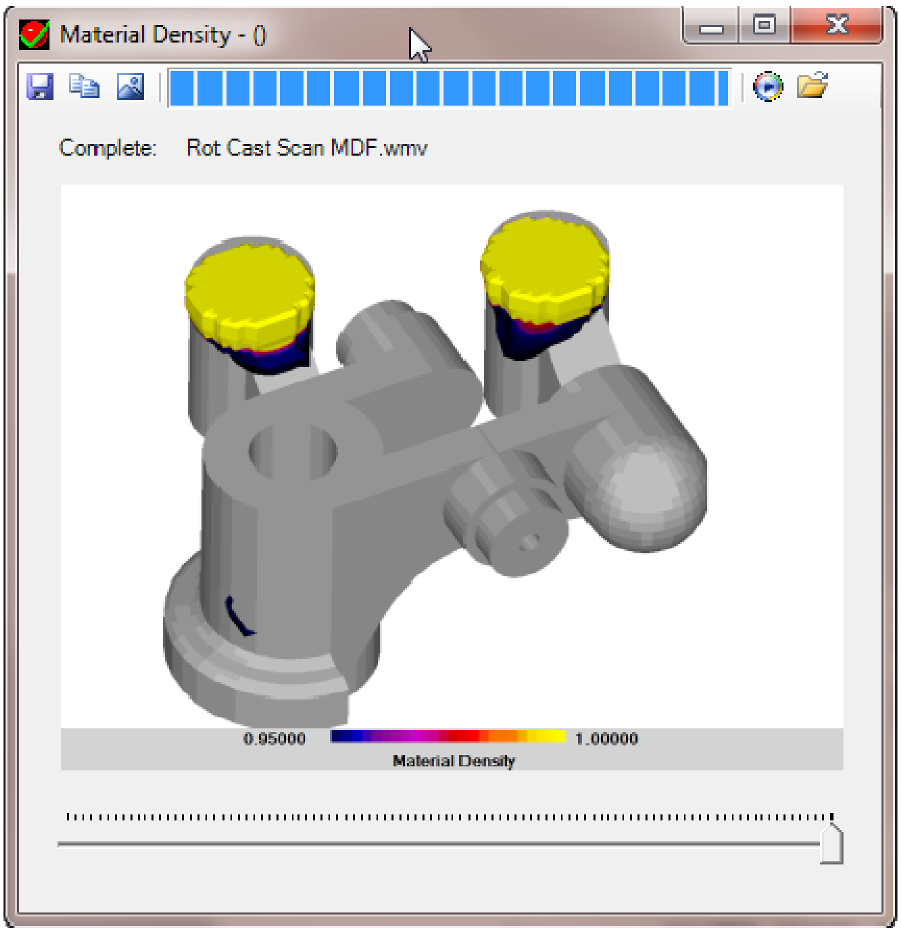
Note that when making Cast Scan movies, the time to create a movie can vary widely. Some of the items that affect the time to make a movie are as follows:

Number of Frames: The greater the number of frames, the longer the time to make the movie.

Surfaces: The more surfaces, the longer the time.

Detail: The higher the detail, the longer the time. (Note that higher detail produces a larger movie when using the default size in the Media Player).

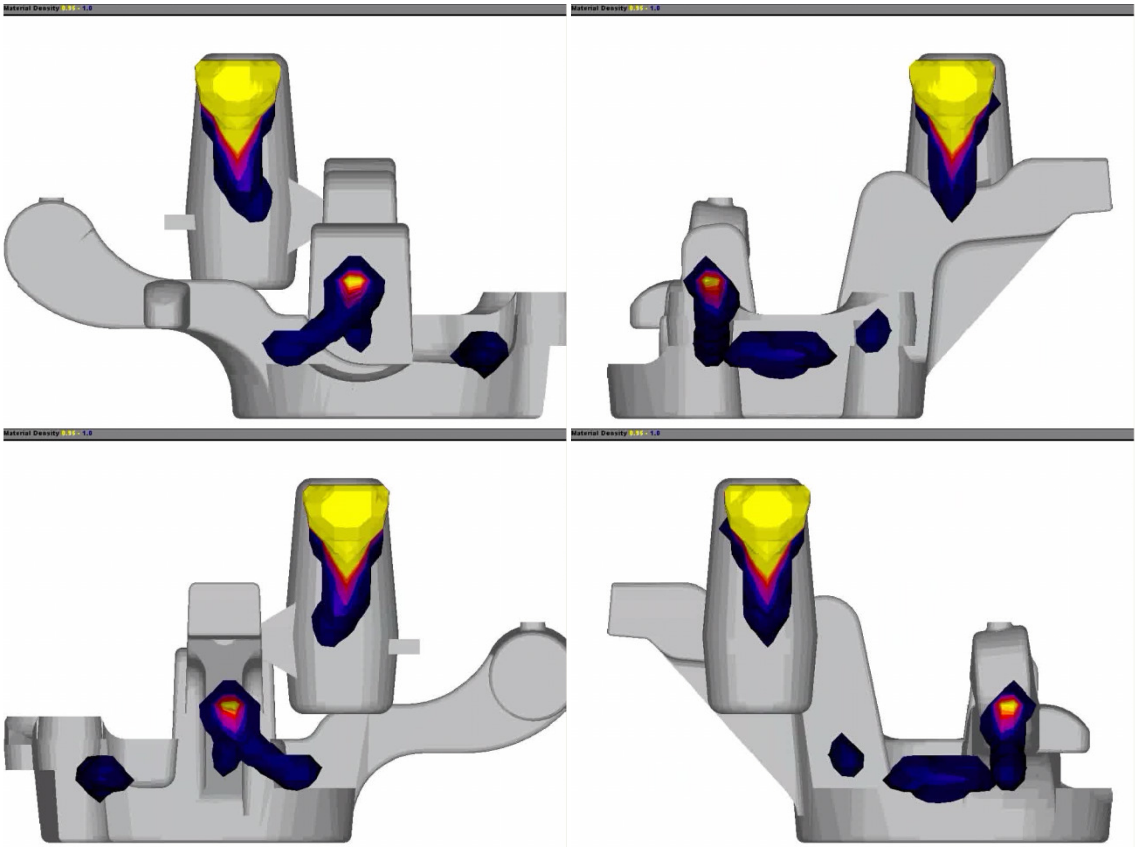
Once the movie is complete you should see a screen like this:



You can click on the icons to run the movie in the Windows Media Player or go to the movie folder. The normal procedure is to set your movie player to repeat, so you can see the movie several times without having to restart it manually each time. Once you have finished watching the movie, close the Media Player to return to SOLIDCast.

Note: While a movie is being created, you can still do other things, either in SOLIDCast or in any other software package. The movie creation process takes place in the background.

The following image shows four frames from a typical example of a Rotating Cast Scan movie, showing a simulation that was run on a casting with risering. The plot is of Material Density, showing riser piping and shrinkage-prone areas in the casting.



These images were created using 8 surfaces in each image.

Any time that you want to later view the movie, or send it to another person to view, you just need to locate the WMV file that was created and double-click its icon to start viewing, or send the file as an email attachment to another person. When the other person receives the file, they can just double-click the icon to view the movie. It is not necessary to have SOLIDCast installed on a computer to view these movie files.

UNIT 38: Components: The Import/Export Functions

SOLIDCast offers the ability to identify one or more shapes within a model, save these as a Component File, and then bring this Component File into another model. This ability allows the user to create a library of standard shapes (such as risers or gating components) and load them into model files. This also allows you to merge shapes from one model file into another, or to save an entire model along with its materials and load it into a new model.

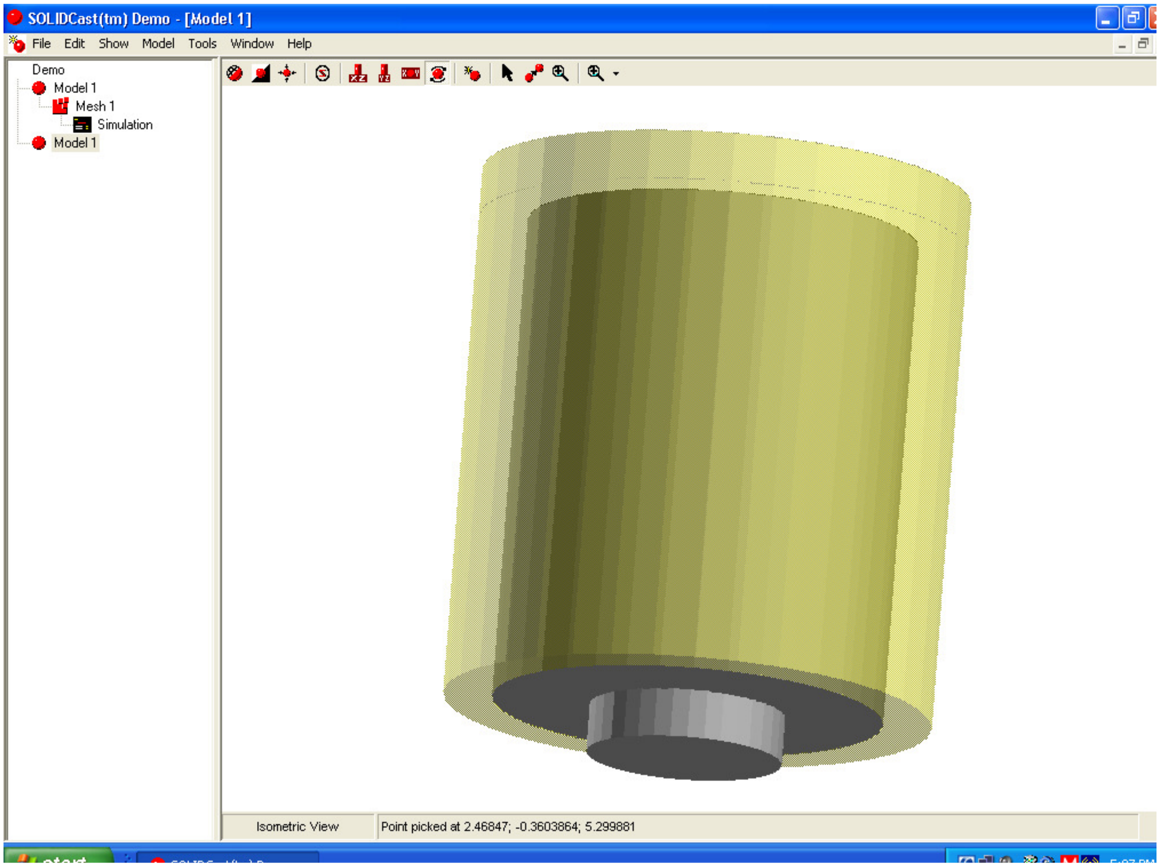
When a Component File is created, the materials (and their properties) that are included in the Component File shapes are saved along with the shape geometry. When you import a Component File into a model, the materials in the Component File are added to the Material List in the model. For example, suppose that you create a Component File of a riser with a particular exothermic material for a sleeve. Later, you create a new model file with only the casting alloy as a material. You then import the Component File of the sleeved riser into the casting model. If you examine the Material List after performing the import function, you will find that the exothermic sleeve material has been added to the Material List.

If you are building a casting model and then import a Component File, the alloy definition will not change. For example, suppose that you create a riser with SS 304 casting alloy and save this as a Component File. Later, you create a casting model using ST 1030 alloy, then import the Component File. The riser in the Component File will be designated as a riser in the casting model, but the alloy will remain ST 1030, regardless of what alloy was in the saved Component File containing the riser.

One exception to the above occurs when you import a Component File into a blank model space, i.e., a new model with no shapes. In this case, the alloy definition switches to the alloy that was used when the Component File was saved. This allows you to treat a Component File as a complete model, with all material definitions intact.

Example: Creating and Using a Library Component Riser

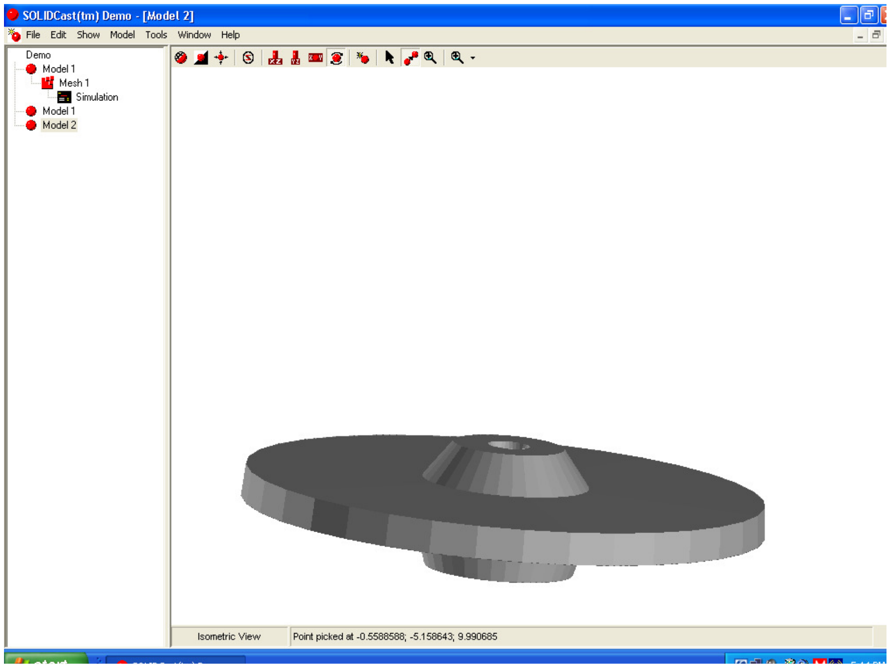
As an example of using the Component Export and Import functions, consider the creation of a Component File containing a riser with an exothermic sleeve. The riser can be created as a complete model, with the bottom centered at the point (0,0,0). Start SOLIDCast, select File… New Model and create the riser and sleeve geometry. In this case, the model is created with a series of cylinders. This riser might appear as follows (the point (0,0,0) is at the very bottom center of the riser neck):



To save one or more shapes as a Component File, it is necessary to select those shapes. In this case, this can be done easily by selecting Edit… Select All Shapes. Now, from the main menu select Model… Export… Selection. You will be asked to enter a name for the Component File. In this case, we enter the name ExoRiser. Note that component files are saved with an extension of .mdc. Therefore, you can locate component files on your system by looking for files that end with an extension of .mdc. In this case, the file that is created is called ExoRiser.mdc.

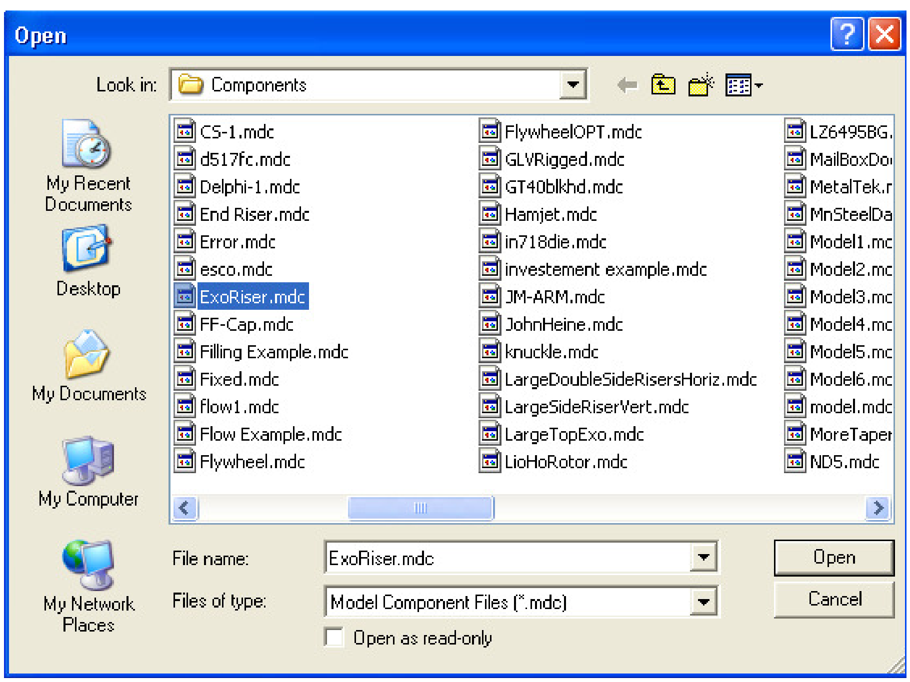
Note that once you save this entire model as a Component File, it is not necessary to save the model as a Project. Also note that Component Files are saved in a compressed format, which saves hard drive space.

Now suppose that we later want to use this standard riser on a casting. First, we would create the casting model, which might appear as follows:



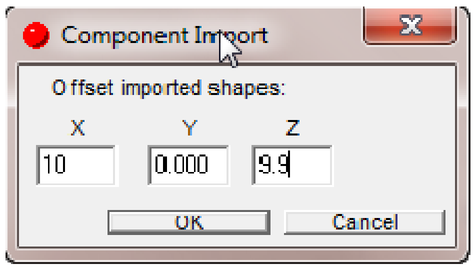
This model was created as a revolution of a 2D shape about the Z axis.

