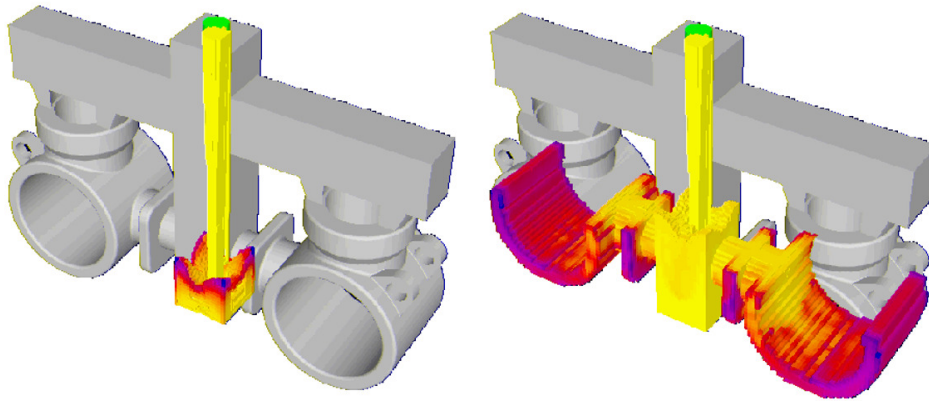


# FLOW CAST

CFD-Based Fluid Flow Analysis Software



Training Course Workbook  
Copyright © 2011



Powered by Altair Engineering's flow simulation technology  
FLOWCast Version 1.74  
SOLIDCast Version 8.1.1

COURSE SYLLABUS

---

- UNIT 1: Overview and Quick Start Guide
- UNIT 2: System Installation and System Considerations
- UNIT 3: Setting Up a Fluid Flow Simulation Run
- UNIT 4: The FLOWCast Toolbar
- UNIT 5: The File Menu
- UNIT 6: The Models Menu
- UNIT 7: The Graphics Menu
- UNIT 8: The Settings Menu
- UNIT 9: The Help Menu
- UNIT 10: Display and View Controls
- UNIT 11: The Display Tab
- UNIT 12: The Filling Tab
- UNIT 13: The Status Tab
- UNIT 14: The Anim Tab
- UNIT 15: The Pathlines Tab
- UNIT 16: Flow Simulation Strategies

## UNIT 1: Overview and Quick Start Guide

---

### Q: What is FLOWCast™?

A: FLOWCast is the fluid flow modeling module for mold filling simulation, which works in conjunction with the SOLIDCast™ modeling system. FLOWCast was developed using flow simulation technology from Altair Engineering, Inc. and is a joint product of Altair Engineering and Finite Solutions.

The FLOWCast module must be purchased separately. After purchasing and installing FLOWCast, you will need to send FLOWCast System Codes to Finite Solutions, Inc. and receive Response Codes to activate the program.

### Q: What can FLOWCast do?

A: FLOWCast simulates the flow of liquid metal into a mold, along with the cooling of the metal and the heating of the mold. FLOWCast has two “modes”:

**Quick Simulation:** This is a fast-filling algorithm, which is not full CFD, but uses a “cascade” type of fill simulation to quickly fill a mold. The Quick Simulation uses less memory and runs faster than the Full Simulation, but it does not provide the same levels of information and accuracy.

**Full Simulation:** This is a full CFD simulation of fluid flow, taking into account momentum, pressure and velocity of the molten metal as well as viscous losses at the mold wall.

### Q: What is CFD?

A: CFD stands for Computational Fluid Dynamics, which refers to a numerical solution of the Navier-Stokes equations that govern fluid flow.

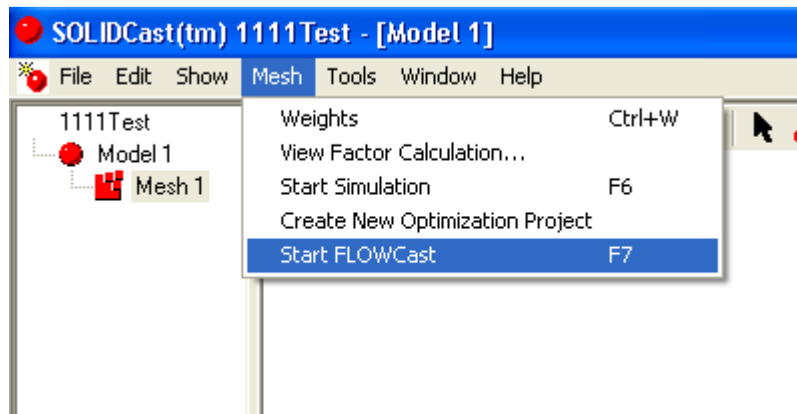
### Q: Does FLOWCast use the same models as SOLIDCast?

A: Yes. FLOWCast uses exactly the same models that SOLIDCast uses. You build your models in the same way, and mesh them the same way. FLOWCast uses the meshed SOLIDCast model to run its simulation.

**Q: How do I run FLOWCast?**

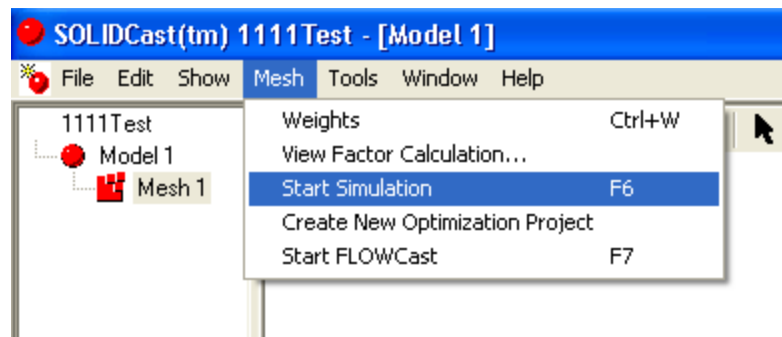
A: First of all, you build your model (with Fill Material) and mesh it just as you normally would in SOLIDCast. There are then two ways to start FLOWCast:

1. If you only want to run a fill simulation with FLOWCast, without a subsequent SOLIDCast solidification simulation, you can select the Mesh icon on the SOLIDCast Project Tree, click on the **Mesh** menu item, and from the drop-down menu you can select **Start FLOWCast**. This will appear as follows:



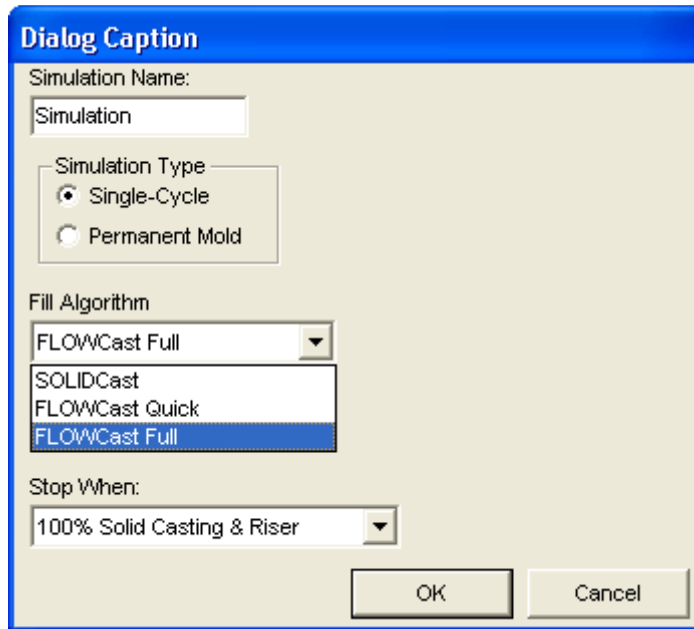
This will bring up FLOWCast in an “Interactive Mode”, when you can use the FLOWCast menu to start a simulation, select between a Quick and Full simulation, change the view angle, set the percent increments for saving data, and make AVI movie files after the simulation is done.