Now, to import the riser into this shape, we select Model… Import…SOLIDCast 5.x Component. In this case we are importing a Component, so we select the first option. This causes the system to display a list of component files (.mdc files), as follows:



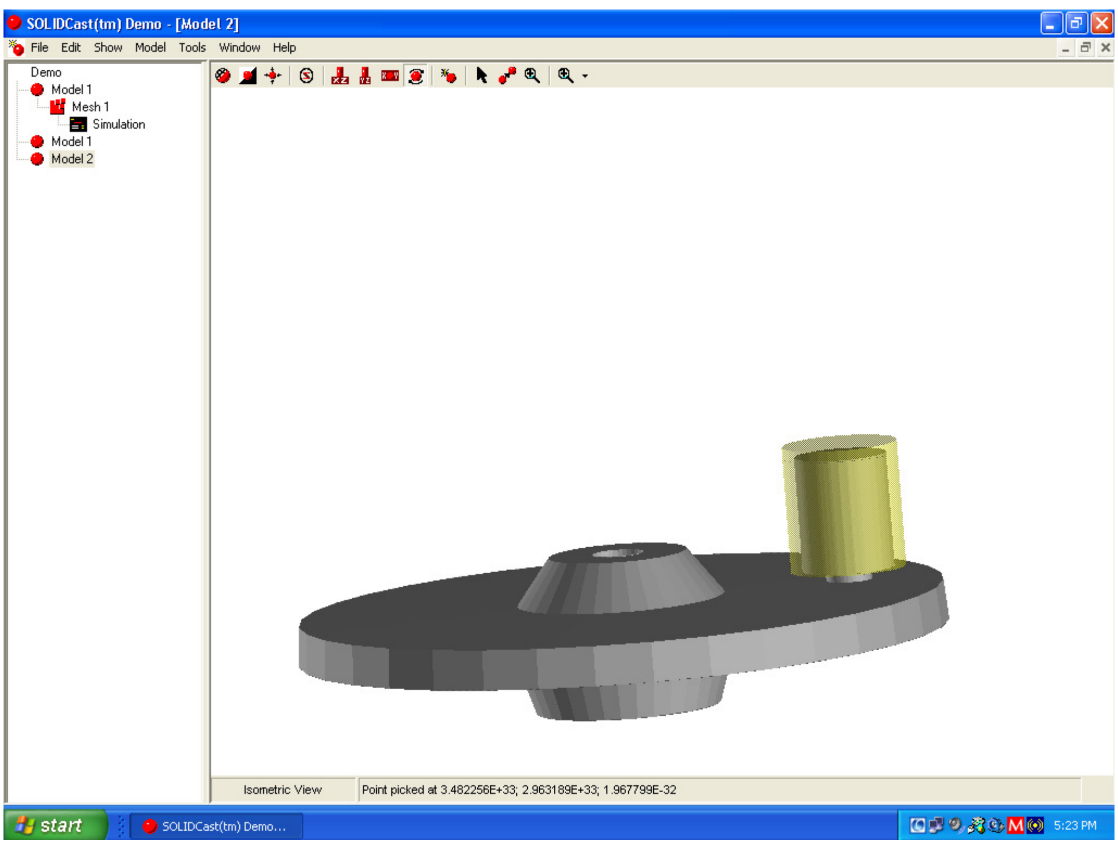
We can navigate to other folders or use the default folder and select the correct Component File. In this case we select the ExoRiser.mdc file which contains the riser created earlier.

Now the system will request that we enter a location to place the datum point of the Component File into the model file. In this case, we want the bottom center of the riser (the (0,0,0) point of the riser model) to be located at the point (10,0,9.9) on the model. This point can be determined by knowing the dimensions of the casting, or by reading the coordinates from the model display, with the model in an orthogonal (XY, XZ or YZ view). The coordinates are entered as follows:



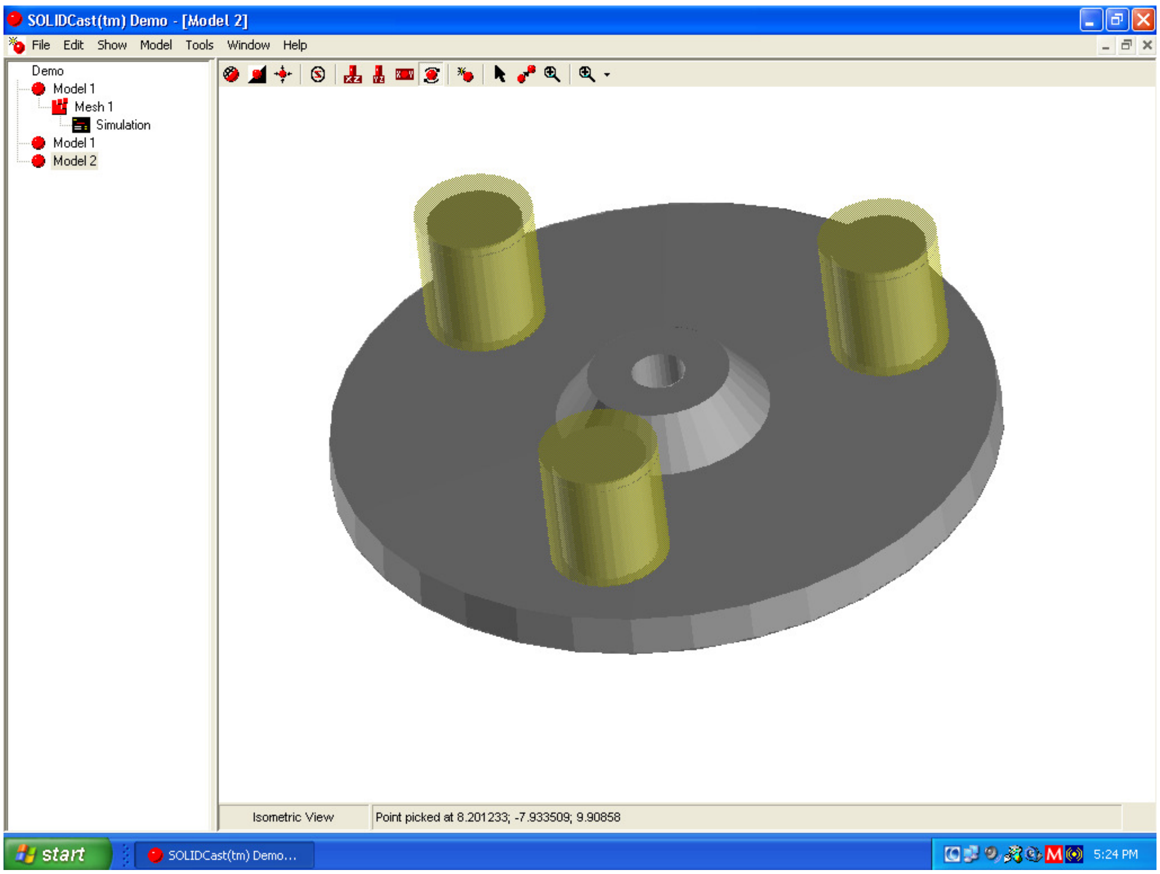
This provides an "offset" for positioning the component in the model.

Clicking on the OK button causes the Component File to be inserted into the model as shown here:



When a Component File is loaded into a model, then all of the shapes included in the Component File are automatically grouped together and can be selected with a single click. In this case, assuming that we want three equally-spaced risers, we can just click on the Select Shape Mode icon, click on the riser, select Edit… Copy and make two copies in a ring about the Z axis at 120 degrees.

The result of this appears as follows:



To summarize, the Component Import and Export function can be used in the following ways:

1. To create libraries of standard shapes that can be imported into model files.

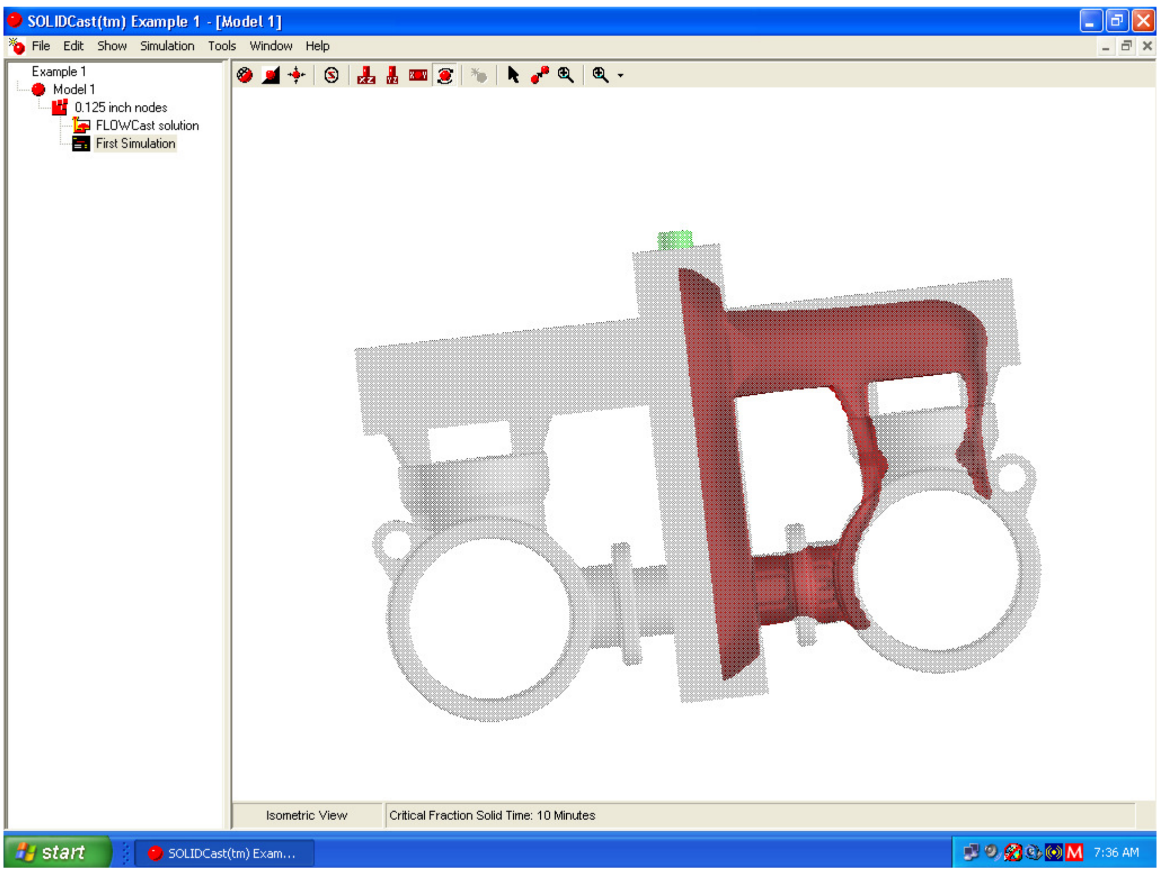
2. To merge shapes from two different model files.

3. To save a subgroup of shapes from one model file and load into a different model file.

4. To save an entire model file so that it can be imported into a new model space, or so that it can be sent to another machine or another user and then imported.

UNIT 39: Mirroring Results

For casting models that are symmetrical, SOLIDCast allows you to specify one or more Planes of Symmetry. This means that a simulation is run on only a portion of the model, which saves time or allows more nodes to be concentrated into a smaller portion of the model (see the section on using Planes of Symmetry). When results are plotted in a model where a Plane of Symmetry has been used, the results are shown only in the section of the model where the model was meshed. As an example, the model below was meshed and simulated with a Lower-X Plane of Symmetry at the center of the model. When Critical Fraction Solid Time is plotted as an Iso-Surface, the result appears as follows:



The Mirror function allows you to mirror the results to the other portions of the model, after the simulation has completed. In this way, you can view results as though the entire model had been simulated.

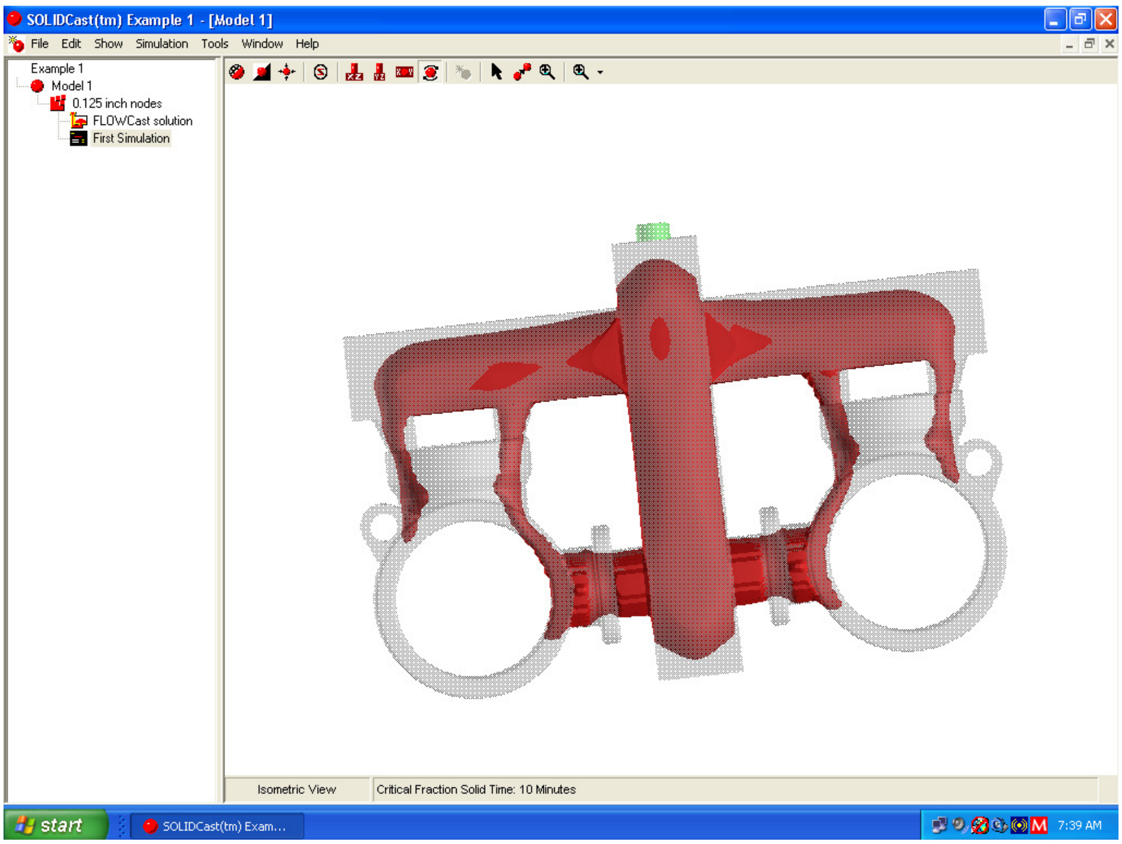
To mirror the results of a simulation, you must have one or more Planes of Symmetry that are NOT ROTATED (i.e., not turned through an angle). If one or more Planes of Symmetry are rotated to any angle other than 0 degrees, mirroring will not work.

To mirror the results, double-click the Simulation icon on the Project Tree, and then close the Simulation Status window. From the menu, select Simulation… Mirror Results. A warning box will appear which tells you that you should be sure that the simulation is complete before performing the mirroring operation. Click OK to proceed. A mirror button will appear briefly on the taskbar. When this disappears, the system has copied the data to the full model.

After the mirroring operation is complete, double-click the simulation icon again and then close the Simulation Status window, then plot results as you would normally.

Mirroring results is a one-time, permanent operation. After performing this operation once for a simulation, the simulation result files are altered and you do not need to perform it again, even if you exit the program and later reload the project. If you try to re-mirror, the system will let you know that this process has already been completed.

In the example model, performing a Mirror function and then plotting Critical Fraction Solid Time shows the following result:



Mirroring affects ALL types of output plots, such as Iso-Surface, Cut Plane, CastPic and all movie plots.

If you have two active Planes of Symmetry (such as a Lower X and a Lower Y) and have simulated one-quarter of the casting, the Mirror function will mirror about BOTH planes of symmetry and allow you to plot data within the entire casting.

UNIT 40: Cloning a Model

The Clone function allows you to create a copy, or a “clone”, of a model within a project. This is a useful feature if you want to create several successive design iterations of a model and retain each as a separate model within a project.

Cloning a model is very simple. On the Project Tree, double-click the icon for the model that you wish to clone. This will cause the model to be displayed in the model window, and that model’s icon will be highlighted. Then select File… Clone Model from the main menu. This will cause a new model to be listed on the Project Tree. The new model will be an exact copy of the currently selected model, including all of the material properties and even the name of the model.

At this point, you can close the model window that you are currently viewing and double-click on the new model that was just cloned. This will open a window on the new model. You can then edit this model, mesh it and run a simulation on the modified model. You may want to rename the newly-created model by triple-clicking on the name in the project tree and typing in a new name.

This feature allows you to retain previous versions of a model if you want to be able to load and view these previous versions, while creating new modified versions for analysis.

UNIT 41: FCC Custom Criterion

The FCC Criterion is a calculation that was developed by Franco Chiesa of the Collège de Trois-Rivières (Québec) for prediction of total evolved microporosity in aluminum castings. This criterion is based on a calculation involving Local Solidification Time (time from Liquidus to Solidus) and Solidification Wavefront Velocity at each point in the casting. In moderately degassed aluminum castings, this criterion gives an estimate of the total percent structural microporosity at each point in the casting. SOLIDCast includes a menu item that allows this calculation to be performed as stored as a Custom Criterion function that can then be plotted.

It has been found that this calculation can also give useful results in other alloys (such as iron and steel) for prediction of areas of potential microporosity or spongy shrinkage. This criterion also has proven useful in identifying thermal centers within castings, so that, for example, the location of possible secondary shrinkage in iron castings might be displayed.