2. If you want to run a FLOWCast fill simulation and then automatically run a SOLIDCast solidification simulation afterward (using the results of the FLOWCast fill simulation as the starting point for the SOLIDCast run), then you would select the **Mesh** menu entry and select **Start Simulation** as shown below:





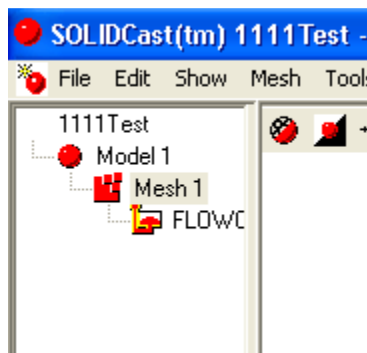
You will then see the typical “Simulation Start” window as shown below, but with a difference: With FLOWCast installed you can select between the traditional SOLIDCast mold filling simulation, the FLOWCast Quick simulation, or the FLOWCast Full simulation, as shown below:



If you select one of the FLOWCast fill algorithms, FLOWCast will run in “Auto-Run” Mode. This means that FLOWCast will start automatically, run to completion, and then SOLIDCast will run a simulation of solidification, using the results of the FLOWCast fill simulation.

**Q: If I run a FLOWCast simulation in Auto-Run Mode, how can I load it to see the fill simulation results?**

A: If a FLOWCast simulation has been run on a mesh (in either Interactive Mode or Auto-Run Mode) you will see a FLOWCast icon on the Project Tree that looks like this:



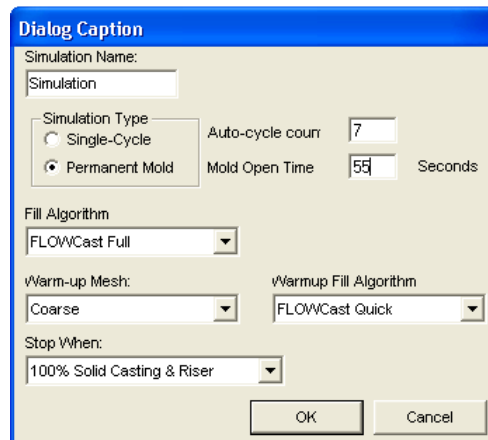
If you select **Mesh** and **Start FLOWCast** from the menu, FLOWCast will load the previous simulation run. You can then view this run or create AVI movies from it.

**Q: If I have run a FLOWCast simulation in Interactive Mode, how can I run a SOLIDCast simulation of solidification using the FLOWCast results?**

A: In this case, select **Mesh** and **Start Simulation**, and then select **SOLIDCast** as the Fill Algorithm. SOLIDCast will first check to see if the results of a FLOWCast simulation are available; if so, it will use these results as the starting point of a solidification simulation.

**Q: Will FLOWCast work with permanent mold (gravity die-casting) simulations?**

A: Yes! FLOWCast can be run in Auto-Run Mode for each cycle of a permanent mold simulation. If you use the option where initial cycles are run with a coarse mesh to warm up the mold and the final cycle is run with a refined mesh, you now have another option. You can choose to select one algorithm to use for the warm-up cycles and a different algorithm for the final cycle. This appears as follows:

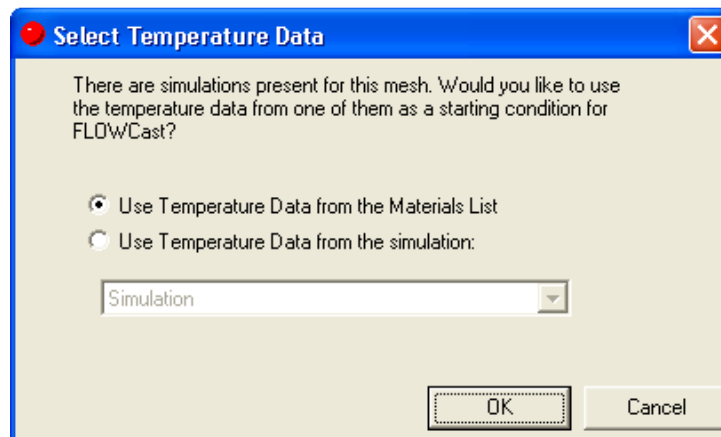


**Q: Can FLOWCast handle tilt-pour permanent mold simulation?**

A: Yes! Just specify the tilt pour using the **File...Options** menu selection in SOLIDCast while building the model. (Note: ONLY the Full FLOWCast algorithm may be used for a Tilt-Pour simulation. The Quick FLOWCast algorithm does NOT simulate tilt pour.)

**Q: Can I run FLOWCast using mold temperatures that have already been calculated in a SOLIDCast simulation?**

A: Yes! If you start FLOWCast in Interactive Mode (by selecting **Mesh... Start FLOWCast** from the SOLIDCast menus), and if a simulation has already been run on that mesh (which you can tell if there is a Simulation icon on the Project Tree) then the following selection box will appear:



This means that, if you start a FLOWCast simulation running, you can select to use either the initial temperature values of the materials or the temperature distribution as a result of the SOLIDCast simulation. So, for example, if you have completed a permanent mold simulation and want to run FLOWCast using the calculated temperature distribution in the mold, you can select **Use Temperature Data from the simulation** and then start FLOWCast running with the mold temperature data.

Note: If a FLOWCast simulation has already been run on a mesh, that simulation will be loaded automatically when you start FLOWCast in Interactive Mode. You can select **Restart** in FLOWCast to start a new flow simulation running.