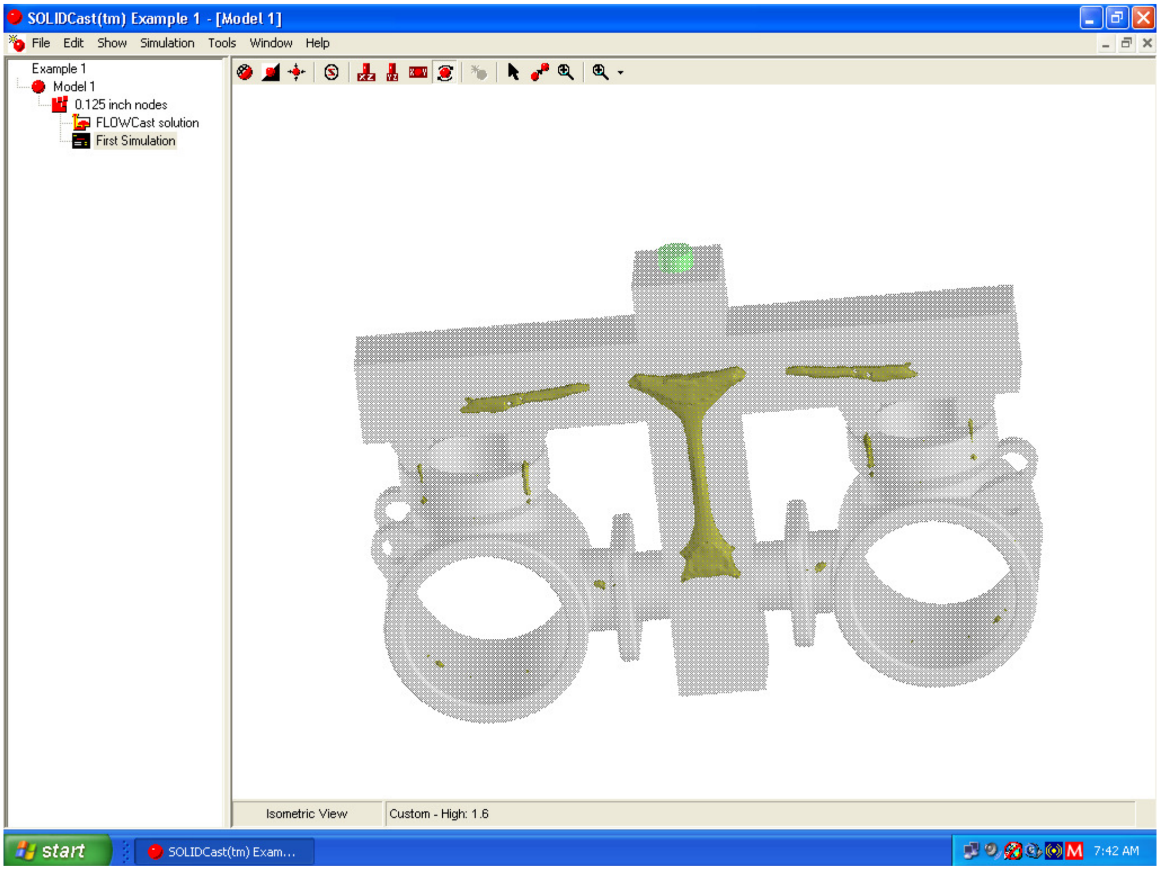
The range of numbers that will be developed for any given casting depends on the alloy and the geometry of the casting, so it is not possible to say ahead of time what a critical range should be for a particular casting. In general, the higher the number, the more potential for microporosity formation. It has been suggested that, to establish a starting point for plotting, look at a number that is at about 40% of the total range. For example, if the total range in a casting is from 0 to 1.9, then a number at about 40% of this range would be 0.76. This might be a good value to start plotting. Numbers higher than this will generally show less indications in the casting (the "more severe" areas) while lower numbers will generally show more indications ("less severe" areas).

Using the FCC Criterion in SOLIDCast is a two-step process:

1. After a simulation is complete, double-click the Simulation icon on the Project Tree to display the Simulation Status window, then close the Simulation Status window. Select Simulation…Calculate FCC Custom Criterion from the Simulation menu. A button will appear on the taskbar while the calculations are taking place.

2. After the calculation is complete, again double-click the Simulation icon on the Project Tree to display the Simulation Status window again. This action loads the minimum and maximum values from the FCC calculation into the system. Now close the Simulation Status window and select a plot. For example, select Simulation… Plot Iso-Surface. Under Select Data Type, click on the selection arrow, and find and select **Custom – High**. The range of the FCC Criterion will be displayed, and you can then begin plotting values.

An example of an Iso-Surface plot using the FCC Criterion is shown as follows:



In this image, you can see several areas in the casting that may be prone to microshrinkage.

UNIT 42: Tilt Pour

SOLIDCast lets you set a starting and ending angle to simulate tilt pouring. Tilt pouring is used primarily in the permanent mold process, and involves the gradual tilting of a mold from a horizontal to a vertical position in order to provide a relatively quiescent fill and also to promote strong temperature gradients and directional solidification within the model. Generally, a basin of liquid metal attached to the mold is filled while the mold is in a stationary position, and then the mold is tilted upright in a controlled process, gradually filling the mold cavity.

Setting up the tilt pour process involves specifying the starting and ending orientation of the mold as angles rotating about either the X axis or the Y axis. It is necessary to have a shape of Fill Material (see the chapter on Mold Filling) that represents the entry location of the molten into the mold. The system assumes that the Fill Time is equal to the Tilt Time. In other words, if you specify a Fill Time of 15 seconds, the system will assume that the time from start of tilt until end of tilt is 15 seconds and that at the end of this time the mold cavity is full of liquid metal. The Early Fill Tilt Pour utility can be used to modify the process, if fill time and tilt time are not the same. See the Stand-Alone Utilities Unit for details.

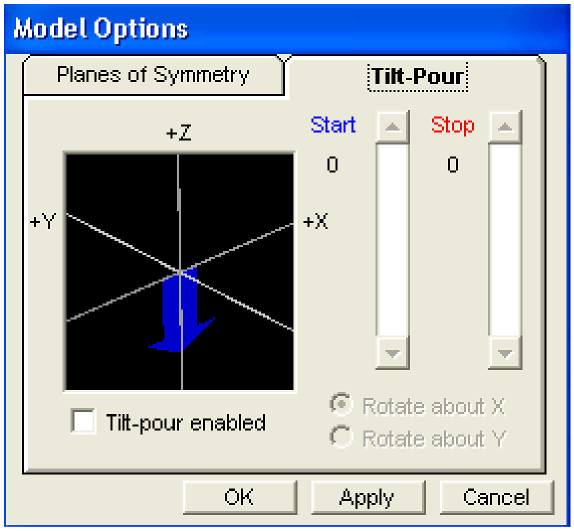
We highly recommend that the model of the casting and model be constructed so that is in the correct orientation at the end of tilting. The default orientation in the system is that +Z represents "UP" and –Z represents "DOWN". Keeping this in mind, construct your model so that the mold is in the correct orientation at the END of the tilting process, i.e., when the casting is solidifying. This makes it easier to properly set the tilt pour angles and interpret the results.

The key to using the Tilt Pour Angles is to remember that, when you are setting them, you are specifying to the system which direction is DOWN at the start and end of tilt. The model itself does not actually rotate. When the simulation is running, the system gradually changes an internal vector that indicates the direction of gravity, to simulate the tilt pour process. Therefore, using the Tilt Pour Angles feature, you are indicating on the current model, as it is currently oriented, which direction is DOWN at the start and at the end of the tilting process. As stated previously, we recommend that the model be oriented such that it is the correct default orientation (-Z down) at the end of tilt. This means that the ending angle for tilt pour should be 0.

Note that if you use the optional FLOWCast module, tilt pour parameters will be used, and you will actually see the casting rotate during the filling process.

To use the Tilt Pour feature, you should have the model displayed in SOLIDCast. Setting of Tilt Pour angles MUST BE DONE PRIOR TO MESHING THE MODEL. The Tilt Pour Angles will not be active if you set them AFTER meshing the model.

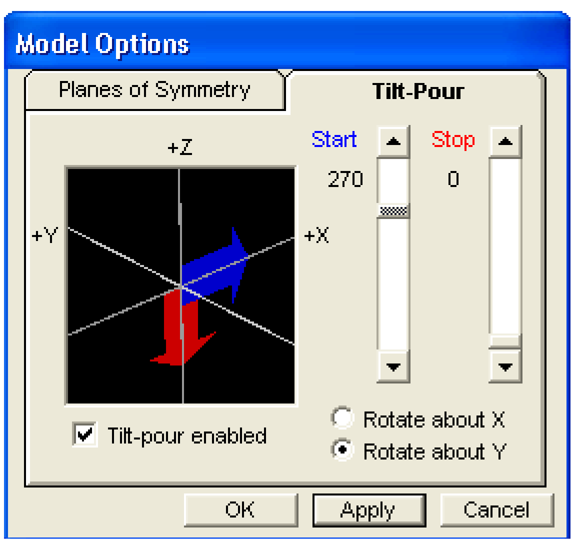
Select Model from the main menu, then select Options. You will see a window which has two tabs, one for Planes of Symmetry and one for Tilt Pour. Select the tab labeled Tilt Pour. You should see the following on the screen:



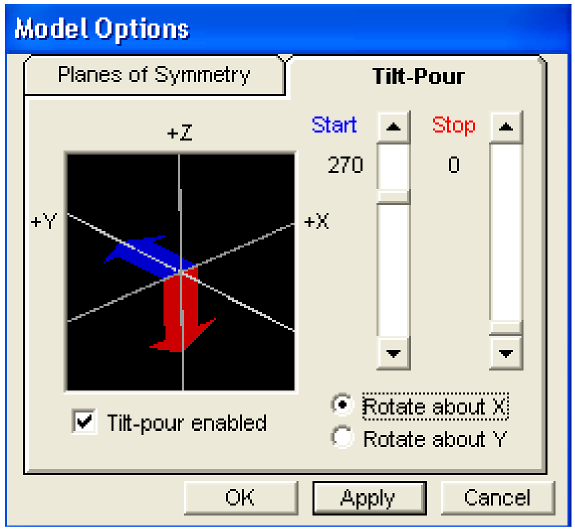
To activate Tilt-Pour, click on the box labeled “Tilt-pour enabled”. You then have the option of rotating the tilt-pour arrows around either the X axis or the Y axis. You can set an angle for the Start position and another for the Stop position. (Remember, we recommend that the model be constructed so that the Stop position is the default position with –Z as down.)

The angles are pointing in the direction that is considered to be DOWN on the model at the start of tilt and at the end of tilt. The arrows are moved by adjusting the slider bars on the right.

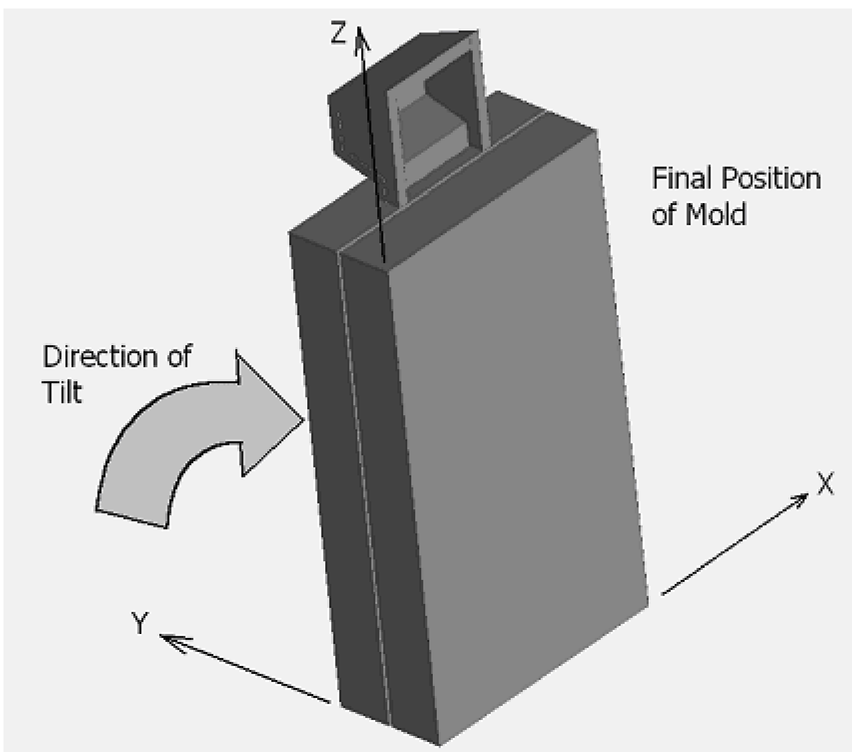
Rotation around the Y axis would appear as follows:



Rotation about the X axis would appear as:

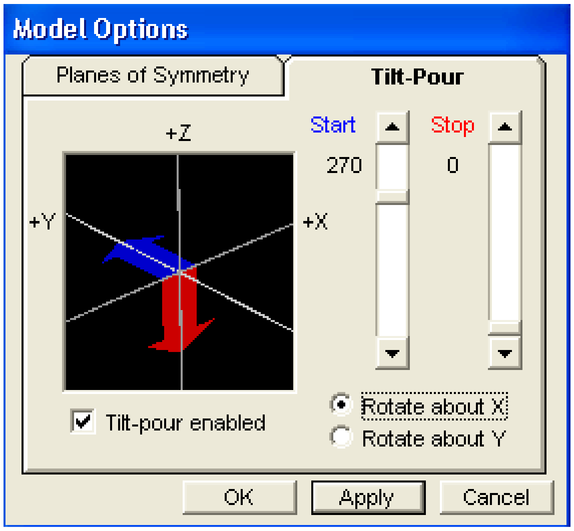


As an example, suppose that we have created a model as follows:



This model has been oriented such that it is in the correct, UPRIGHT position at the end of tilt, i.e., the –Z direction is considered to be DOWN at the end of tilt. The Stop angle would then be set to 0 degrees, since there would be no rotation from the default –Z direction at the end of tilt.

The question then is how to set the Start angle for this model. As shown, the model as it has been constructed would rotate about the X axis during the tilt. The direction which is DOWN when the mold is in the Start position is the +Y direction on the model in its current orientation (remember, we are specifying which direction would be DOWN on the model, as it is CURRENTLY ORIENTED, when it is in the Start position). Therefore, we would want to set the Start angle to the +Y direction on this model. This would appear as follows:



The rotation is about X, and the Start angle is set to 270 degrees, which we can see from the display is in the +Y direction.

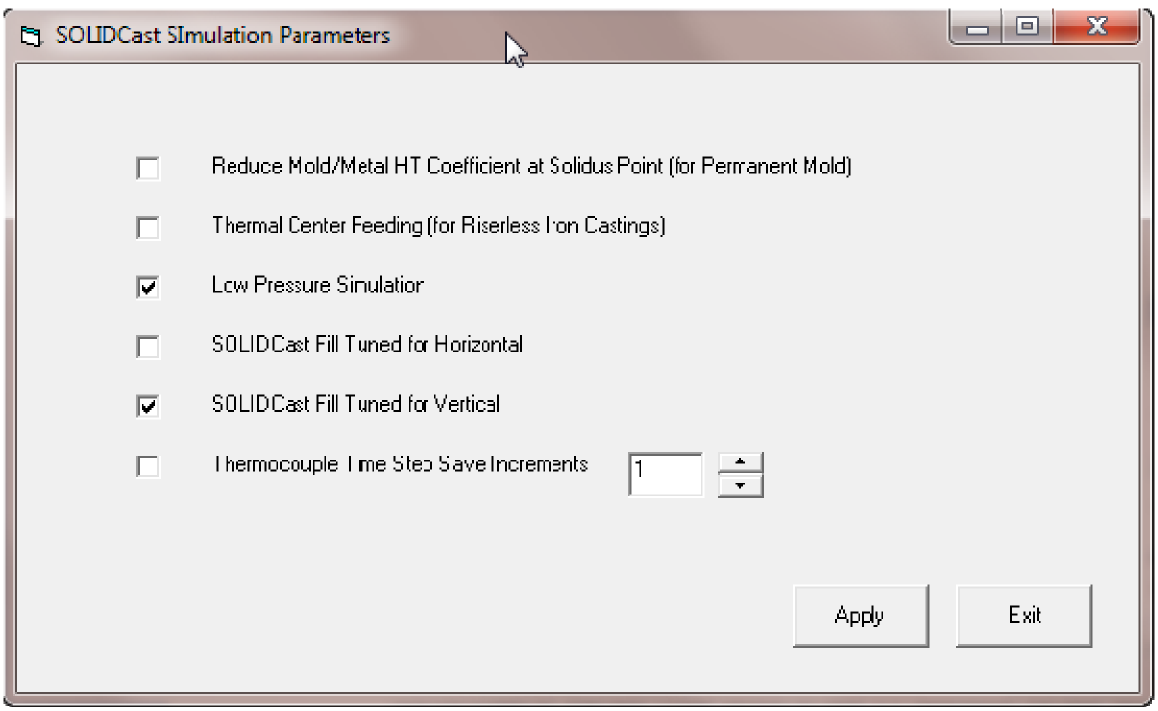
Be sure to press the “Apply” button to activate your selection of Tilt Pour angles.

Once the Apply button has been pressed, the tilt pour setup is enabled and any simulation involving filling, using SOLIDCast or FLOWCast, will use these parameters.

UNIT 43: Low Pressure Permanent Mold Casting

SOLIDCast has the ability to better simulate the low pressure permanent mold process. This feature is really quite simple. What happens is that the gravity vector is inverted at the end of filling. This means that, if you build a model in the normal way (+Z = up), the filling will be shown as normal with gravity pulling down, but at the instant the fill is complete the direction of gravity switches 180 degrees, effectively inverting the mold and feeding from the bottom rather than from the top.

To activate this feature, click on Tools…Simulation Parameters. To activate Low Pressure Simulations, check the box labeled Low Pressure Simulation, then click the Apply button, then click Exit.



The sprue in this case will act exactly like a riser would if you inverted the model. Therefore, you want the sprue to have enough mass, and extend far enough below the casting model, so that it does not get emptied out by the casting contraction…otherwise it will show shrinkage from the sprue into the casting, which would not be too realistic. What you are trying to do with this is to show potential for internal shrinkage in isolated hot spots that might be forming in the casting. Niyama and the FCC Criterion can also be used for prediction, but they are independent of orientation and are not affected by this feature.

Note that if you set a Simulation Parameter using this utility, it stays set until you come back in and change it. So, if you only do low pressure simulations occasionally, remember to come back in and reset the parameter when you are done, or you may mistakenly apply low pressure to normal gravity processes.