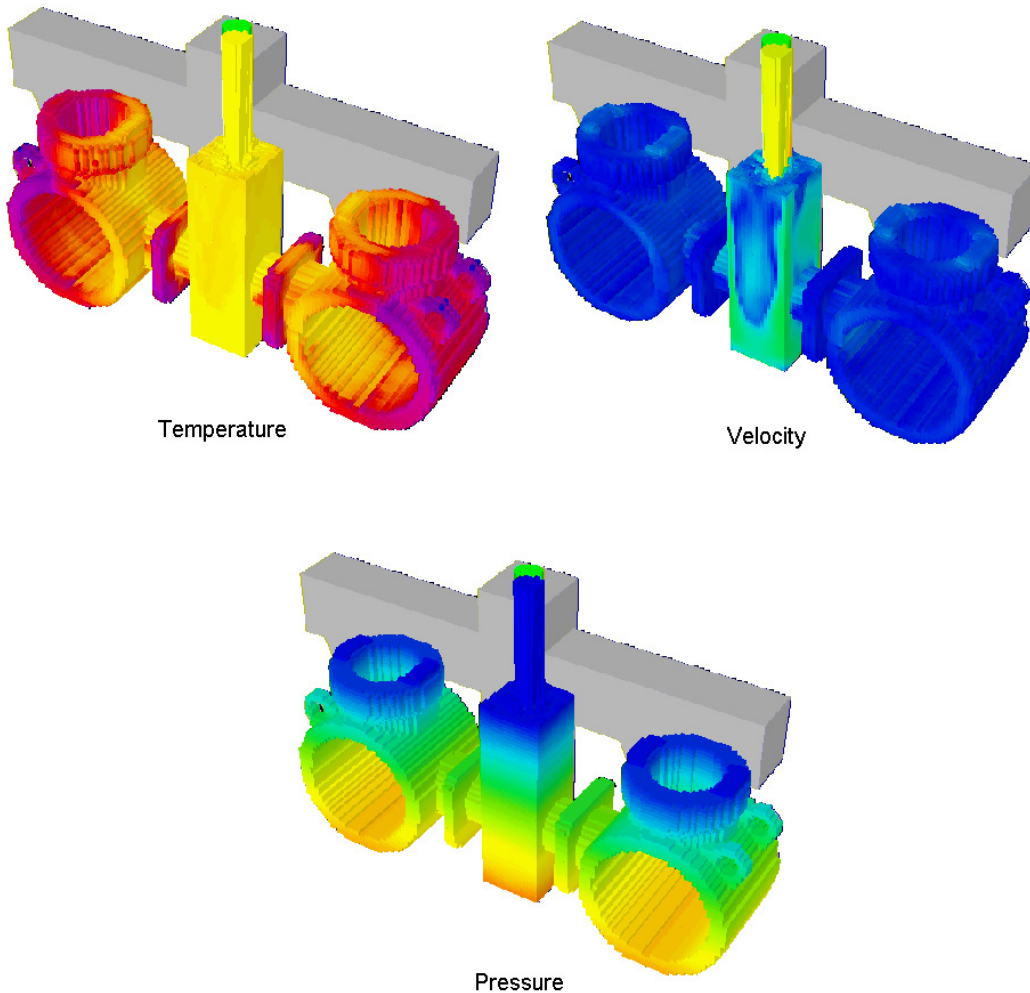
**Q: How is FLOWCast data stored?**

A: Before running a FLOWCast simulation, the user specifies a % increment for saving time-step data. This means that after the simulation is complete, you can view the filling at each of these percent increments but not in between. When AVI (video) files are created, they are created using these saved time steps. Saving at each 5% creates relatively large time steps; each 1% creates a relatively “smooth” video file. The size of the output data files depends partly on the number of time steps, and partly on the number of nodes. Note that you can set a separate increment for temperature data only. This allows you to save more temperature data, which is used most often, than velocity and pressure data. This helps to minimize file sizes.

FLOWCast creates large data sets. It is recommended that, after creating the output views or AVI files that you want from the simulation, you archive the project folder to a CD or other off-line storage medium, or delete the FLOWCast information from the project tree.

**Q: What information can FLOWCast display?**

A: From a Full FLOWCast simulation, the system can display (and make AVI files of) temperature, velocity and pressure during the fill. A Quick FLOWCast simulation can show temperature and velocity. These types of displays are as shown below (in a partially-filled investment casting model):



You can also add flow pathlines and metal velocity vectors to the 3D plots and animations.

**Q: What kind of computer should I have for running FLOWCast?**

A: FLOWCast is very computationally intensive and creates a lot of data. Therefore, we recommend the fastest and largest computer that your budget will allow. At this time, this would mean a Pentium 4 machine running at 3.6 or more GHz, with 3 or 4 GB RAM, and at least a 100 GB hard drive.

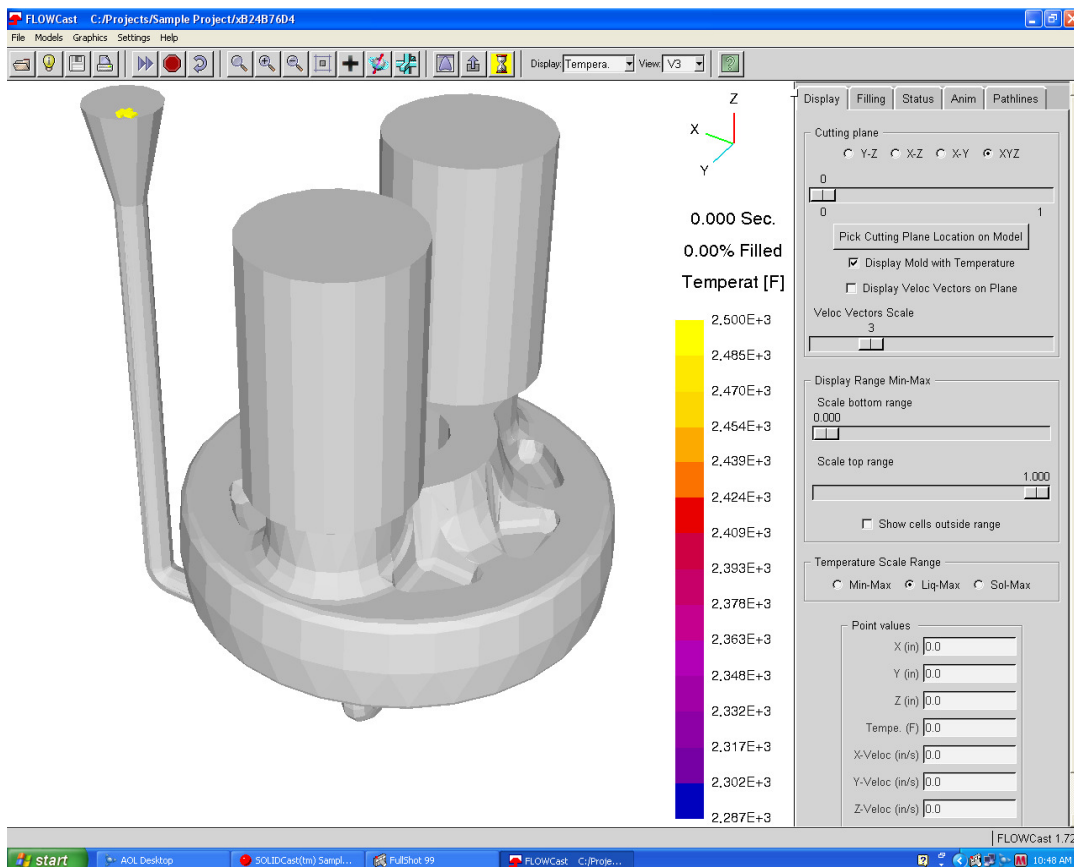
**Q: Can I run more than one FLOWCast simulation simultaneously?**

A: No. Only one instance of FLOWCast may be running on a single computer at one time. Since FLOWCast uses such a large amount of memory, running multiple simulations, even on multi-processor machines is not practical.

**Using FLOWCast’s Interactive Mode**

This section will take you on a quick tour of FLOWCast’s most useful commands. In this Quick Start Guide, you should learn enough to be able to start and stop FLOWCast, change simulation and display modes, and generate video files of mold filling.

When you start FLOWCast by selecting **Mesh... Start FLOWCast** in the SOLIDCast menu, you are entering FLOWCast’s Interactive Mode. When FLOWCast first appears, you should see a screen similar to this:

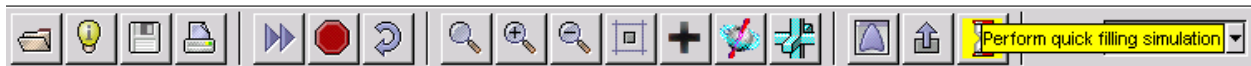


Note the Toolbar at the top of the screen, which looks like this:



The buttons on the Toolbar are used for controlling the simulation.

**Q: How do I select between a Full Simulation and a Quick Simulation?**



A: The hourglass button is used to toggle between Quick and Full. If this button is depressed, you are in Quick Mode. If it is “popped out”, you are in Full Mode. When FLOWCast is first started in Interactive Mode, it is in Full Mode.

**Q: How do I start a FLOWCast simulation?**



A: Just click the double arrow button to start the simulation running.

**Q: How do I interrupt (stop) a FLOWCast simulation that is running?**

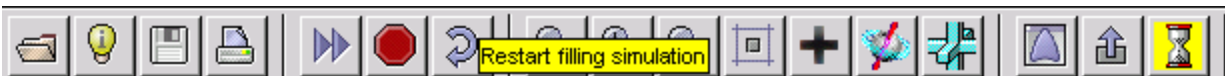


A: Click the “Stop Sign” button. You may need to wait a few seconds for the simulation to actually stop, as FLOWCast needs a little time to organize its data for the stop.