UNIT 44: Cooling Channels in Permanent Mold Casting

When you have a model displayed in SOLIDCast and you select Model… Materials List, then select the Mold Materials tab, you are displaying the selection of materials can be used in a model other than the casting alloy. These are the Mold Materials. There are various types of Mold Materials which can be represented in a model. The type of material is indicated by the selection labeled Material Type on the screen, which has a small arrow next to it. If you click on this arrow, you will see a list of different material types which are available to use for materials in the mold. One of these is the Cooling Channel.

A Cooling Channel is typically used in a permanent mold die. This is a channel or area in the die where a fluid is passed, which is assumed to have a constant temperature, for the sake of cooling that area of the die. The Cooling Channel type of material allows this channel to be turned ON or OFF at various times during the process.

If you have a cooling channel in your die which is continually on (it is not turned off during the process), then instead of using the Cooling Channel type of material, you can use the Constant Temperature type of material. A Constant Temperature material is maintained at the given temperature throughout the entire simulation; it is never turned on or off.

If the cooling channel in your process cycles on and off during the process, then you should use the Cooling Channel material type. This type of mold material allows the cooling channel to be controlled in two ways:

1. The first method of turning the Cooling Channel on and off is through the use of a timer. In this case, the system allows you to specify a Start Time and a Stop Time, in minutes. These are measured from the start of each cycle, i.e., from the start of pouring. Before the Start Time, the cooling channel is not active, i.e., it is turned OFF. After the Start Time and before the Stop Time, the cooling channel is ON, and then after the Stop Time the cooling channel is OFF again. This is repeated for each cycle of a permanent mold simulation.

2. The second method of turning the Cooling Channel on and off is through the use of a thermocouple which is placed at a certain (x,y,z) location in the model. In this method, a High Limit (or a Low Limit) can be specified for operation of the channel. If a High Limit is specified, then the cooling channel starts out turned OFF, and the system monitors the temperature reading of a thermocouple at the given (x,y,z) location in the model. Once the thermocouple registers a temperature above a given Set Temperature, then the cooling channel is turned ON. If the thermocouple reading drops below the Set Temperature, then the cooling channel is turned OFF. This type of control acts continuously throughout the simulation. (Note: There is also a setting called Low Limit, which will turn the channel ON when temperature drops BELOW a given Set Point, and OFF when the temperature rises above the Set Point. The Low Limit control is typically used to control a heating element rather than a cooling channel).

As an example, suppose that you have a cooling channel in a die, such that water runs through the channel. You place a thermocouple at a specific position in the die such that its location is at X = 165 mm, Y = 0 mm and Z = 102 mm. If the temperature recorded by this thermocouple goes above 345°C, then you want the cooling channel to be turned ON. Once the temperature drops below 345°C, then the cooling channel should be turned OFF.

In order to simulate this condition, you must be able to specify the temperature of the water in the cooling channel, and also the heat transfer coefficient at the inside surface of the channel (the fluid HTC).

Some typical values for a water cooling channel might be:

Temperature: 60°C (140°F)
Heat Transfer Coefficient: 1532 W-m^2/°K (270 BTU/Hr-Ft^2-°F)

Typical values for an air cooling channel might be:

Temperature: 32°C (90°F)
Heat Transfer Coefficient: 142 W-m^2/°K (25 BTU/Hr-Ft^2-°F)

These values would be in effect ONLY while the cooling channel is turned ON. When the channel is turned OFF, it has little or no effect on the heat transfer in the die.

In order to create a material to use to simulate this cooling channel, you would select Model … Materials List and then enter the following data for the material at the bottom portion of the window:

Name: Cooling Channel 1
Type: Cooling Channel
Initial Temp: 38°C (See Note 1 below)
Thrm Cond: 17 W/M-°K (See Note 1 below)
Spc Ht: 837 J/Kg-°K (See Note 1 below)
Dens: 6400 Kg/M^3 (See Note 1 below)
Cooling Channel Type: High Limit
Temperature ON: 60°C
HT Coeff. when ON: 1532 W-m^2/°K
Temperature Set Point: 345°C
X: 165 mm
Y: 0 mm
Z: 102 mm

(Note 1: Thermal Conductivity, Specific Heat, Density and Initial Temperature are properties which are NOT USED for cooling channel type of materials. Therefore, you can enter any data into these fields.)

Now, click on the button labeled "Add to List". This action will create a cooling channel material in the "Materials in List" box on the screen, with the properties as specified. Please note that this only creates a material; in order to actually use a cooling channel in the model, it is necessary to actually create a shape in the model, and to designate its material type as being the cooling channel material type which is on the mold material list. There is no limit to the number of shapes which can be created using this material type. For example, if you have four channels in a mold and they all have the same properties (they are all controlled the same way, and have the same temperature and HTC when ON), then they can all be designated as this type of cooling channel material.

You can create up to 8 different types of cooling channel material in a single model. This means that you can vary the types of control, the temperature when ON and the HTC when ON up to eight different ways in a model. Typically, we would name these something like CC1, CC2, CC3, etc. As mentioned, there is no limit on the number of shapes that can be created in the model with each cooling channel material type.

Whenever there is one or more cooling channel material listed in the Mold Materials for a model that is designated either as a High Limit or a Low Limit type of control, and a simulation is run, a file is created which contains the time and temperature data for the readings from that thermocouple. This file is named tcdata.tmp, and can be found in the folder that contains the simulation result data. This is a text file that can be imported into programs such as Microsoft Excel for viewing or plotting.

One important note is that a time/temperature data file will be created in the simulation result file, even if there is no cooling channel shape in the model, as long as a cooling channel material (either High Limit or Low Limit type) is listed in the mold material list for a model. This allows the user to place from 1 to 8 thermocouples in a model, without requiring that any cooling channels actually exist in the model.

By default, a thermocouple time/temperature data file creates an entry for every time step in a simulation. If you have a large simulation with many cycles, this can create a very large file with many thousands of entries. There is a method by which the number of time/temperature entries can be reduced. It is possible to create a small text file called $tcinc.500 in the SOLIDCast installation folder (this folder is normally c:\Program Files\SOLIDCast). This $tcinc.500 file can be created with NotePad or any other Windows program that can create a text file. This file should contain a number. For example, suppose that the file $tcinc.500 exists and contains the number 10. This means that any time/temperature data will be written only every 10 time steps instead of every time step. By using this method, the amount of time/temperature data can be reduced by a factor of 10. If the file $tcinc.500 exists and contains the number 100, then data will be written to the tcdata.tmp file every 100 time steps. If $tcinc.500 does not exist, the system writes time/temperature data every time step.

For calculating the surface heat transfer coefficients inside cooling channels, we have the HTC Calculator utility. This program can calculate HTC's for air and water cooling channels, given the flow rate and the channel diameter. The calculated value would then be used when creating a cooling channel type of material, under the "HT Coeff. when ON" entry.

UNIT 45: Riser Design Wizard

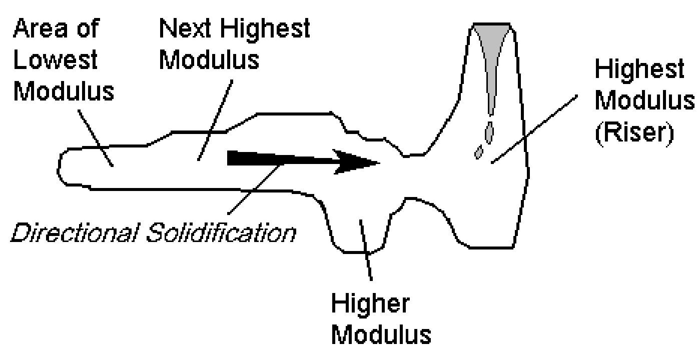
The **Riser Design Wizard** allows you to analyze a casting model without risers or feeders, and determine the number, placement and size of risers or feeders that should be attached to the casting. After performing these calculations, you should add the risers to the model and perform a verification simulation to determine whether any adjustments in design may be necessary.

This wizard is designed primarily for shrinking alloys, such as steel, aluminum, copper and white cast irons. Gray and ductile irons, which typically have an expansion phase, have their own riser design functions built into the Iron Property Calculator utilities which are a part of SOLIDCast.

BASIC CONCEPTS

The **Riser Design Wizard** uses the concept of a "modulus" to analyze the casting, differentiate feeding areas, and design appropriately-sized risers.

The casting modulus is a term coined years ago to describe the ratio of the volume to surface area of a casting, or of its various sections. According to Chvorinov's rule, the sections of a casting which have a higher modulus (volume:surface area ratio) will freeze last, while those sections with a lower modulus value will freeze earlier. The ideal situation is for the casting to undergo directional solidification toward the riser (or feeder); this generally means that a riser needs to be attached to a section of the casting that has the highest modulus value. The modulus of the riser should be equal to or greater than that of the casting to which it is attached. This, in theory, will allow any contraction that is taking place as the casting solidifies to be fed by a pool of molten liquid metal in the riser, and thus prevent shrinkage from occurring, as illustrated in the following figure:

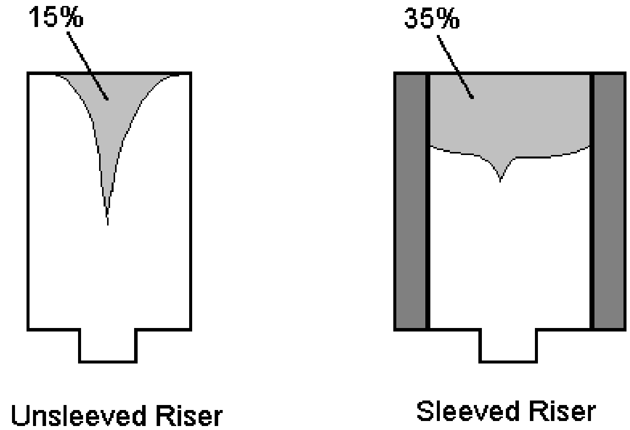


Since modulus is defined as volume / surface area, the units generally used units of length as follows:

English Units: $in.^3 / in.^2 = in$ Metric Units: $cm^3 / cm^2 = cm$

The **Riser Design Wizard** calculates an "effective modulus", or "thermal modulus", from the results of a simulation of an unrigged casting. This approach gives a modulus value equivalent to the volume:surface area ratio for each point within a casting, depending on how the casting solidified. This is considerably more accurate than the traditional approach of estimating volumes and surface areas of a casting through manual calculations, as it takes into account the dynamics of the process, such as heat saturation, etc.

Another consideration for proper riser design is how much liquid metal is required by the casting, and how much metal a riser of a given size can deliver. The amount required by the casting is a function of the volume of the feeding area and the contraction of the alloy. The amount of metal which can be delivered by a riser is a function of its Riser Efficiency; a riser which has an efficiency of 15%, for example, can deliver 15% of its total volume of metal to feed the contraction which is occurring in both the casting and the riser. An illustration of two common riser types and typical values which might be used for efficiency, measured as the amount of feed metal delivered, is as follows:



Riser efficiency has been estimated a number of different ways. Two of these are incorporated into the **Riser Design Wizard**:

1. AFS Method: AFS/CMI publishes a handbook entitled Basic Principles of Gating and Risering. This handbook contains Figure 8.2, "Surface Area to Volume Relationships for Various Riser Types". This chart shows the appropriate ratio of Riser:Casting volume for various Riser:Casting modulus ratios and various types of risers. A Riser Efficiency Factor can be derived from this chart which gives the efficiency of a specific riser attached to a specific casting. This is the "default" efficiency which is used in the **Riser Design Wizard**.

2. Wlodawer Method: A number of foundry engineers use an approach pioneered by Wlodawer, who defined an efficiency according to the following equation:

$$(Vc + Vr)\, s = Vr\, E$$

Where Vc = volume of casting
Vr = volume of riser
E= Efficiency of riser (%)
s= shrinkage of alloy (%)

Rearranging terms in this equation, the required riser volume is

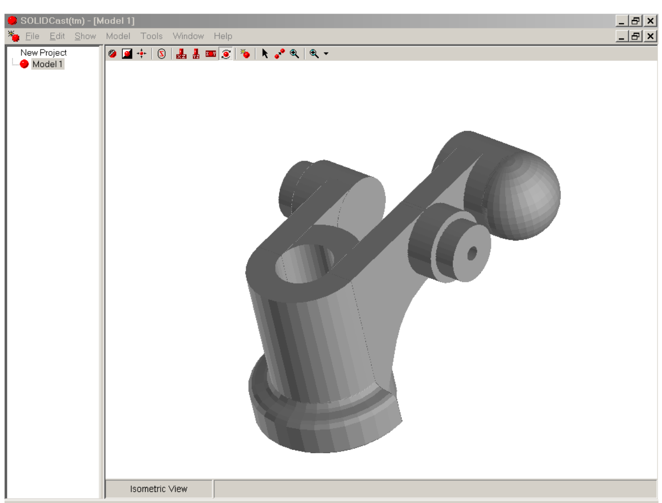
$$Vr = Vc / (E/s - 1)$$

Most people using the Wlodawer approach assume a specific efficiency (E) for a specific type of riser. These assumed efficiencies are built into the **Riser Design Wizard** for unsleeved risers, insulating sleeves and exothermic sleeves. You can also enter a different riser efficiency factor, based on your experience with a particular type of riser.

It should be noted that the **Riser Design Wizard** uses the **Critical Fraction Solid** point on the alloy curves to determine the amount of shrinkage that the alloy undergoes. This value can be read on the alloy curve at the CFS point; note that it may be substantially lower than the value of the shrinkage entered in "System Parameters/Alloy Curves" at the beginning of the simulation.

USING THE RISER DESIGN WIZARD

To use the **Riser Design Wizard**, you should first run a simulation of a casting model with no risers or feeders attached. We call this ‘freezing naked’. You may optionally include gating in this model in order to take into account temperature distribution within the casting due to filling. You may also include such items as chills, if you plan to use these and want their effect to taken into account when designing the risers.



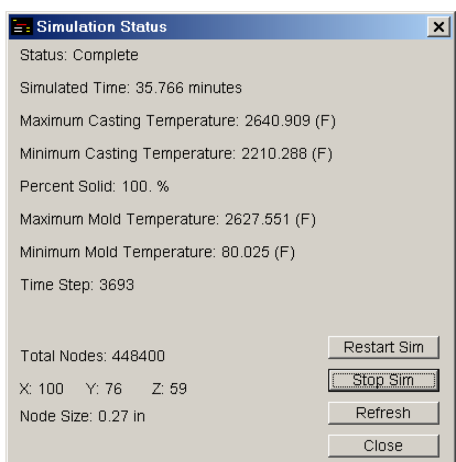
As an example, consider the casting model shown on the previous page. Here we have a model of a casting only, with no risers, gating or chills. The first step is to mesh this casting model and run a simulation.

> **Hint:** In order to make this step quicker, there are some things that we can do to make the simulation run faster than normal. First of all, since riser calculations are generally not as exacting as a full simulation, we can generally mesh the casting model with fewer (larger) nodes to create a coarser mesh which runs faster. Also, the Material Density (volumetric) calculations are not used when analyzing the results for riser design. Therefore, from the main SOLIDCast menu you can select **Tools**… **System Parameters**… **Model & Sim** and set the **Volumetric Calculation Interval** to a high number (for example, 1000). This will cause the volumetric calculations to be performed less frequently during the simulation, and will speed up the overall speed of the simulation. Be sure to reset this entry to a normal setting (the default value is 10) before running any other simulations, or your Material Density results will be unusable.

It is important to remember that you should place mold material around the casting model that is representative of the type of mold material that will be used to produce the casting. Normally, you should surround the casting completely with the mold material – this means that, if you are creating a mold during meshing, you should have the "Open Top" option turned OFF. Also, if you are creating a shell around the casting model as in investment casting, the shell thickness should be representative of the actual shell thickness, and you should run the View Factor Calculation on the meshed model prior to running the simulation.

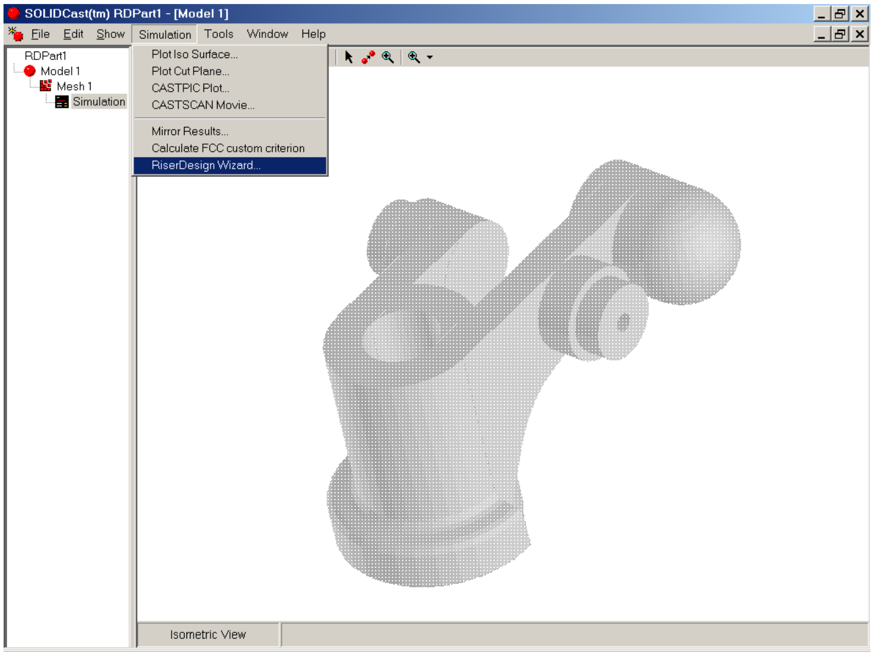
Also, be sure to allow the simulation of the casting to run until the casting is 100% solid. This is the default setting when starting a simulation.

Once the simulation has completed, double-click the Simulation entry on the Project Tree (on the left side of the screen). This displays the Simulation Status window as shown in the example below; the status should say "Complete" as shown.



Close this window to proceed to the next step.

At the SOLIDCast main menu, select **Simulation.** You will see an entry on the sub-menu which is titled **Riser Design Wizard**, as shown in the image below:

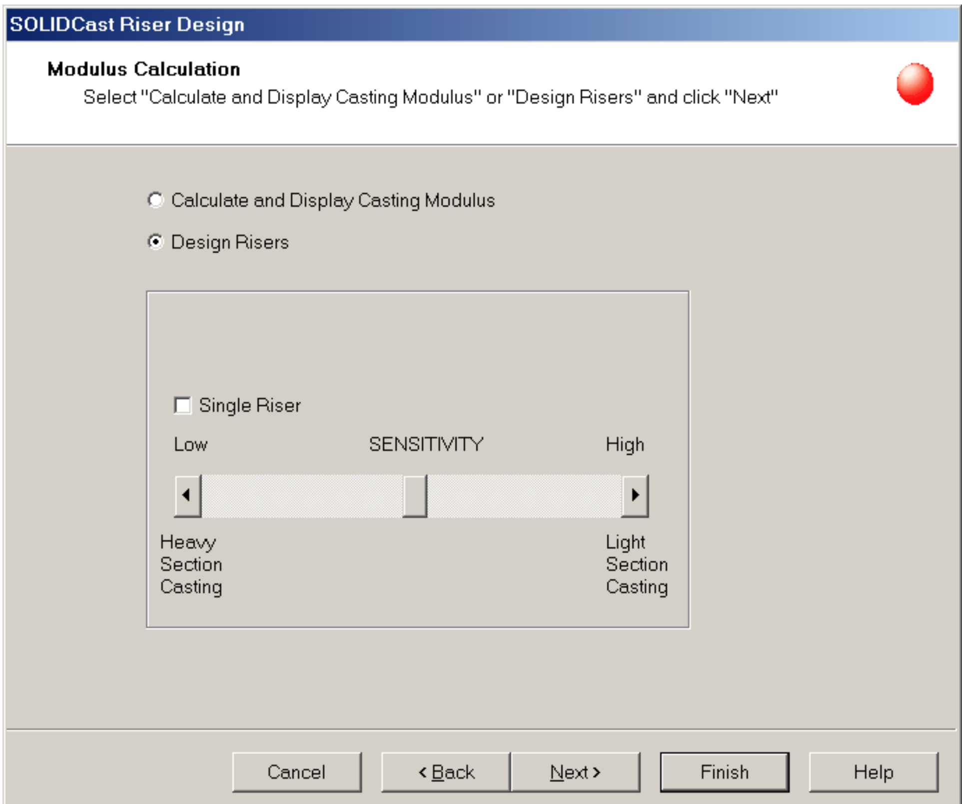


Select **Riser Design Wizard** from the menu. The following screen should appear:

This is the initial introduction screen for the **Riser Design Wizard**. If you don't want this screen to appear each time you start the Wizard, just click on the box labeled "Do not show this step next time."

Notice at the bottom of this window are a set of buttons labeled **Cancel**, **Back**, **Next**, and **Finish.** These are the buttons that you can use to navigate through the wizard to perform its functions.

Now click on the button labeled **Next**. You will see the following:



This screen allows you to perform one of two functions:

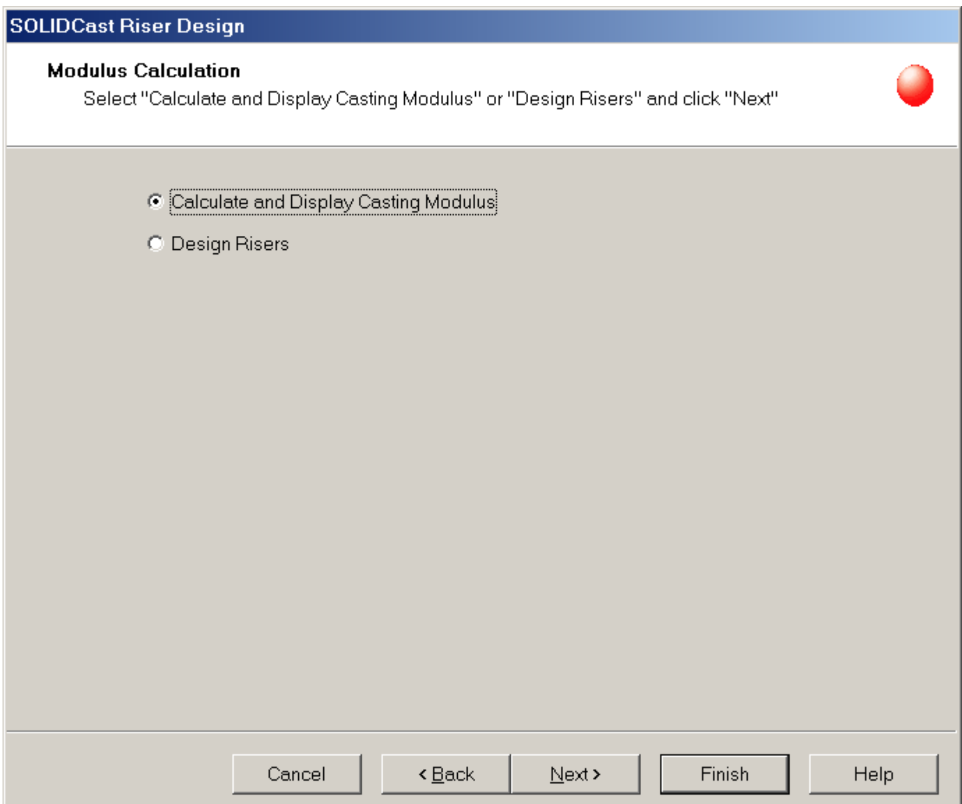
1. Calculate and display modulus values throughout the casting.

2. Calculate and display separate feeding areas within the casting, and design a riser for each of these areas.

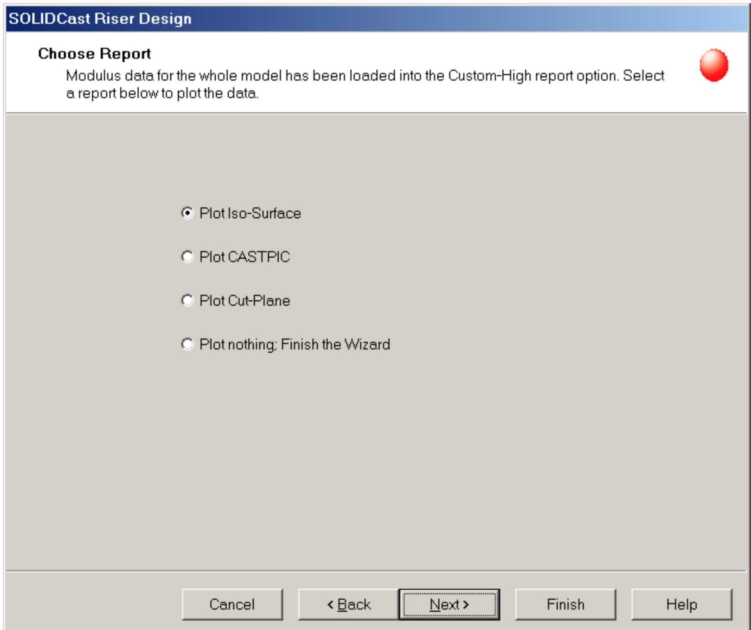
CALCULATE AND DISPLAY CASTING MODULUS

This feature allows you to view modulus values throughout the casting. This can be useful as a way of visualizing which portions of the casting will freeze in which order, and identifying which sections of the casting are heaviest from a thermal standpoint. This information can also be of value for process information; for example, in some iron casting processes, the modulus value is part of determining how much expansion might occur, and whether a given casting is a good candidate for production as a riserless casting.

To view modulus values, on the window click the button labeled **Calculate and Display Casting Modulus**. The window should change so that it appears as follows:

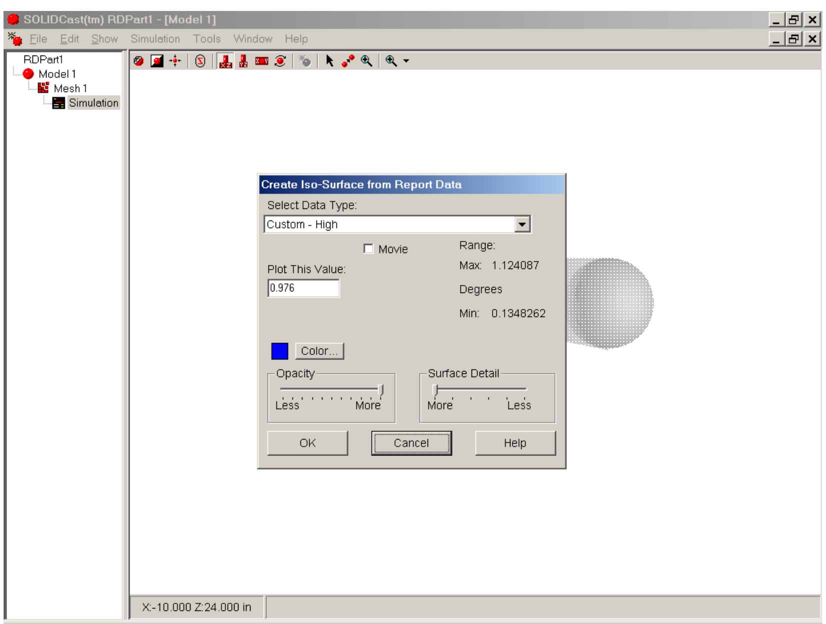


Now, click the **Next** button. The modulus values within the casting will be calculated, and the following window will appear:

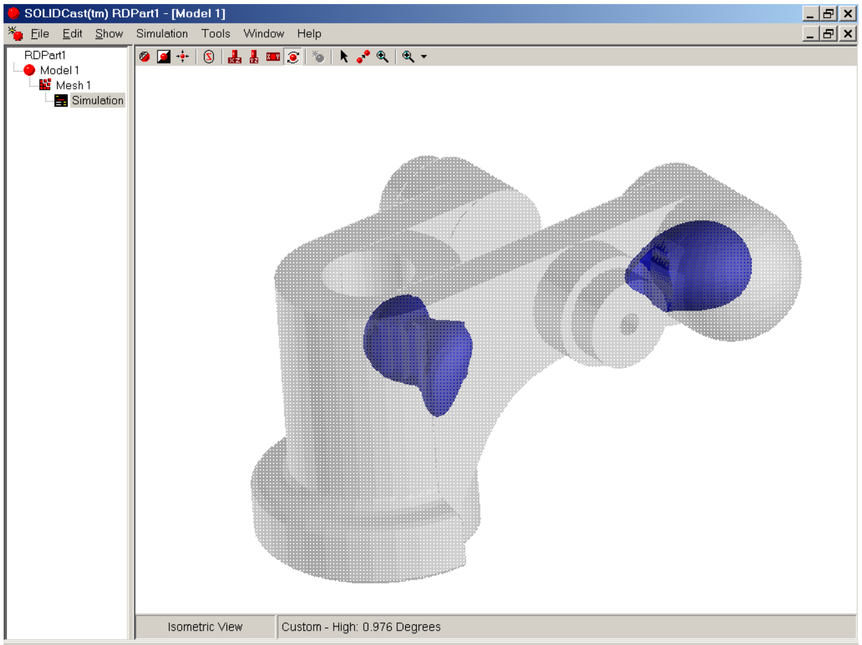


This window allows you to select which option you want to use to plot the values of modulus that have just been calculated. Probably the most convenient option to view these values is the Iso-Surface plot type, although you can select CASTPIC or Cut-Plane also if you prefer. Once you have selected the type of plot, click the **Next** button.

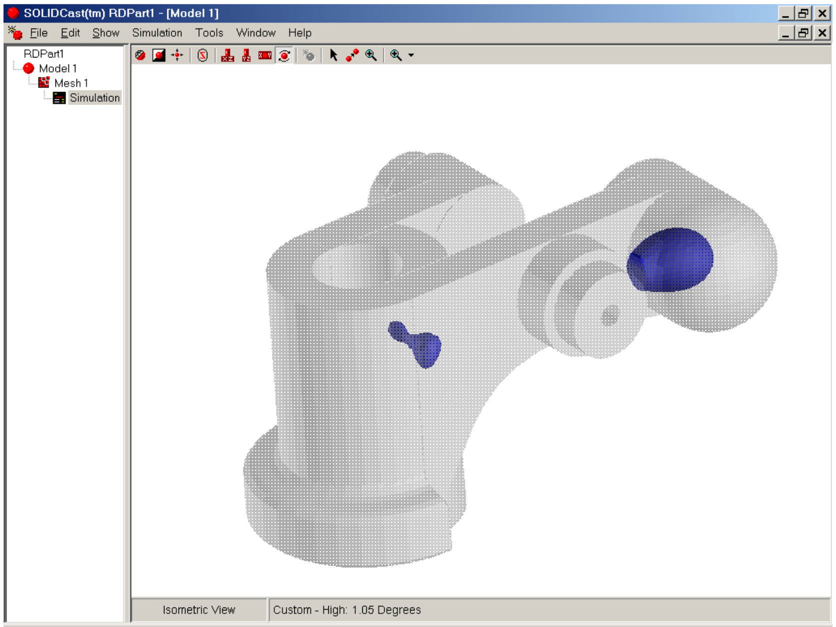
The following screen should appear (this will vary depending on the type of plot that you have selected):



This is a standard Iso-Surface type of plot that you could select manually as a normal SOLIDCast function. Notice that the modulus data has been placed into the Custom-High criterion function. Notice also that the system has suggested a plot value for modulus plotting. If we just click the **OK** button, this value will be plotted as follows:

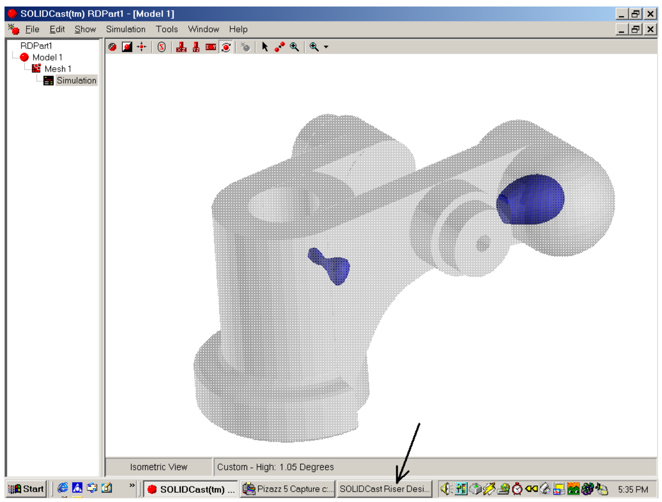


This shows two obvious heavy areas within the casting. If we plot a higher value of modulus (close to the maximum value), the areas of very highest modulus values will be shown. This is done in the normal way, selecting **Simulation** from the menu at the top, and then selecting **Plot Iso Surface**, entering the value to plot, and clicking on the **OK** button. Doing this for a higher value of modulus will show the following:

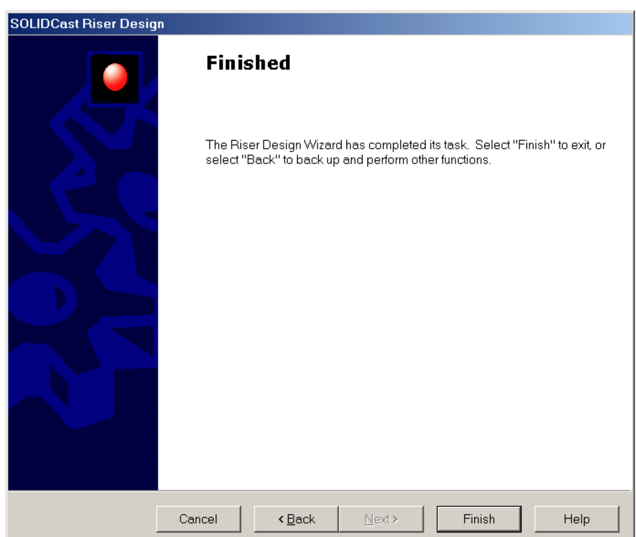


This allows us to plot various values to examine what might be happening in this casting. Keep in mind that we are plotting modulus values, which indicate the heaviness of various sections within the casting and the order in which these sections will likely solidify.

How do you get back to the **Riser Design Wizard** after plotting various values of modulus in SOLIDCast? Notice on the Windows Taskbar that there is a button labeled **SOLIDCast Riser Design Wizard**. This means that the wizard is still active while you are plotting.



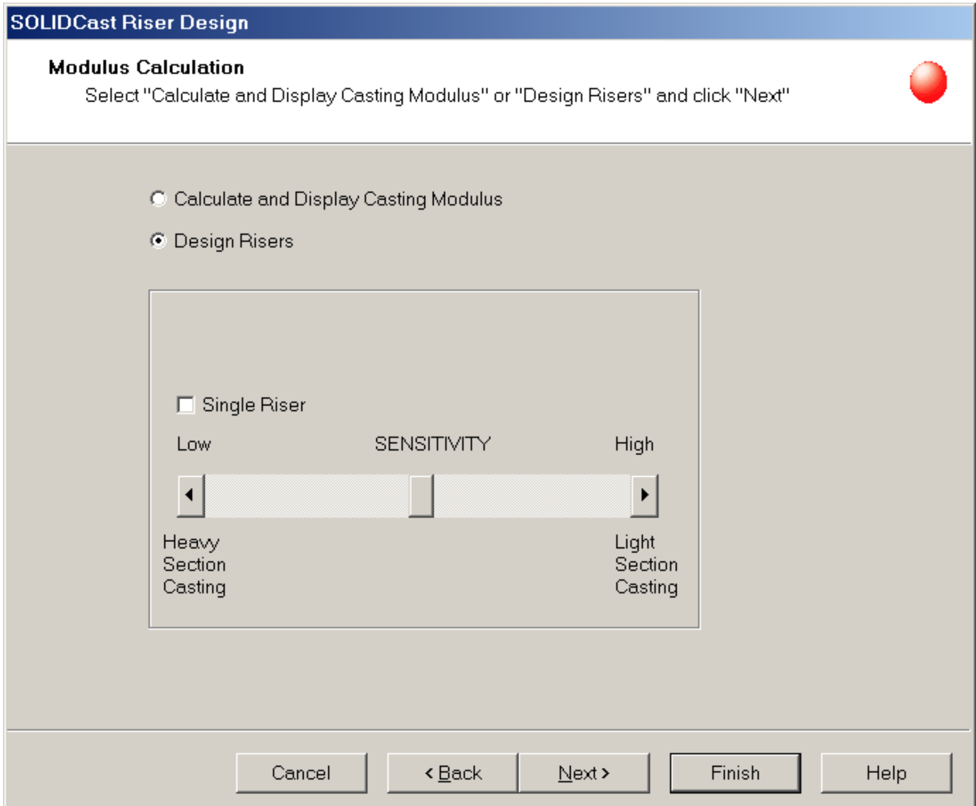
To get back to the wizard, simply click on the button on the **SOLIDCast Riser Design Wizard** Taskbar as shown. This will restore the wizard on the screen, and it will appear as follows:



Now, to perform other tasks with the **Riser Design Wizard**, we can just click the **Back** button until we get to the desired screen. If you wanted to exit the **Riser Design Wizard** at this point, you could click the **Finish** or **Cancel** buttons.

DESIGNING RISERS

As mentioned in the previous section, the first screen that you see after the initial Introduction screen for the **Riser Design Wizard** is the following:



The previous section showed us how to calculate and display the casting modulus. Now we will learn to use the **Riser Design Wizard** to actually design risers for a casting. This is done by selecting the option titled **Design Risers** as shown above.

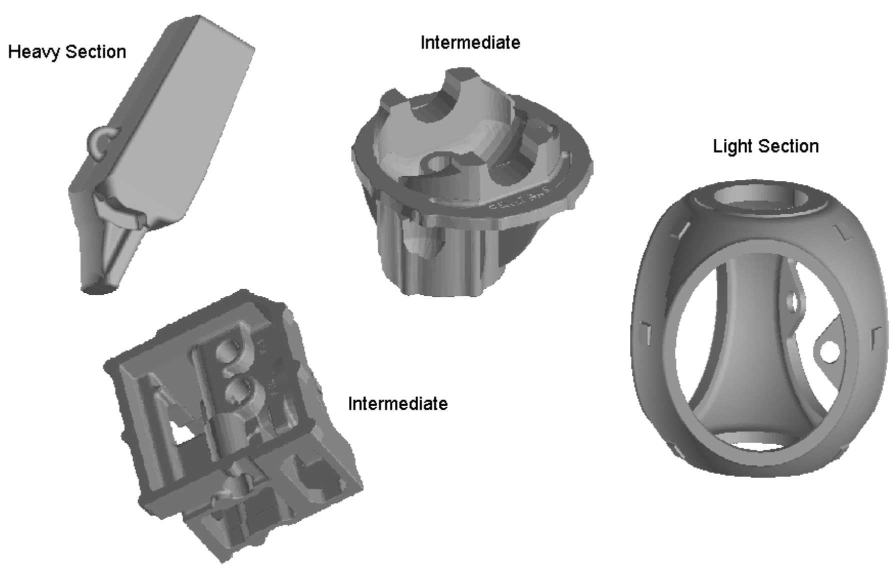
When the **Design Risers** option is selected, the **Riser Design Wizard** will first perform a calculation of modulus values. It then uses pattern-recognition software to identify separate feeding areas within the casting (consisting of isolated areas of higher modulus values), each of which may require a separate riser.

You will notice that on the **Design Risers** screen there is a slider bar labeled SENSITIVITY. This bar determines how sensitive the pattern-recognition software is in identifying isolated areas requiring feeding. There is also a check box which is labeled Single Riser. These selections will determine how many risers are identified as being required for a casting, as follows:

- When the slider bar is moved to the left, this indicates LOW SENSITIVITY. At this setting, only larger isolated areas will be identified as being separate feed areas requiring a riser. Very small isolated areas will be ignored. This setting would typically be used for **Heavy Section Castings**.
- When the slider bar is moved to the right, this indicates HIGH SENSITIVITY. At this setting, even very small isolated areas will be identified as being separate feed areas requiring a riser. This setting would typically be used for **Light Section Castings**.
- When **Single Riser** is selected, the system will consider only one riser for the entire casting, regardless of how many separate feed areas occur within the casting. This selection should be used with some caution.

It may take some experimentation with castings of various types to determine the appropriate settings for the castings that you are producing. One approach is to try analyzing a casting at both the LOW SENSITIVITY and HIGH SENSITIVITY settings to determine if there is a difference. With some castings, the results will be the same (same number of risers), with other castings the results will be different.

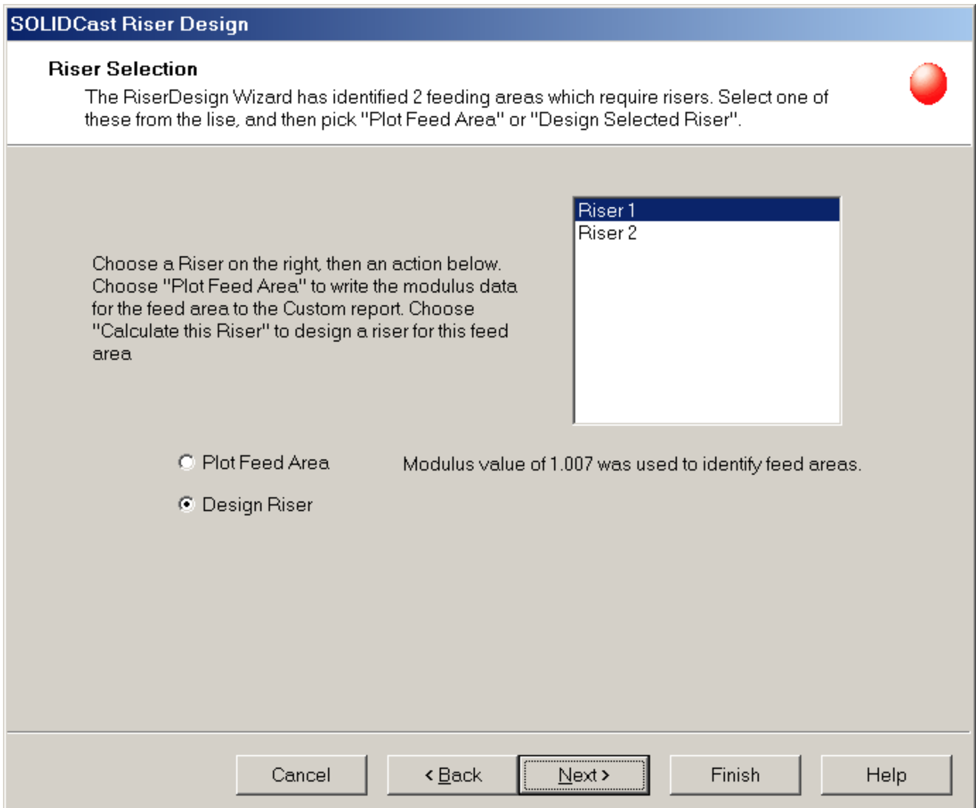
The figure below shows some examples of castings that might be classified as heavy section, intermediate, and light section:



It must be emphasized that, in all cases, after designing risers with the Riser Design Wizard, a simulation of the casting and risers should be run with SOLIDCast in order to verify the design.

Designing risers in this way is only an approximation in order to get a starting point for the final design; only by running a full simulation can you verify whether this design is appropriate or whether further modifications to the casting design or riser design are necessary.

Once you have selected the settings for analysis of your casting, click the **Next** button on the **Riser Design** screen. You will see a wizard screen similar to the following:



This screen indicates that, at the settings that were used on the previous screen, the **Riser Design Wizard** has determined that there are 2 separate, isolated feeding areas within the casting requiring risers. This number, of course, will vary from one casting to another, and also with the settings used on the previous screen.

Note that on this screen, there is a message which says **Modulus value of 1.007 was used to identify feed areas.** This means that the pattern recognition software determined that the isolated feeding areas could be seen most clearly by plotting a modulus value of 1.007. This indicates that you could visualize these isolated areas by having the system calculate the modulus and then plot a value of 1.007 (see the previous section on CALCULATING AND DISPLAYING MODULUS VALUES).

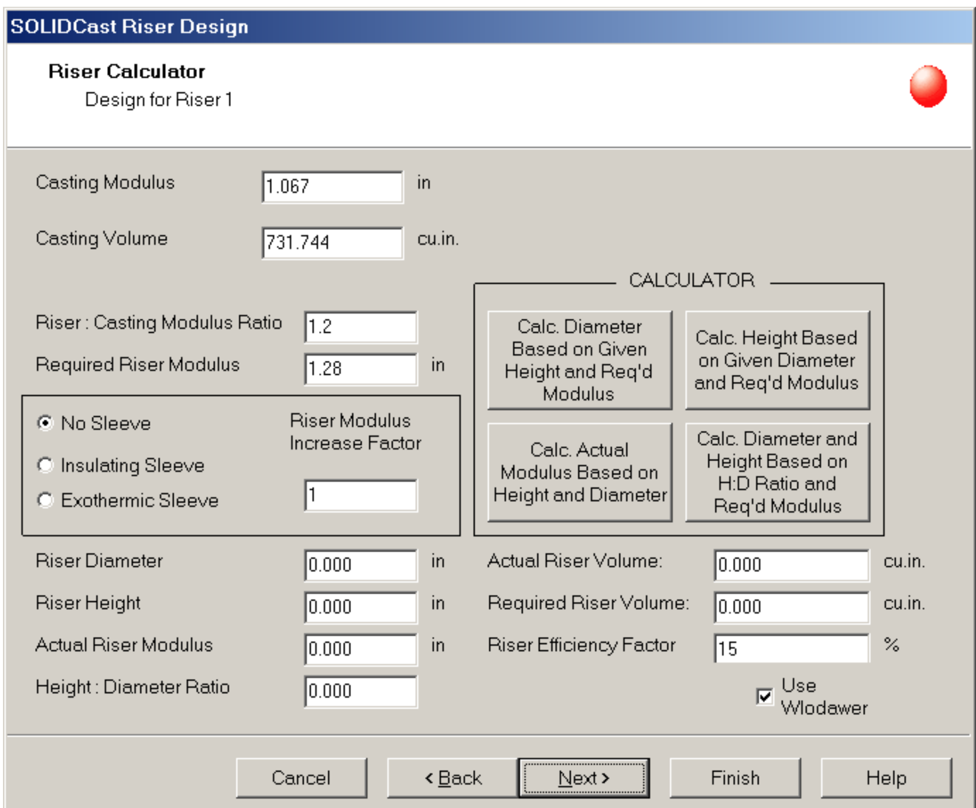
At this point, we have two options:

1. We can view the model to determine where these feeding areas are located.
2. We can calculate the appropriate riser size based on modulus and volume requirements.

We will examine the second option (riser size calculation) first, as this is the option which is selected by default.

CALCULATING RISER SIZE

To calculate the size for a riser, select one of the risers shown above and highlight it (this is done with a single click of the mouse on Riser 1, Riser 2, or whichever). You should see a screen appear as follows:



This is the **SOLIDCast Riser Calculator**. There are a variety of displayed values and calculation buttons. The meaning of each of these is as shown on the following pages.

Casting Modulus

This is the maximum modulus of the casting *within the feeding area which this riser is intended to feed*. It is NOT necessarily the maximum modulus within the casting. Normally, you would want the riser to have a modulus value equal to or greater than this Casting Modulus, for proper directional solidification.

Casting Volume

This is the volume of the casting *within the feeding area which this riser is intended to feed*. This will be an important consideration in the design of this riser. The riser must be able to provide enough volume of feed metal to compensate for the contraction which will occur in this casting volume.

Riser:Casting Modulus Ratio

This indicates the ratio of the modulus of the riser to the modulus of the casting. A typical value would be 1.2. The wizard initially defaults to this value. You may enter a different value if you want. Remember, for proper functioning of the riser, its modulus should normally be equal to or greater than that of the portion of the casting to which it is attached.

Required Riser Modulus

This is a calculated field giving the modulus that you will require for this riser. Its formula is:

Casting Modulus X Riser:Casting Modulus Ratio

Riser Modulus Increase Factor

This is a factor which is to be applied to the riser modulus based on whether the riser is unsleeved, has an insulated sleeve, or an exothermic sleeve. The concept here is that a sleeve acts to insulate the riser and reduce heat transfer to the mold. The effect is as though the surface area of the riser has been reduced, thus the “effective” modulus of the riser has been increased. Riser Design uses a set of built-in factors, depending on whether you select **No Sleeve**, **Insulating Sleeve** or **Exothermic Sleeve**. You can also enter your own factor into this field if you don’t wish to use the built-in factors.

Riser Diameter

This is the diameter of a cylindrical riser. It can be the result of a calculation, or it can be entered manually (see the following section on Calculation Buttons).

Riser Height

This is the height of a cylindrical riser. It can be the result of a calculation, or it can be entered manually (see the following section on Calculation Buttons).

Actual Riser Modulus

This is the actual modulus calculated for a riser with a given height and diameter.

Height:Diameter Ratio

This is the ratio of the height to the diameter the riser. It can be the result of a calculation, or it can be entered manually (see the following section on Calculation Buttons).

Actual Riser Volume

This is the calculated volume of the riser, given its height and diameter.

Required Riser Volume

This is the volume which is required for this riser, given the casting volume, the amount of contraction of the alloy (measured from the volume change curve at the CFS point) and the efficiency of the riser.

Riser Efficiency Factor

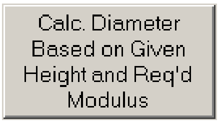
This is the stated efficiency for the riser, which is a measure of the percent of liquid metal that the riser is able to supply to the casting. This can be based on the AFS/CMI curves for riser type, or on the Wlodawer approach (see the section on BASIC CONCEPTS). You can also enter your own factor depending on the specific type of riser you are using, and your experience. For example, there are some exothermic "mini-risers" which are stated to have an efficiency of 80%; you could enter 80 here to perform calculations for this type of riser. When you select a specific type of riser (**No Sleeve**, **Insulating Sleeve** or **Exothermic Sleeve**) the efficiency is filled in for you; the value depends on whether you have elected to use the AFS/CMI approach or the Wlodawer approach. As stated, however, you can enter a different efficiency if you like.

Use Wlodawer

This check box allows you to select whether you want to use the Wlodawer approach or the AFS/CMI approach for the Riser Efficiency Factor. If this is checked, then typical values for the Wlodawer approach will be placed in the Riser Efficiency Factor box. If this is not checked, then values derived from the AFS/CMI curves will be used (see the section on BASIC CONCEPTS).

Calculation Buttons

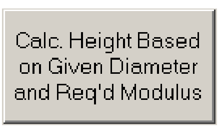
These buttons perform the actual “work” of designing risers in the **Riser Design Wizard.** They function as follows:



Enter **Riser Height.** The system will calculate the riser size based on the **Required Riser Modulus** and display:

Riser Diameter
Actual Riser Modulus
Height:Diameter ratio
Actual Riser Volume
Required Riser Volume

*Be sure to check the **Actual Riser Volume** against the **Required Riser Volume**. If the **Actual Riser Volume** is less than **Required Riser Volume**, then a message will appear which indicates **RISER VOLUME TOO SMALL**. In this case, you will need to increase the riser dimensions.*



Enter **Riser Diameter.** The system will calculate the riser size based on the **Required Riser Modulus** and display:

Riser Height
Actual Riser Modulus
Height:Diameter ratio
Actual Riser Volume
Required Riser Volume

*Be sure to check the **Actual Riser Volume** against the **Required Riser Volume**. If the **Actual Riser Volume** is less than **Required Riser Volume**, then a message will appear which indicates **RISER VOLUME TOO SMALL**. In this case, you will need to increase the riser dimensions.*

Calc. Actual Modulus Based on Height and Diameter

Enter **Riser Height** and **Riser Diameter**. The system will calculate the riser modulus and volume and display:

> **Actual Riser Modulus**
> **Height:Diameter ratio**
> **Actual Riser Volume**
> **Required Riser Volume**

*Be sure to check the **Actual Riser Volume** against the **Required Riser Volume**. If the **Actual Riser Volume** is less than **Required Riser Volume**, then a message will appear which indicates **RISER VOLUME TOO SMALL**. In this case, you will need to increase the riser dimensions.*

Calc. Diameter and Height Based on H:D Ratio and Req'd Modulus

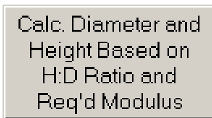
Enter **Height:Diameter Ratio**. This is entered as a single number, for example 1, 1.5 or 2. The system will calculate the riser size based on the **Required Riser Modulus** and display:

> **Riser Diameter**
> **Riser Height**
> **Actual Riser Modulus**
> **Height:Diameter ratio**
> **Actual Riser Volume**
> **Required Riser Volume**

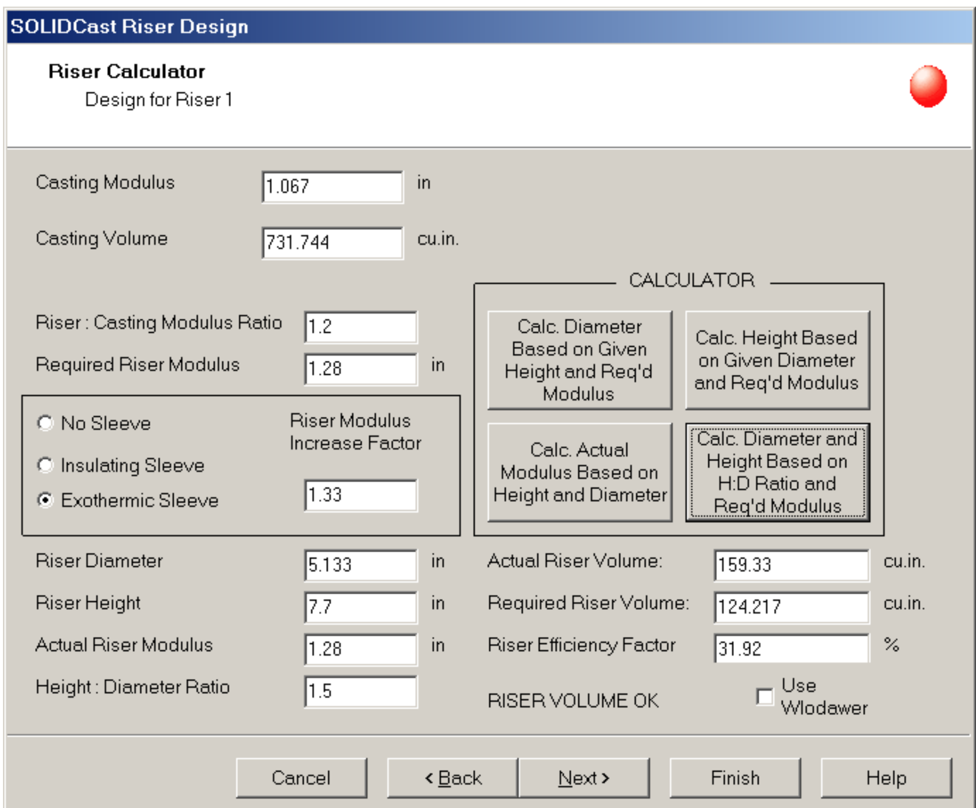
*Be sure to check the **Actual Riser Volume** against the **Required Riser Volume**. If the **Actual Riser Volume** is less than **Required Riser Volume**, then a message will appear which indicates **RISER VOLUME TOO SMALL**. In this case, you will need to increase the riser dimensions.*

Example #1:

Select **Exothermic Sleeve**
AFS/CMI curves used for **Riser Efficiency Factor** (**Use Wlodawer** turned off)
Enter Height:Diameter ratio of 1.5
Press



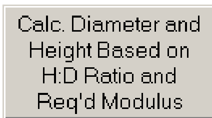
The result appears as follows:



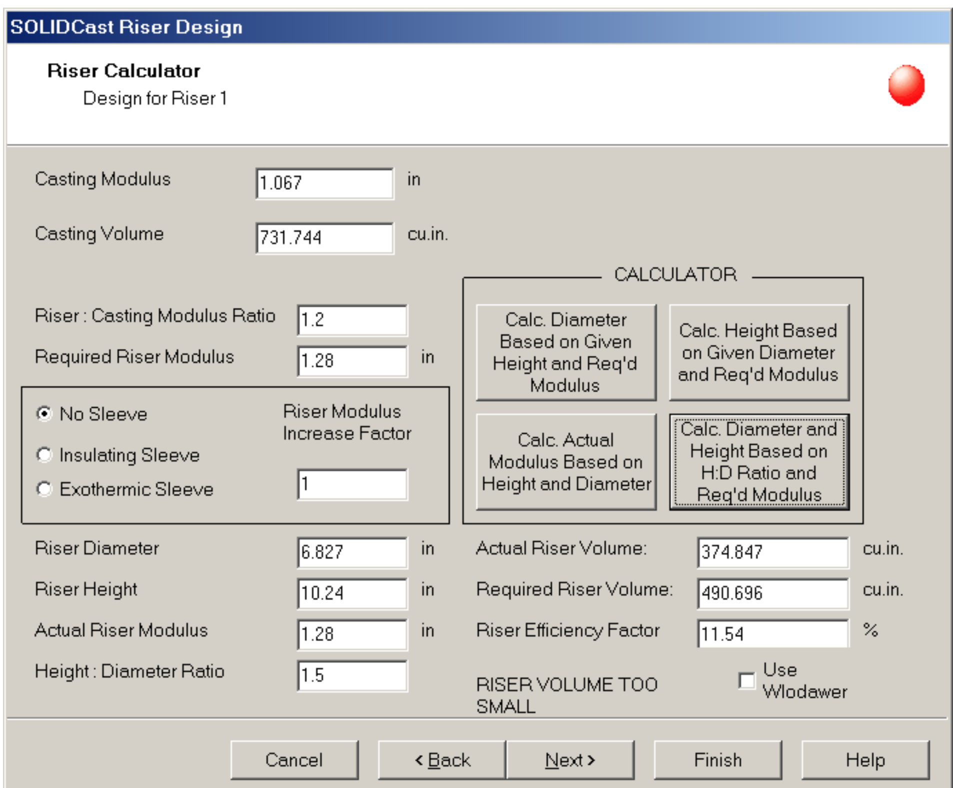
This gives us the **Riser Diameter** and **Riser Height** for this riser. Note that the message **RISER VOLUME OK** appears in the lower right corner. This indicates that the **Actual Riser Volume** is greater than the **Required Riser Volume**.

Example #2:

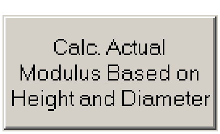
Select **No Sleeve**
AFS/CMI curves used for **Riser Efficiency Factor** (**Use Wlodawer** turned off)
Enter Height:Diameter ratio of 1.5
Press



The result appears as follows:

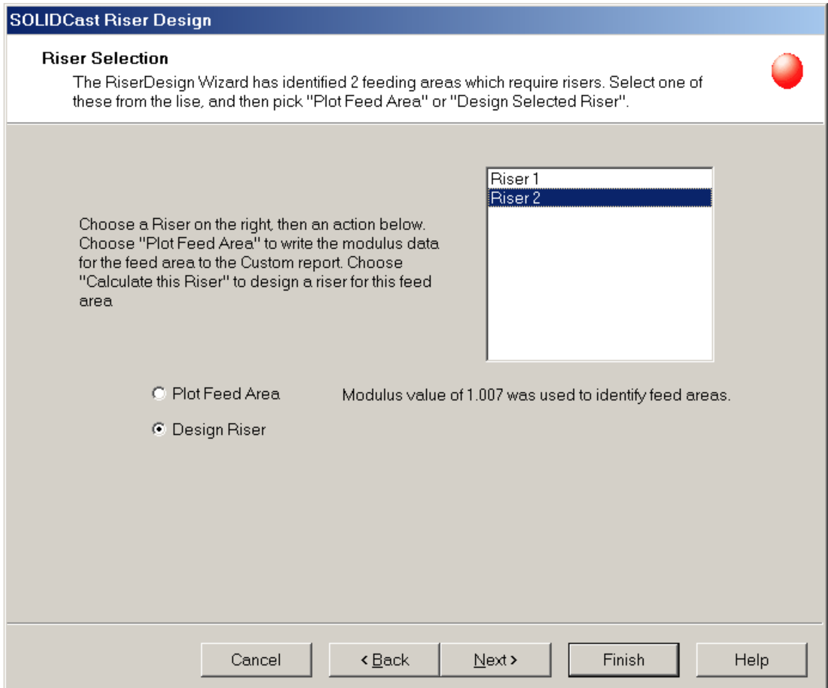


This gives us the **Riser Diameter** and **Riser Height** for this riser. However, note that the message **RISER VOLUME TOO SMALL** appears in the lower right corner. This indicates that the **Actual Riser Volume** is less than the **Required Riser Volume**. At this point, it would be necessary to increase the riser size with a larger diameter or height (or both), and then click:

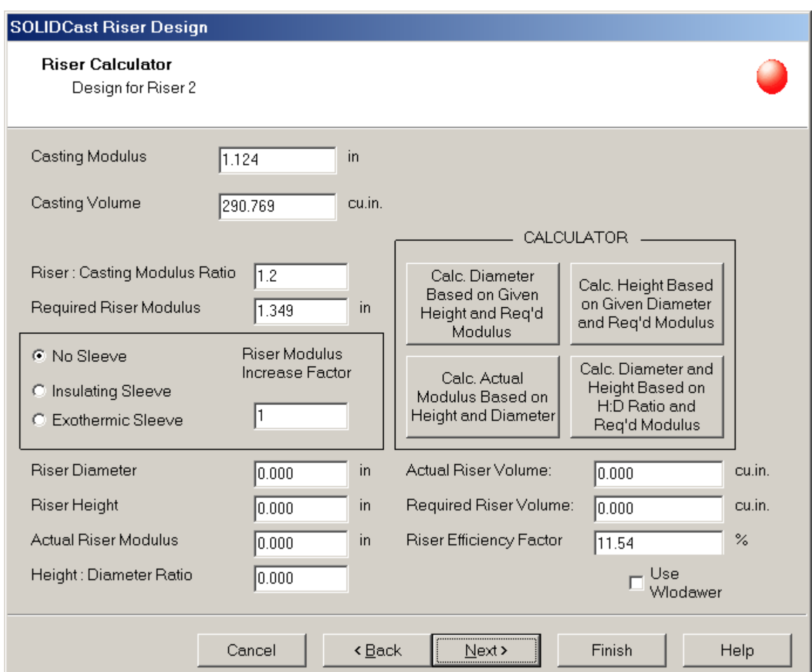


This will determine whether the riser volume is now adequate.

After performing this calculation for Riser 1, we can press the **Back** button to return to the previous screen. Here, we select Riser 2 as shown …



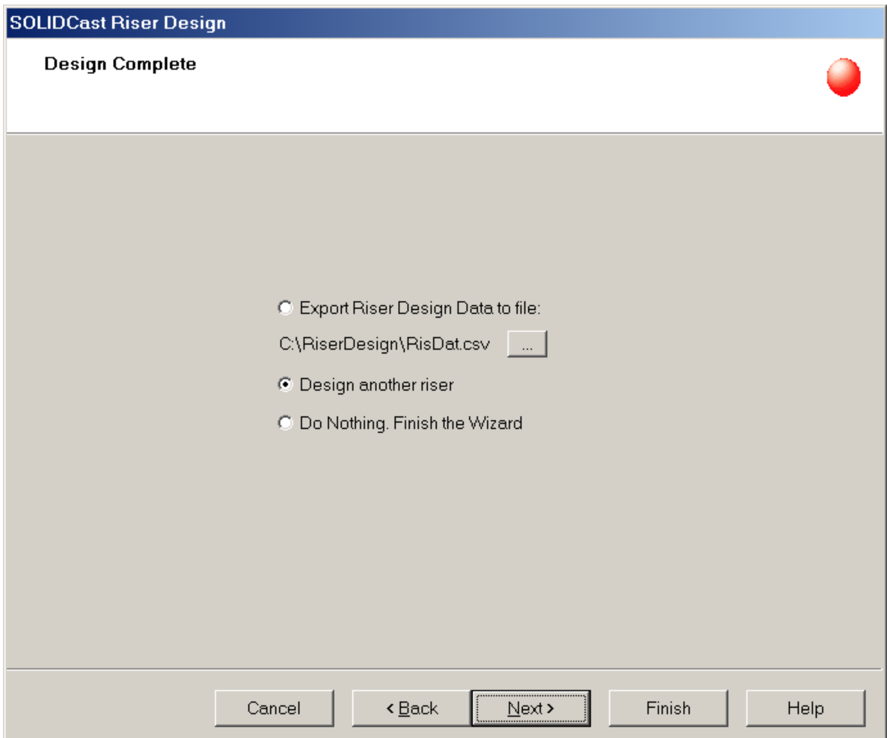
…then click on the **Next** button to perform design calculations for Riser 2. In this case, we would again see a screen as follows:



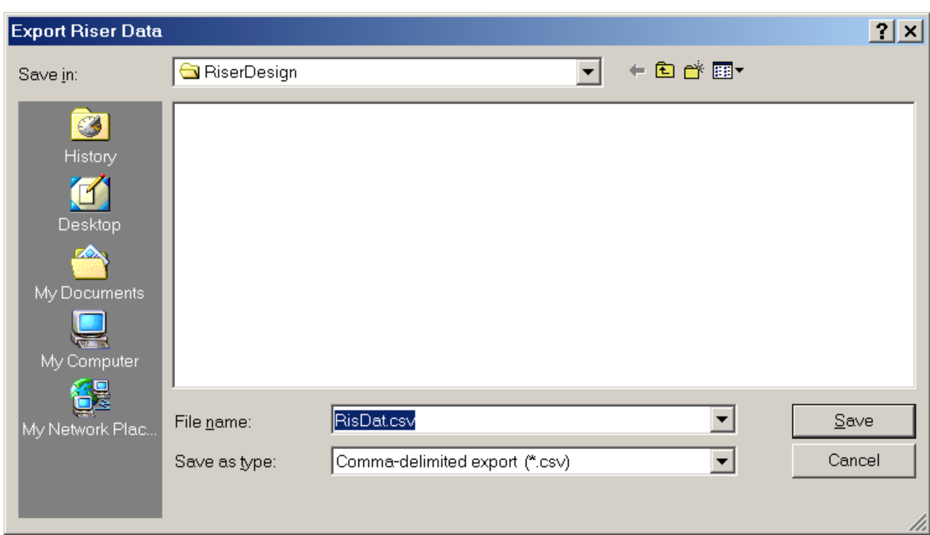
From this screen, riser calculations can be performed for Riser 2.

By selecting the **Next** and **Back** buttons, it is possible to perform calculations for each riser which has been identified by the **Riser Design Wizard**.

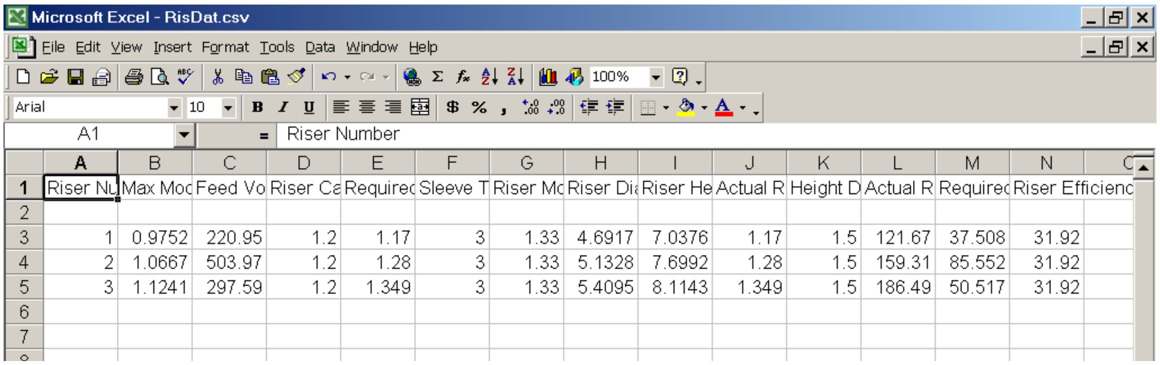
When calculations have been performed for the last riser, then from the **Riser Calculator** screen (the screen shown above) you can press the **Next** button. The following will appear:



Using this screen, you can export your riser data to a file which can then be loaded into an Excel spreadsheet. You can pick the file name by clicking on the small button with three dots (…). The following will appear:

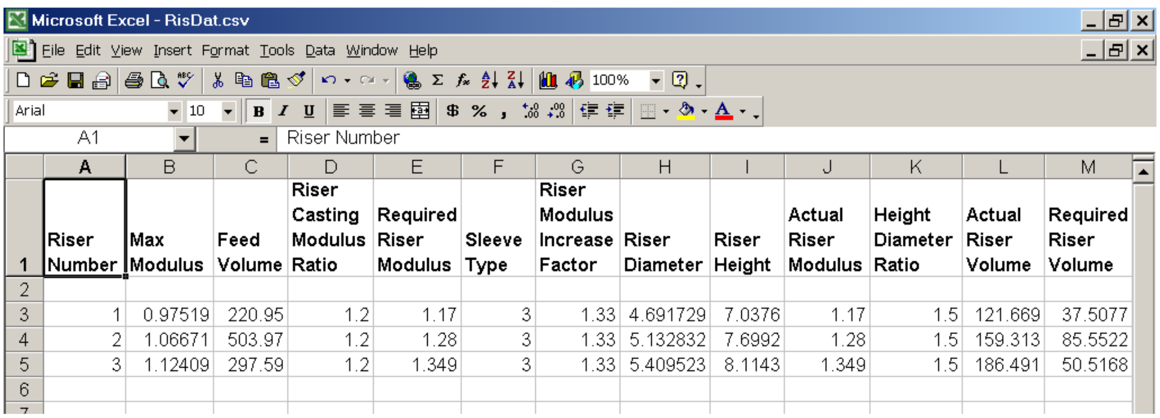


Enter the name in the **File name** field and click the **Save** button. This will create a type of file called a Comma Separated Variable file (.csv). This file can be loaded directly into Excel or other spreadsheet programs. For example, in this case, starting Excel and loading the file called RisDat.csv would display the following data:



	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	Riser Nu	Max Moc	Feed Vo	Riser Ca	Requirec	Sleeve T	Riser Mc	Riser Di	Riser He	Actual R	Height D	Actual R	Requirec	Riser Efficienc
2														
3	1	0.9752	220.95	1.2	1.17	3	1.33	4.6917	7.0376	1.17	1.5	121.67	37.508	31.92
4	2	1.0667	503.97	1.2	1.28	3	1.33	5.1328	7.6992	1.28	1.5	159.31	85.552	31.92
5	3	1.1241	297.59	1.2	1.349	3	1.33	5.4095	8.1143	1.349	1.5	186.49	50.517	31.92

Slightly modifying the title cells for readability, this can appear as follows:



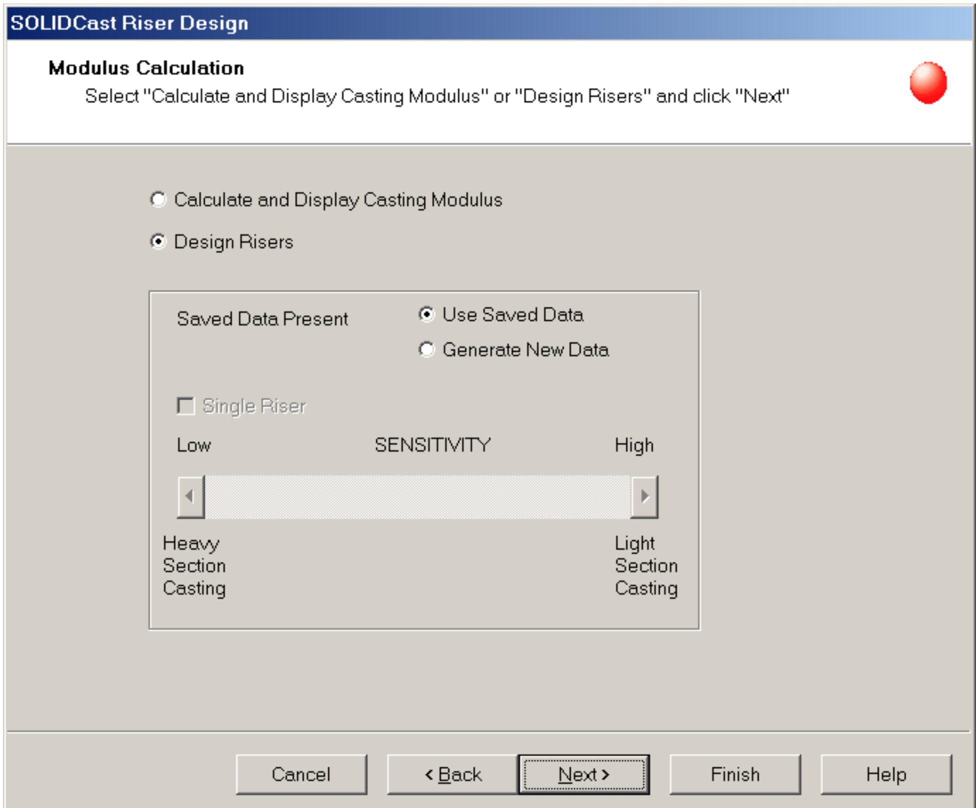
	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Riser Number	Max Modulus	Feed Volume	Riser Casting Modulus Ratio	Required Riser Modulus	Sleeve Type	Riser Modulus Increase Factor	Riser Diameter	Riser Height	Actual Riser Modulus	Height Diameter Ratio	Actual Riser Volume	Required Riser Volume
2													
3	1	0.97519	220.95	1.2	1.17	3	1.33	4.691729	7.0376	1.17	1.5	121.669	37.5077
4	2	1.06671	503.97	1.2	1.28	3	1.33	5.132832	7.6992	1.28	1.5	159.313	85.5522
5	3	1.12409	297.59	1.2	1.349	3	1.33	5.409523	8.1143	1.349	1.5	186.491	50.5168

You can then save this as a normal Excel spreadsheet (.xls file) for later reference.

Note: The Riser Type number in this spreadsheet refers to the following:

Type 1 = No Sleeve
Type 2 = Insulating Sleeve
Type 3 = Exothermic Sleeve

You will find that if you have performed Riser Design calculations for a simulation, the **Riser Design Wizard** will retain these calculations. The next time that you activate the **Riser Design Wizard** for this simulation, you will see the following screen:



Notice that this screen contains the message **Saved Data Present**, and allows you to select whether you want to use the saved data from the previous calculations.

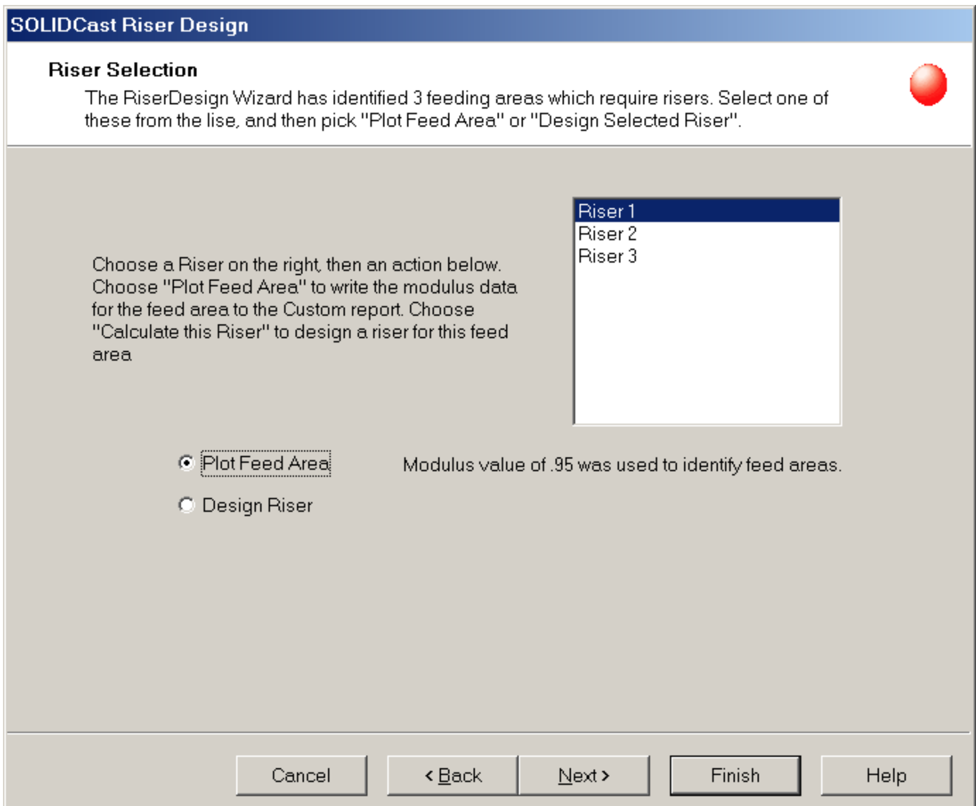
If you select **Use Saved Data**, then the design calculations as they were previously performed will be displayed.

If you select **Generate New Data**, then you will be able to perform a new set of calculations using different settings, if desired.

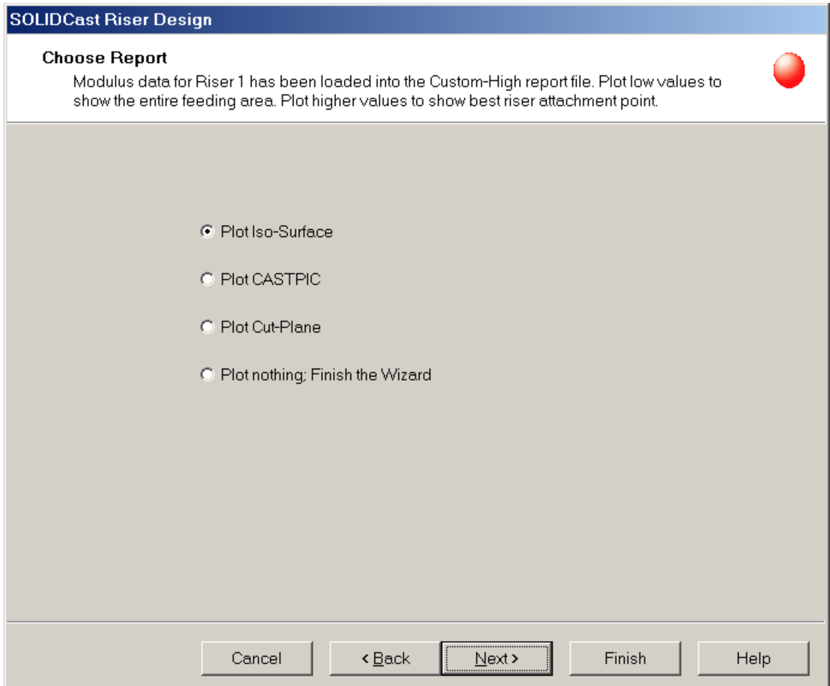
PLOTTING AND VIEWING FEED AREAS AND RISER LOCATIONS

Returning to the **Riser Selection** screen (after selecting **Design Risers**), note that you have two choices: **Plot Feed Area** or **Design Riser**.

We previously discussed the choice labeled **Design Riser**, which allows you to perform calculations for riser size. The **Riser Design Wizard** can also plot each area within the casting which has been designated as needing a riser. To perform this function, you should select a riser, then select **Plot Feed Area** and click on the Next button. In the example below, the **Riser Design Wizard** has decided that three risers are required. We select Riser 1, select **Plot Feed Area** and then click on the **Next** button.

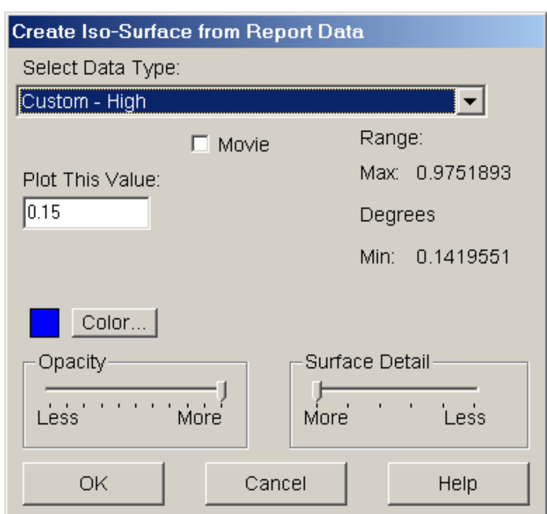


This will cause the following screen to appear:



The **Plot Feed Area** function actually places modulus data into ONLY the portion of the casting that is in the feed area for the selected riser. Therefore, we can plot this data and view which portion of the casting is to be fed by this riser.

Using the above screen, we can select one of the standard SOLIDCast plot functions to view this data. Here we have selected **Plot Iso-Surface**. We then click the **Next** button. This causes the standard SOLIDCast Iso-Surface dialogue box to appear, as follows:

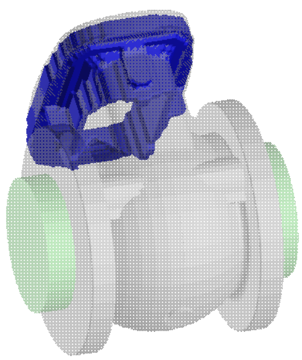


Note that the data is placed in the Custom-High criterion function.

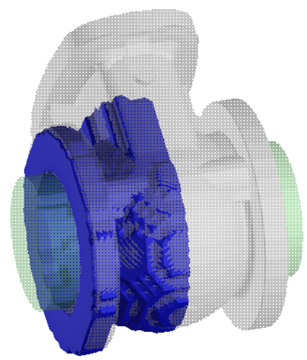
The data that is placed into the Custom-High criterion function is modulus data. This data is placed ONLY in the feeding area of the selected riser. There is no data in the other portions of the casting. This means that we have the ability to do two things:

1. By plotting a LOW value of the modulus data, we can see the entire feeding area.
2. By plotting a HIGH value of the modulus data, we can see the last area to solidify in that particular region of the casting. This will give us an indication as to where the riser attachment point should be placed.

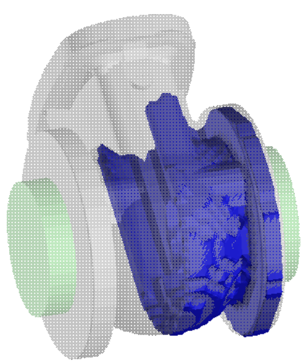
As an example of visualizing feeding areas by plotting LOW values of modulus, consider the following valve body casting. The **Riser Design Wizard** indicated that three risers are required for this casting. Plotting a low value of modulus in each feeding area shows which area of the casting will be fed by each riser, as follows:



Feeding Area: Riser 1



Feeding Area: Riser 2

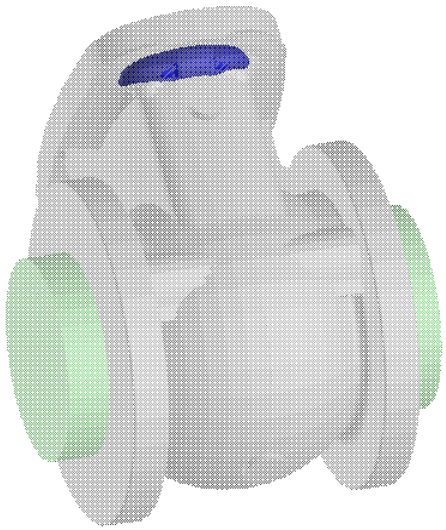


Feeding Area: Riser 3

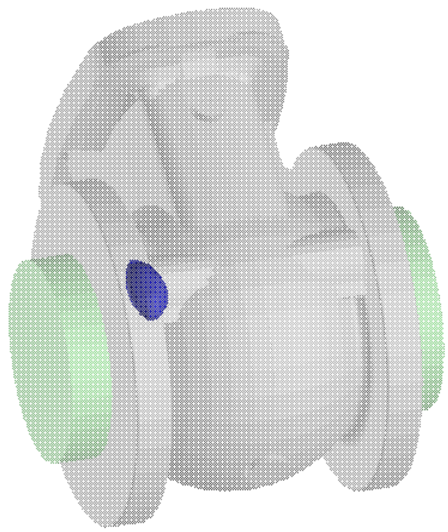
To plot each successive area, just use the **Next** and **Back** buttons, select each riser, and then plot its feeding area.

When **Plot Feed Areas** activates the Iso-Surface plotting function, it automatically places a LOW value in the Plot Value field so that you can view the entire feeding area.

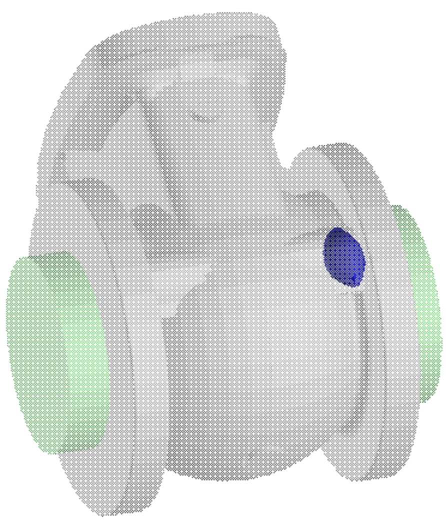
The next function that would typically be performed with **Plot Feed Areas** would be to plot modulus values that are close to the maximum. This shows where the last points to solidify would be located, and typically is a good indication where the riser attachment points should be. For this same casting, plotting the higher modulus values appears as follows:



Area of Highest Modulus: Riser 1

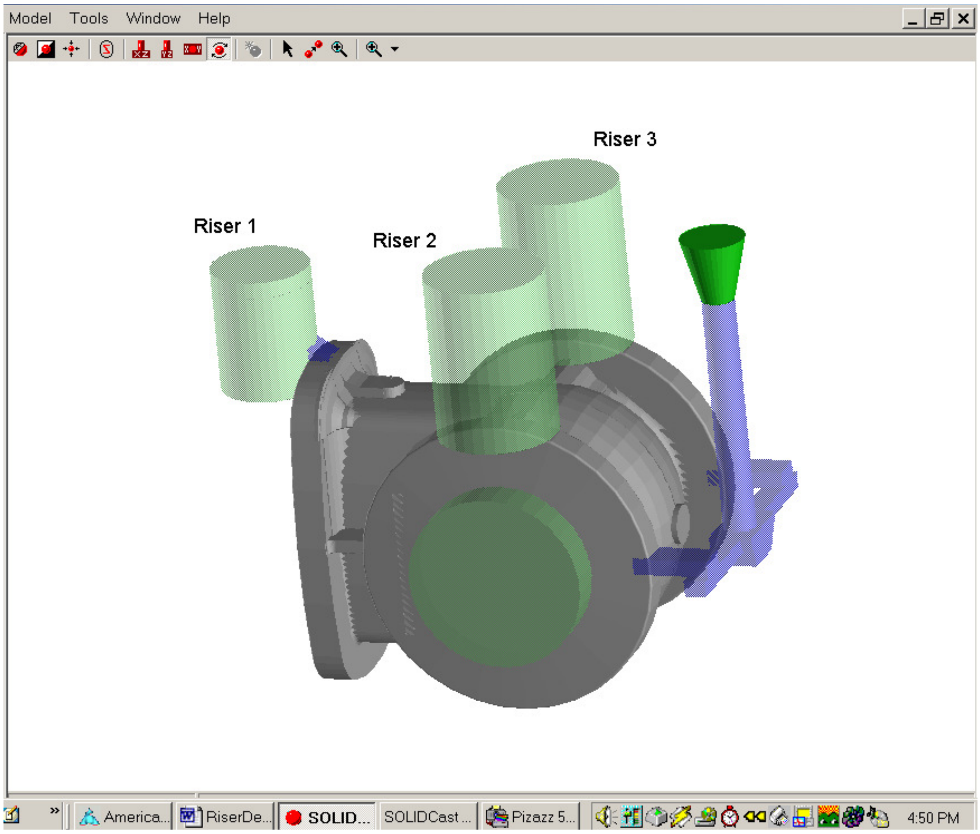


Area of Highest Modulus: Riser 2



Area of Highest Modulus: Riser 3

Based on this information, and on riser sizes as calculated by the Riser Calculator, risers can be placed on the SOLIDCast model and a verification simulation can be run. The final design model for this casting would appear in SOLIDCast as follows:



FINAL NOTES

The **Riser Design Wizard** can be very useful for understanding where risers should be placed and what size they should be to properly feed a casting.

A couple of additional notes:

1. If you are using Planes of Symmetry in a model, then the volume calculations required for proper riser design may not be accurate due to the fact that all of the volume of the casting may not be accounted for across the Plane of Symmetry. It is recommended that you perform a Mirror function prior to starting the **Riser Design Wizard.**

2. In a situation where there is one large area with a high modulus (for example, in a ring-shaped casting) you may want to feed this area with multiple risers. In this case, the casting can be sized using the calculated modulus, but the required volume which is displayed should be divided by the number of risers to calculate the required volume per riser.