**Q: How do I restart a FLOWCast simulation that has been interrupted?**



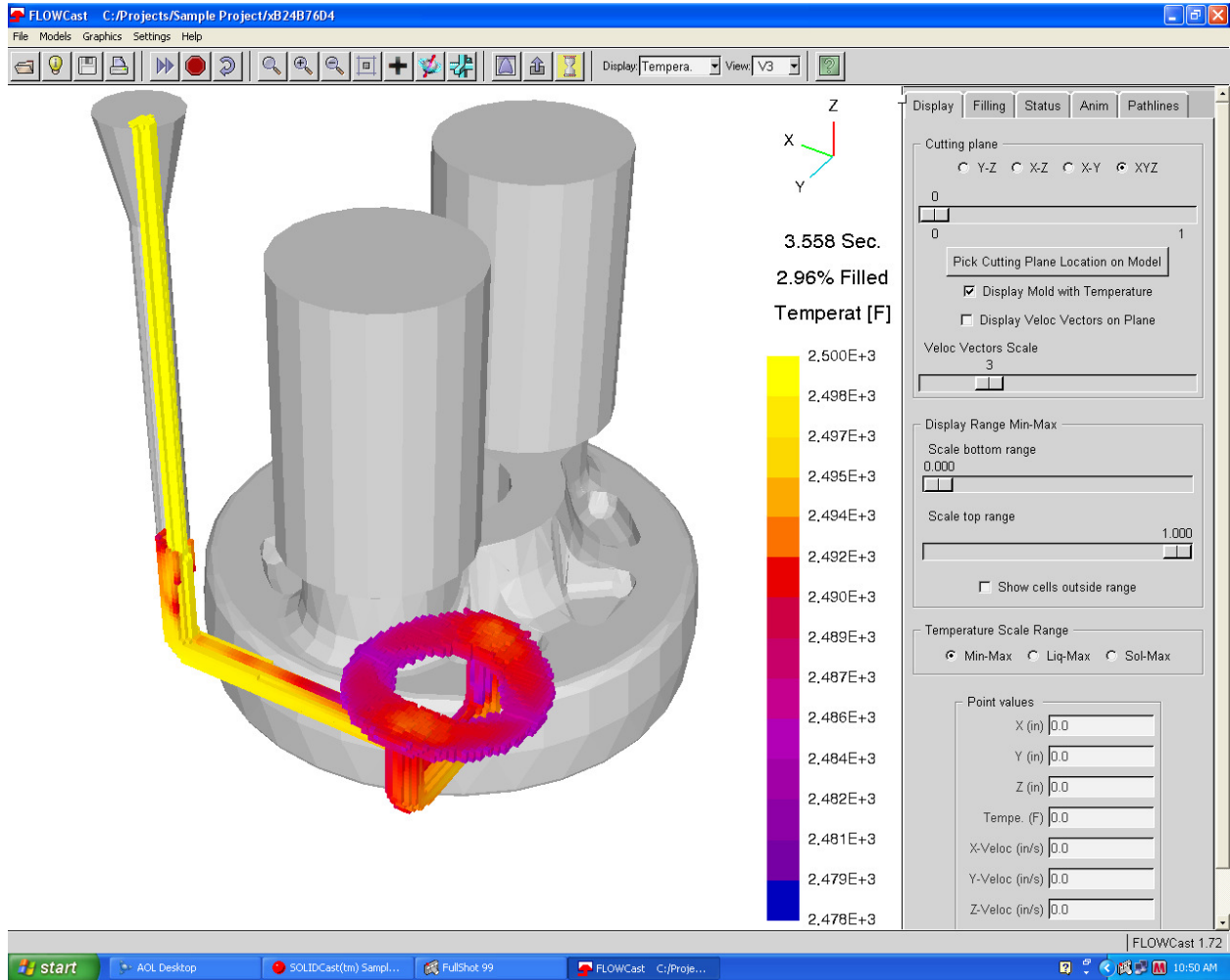
A: If you want to start from the same point, just click the double arrow again.



However, if you want to restart the simulation from the beginning, first click the Restart button as shown above. This will reset FLOWCast to the beginning. You can then make changes in parameters and press the Start button (double arrow) to start again.

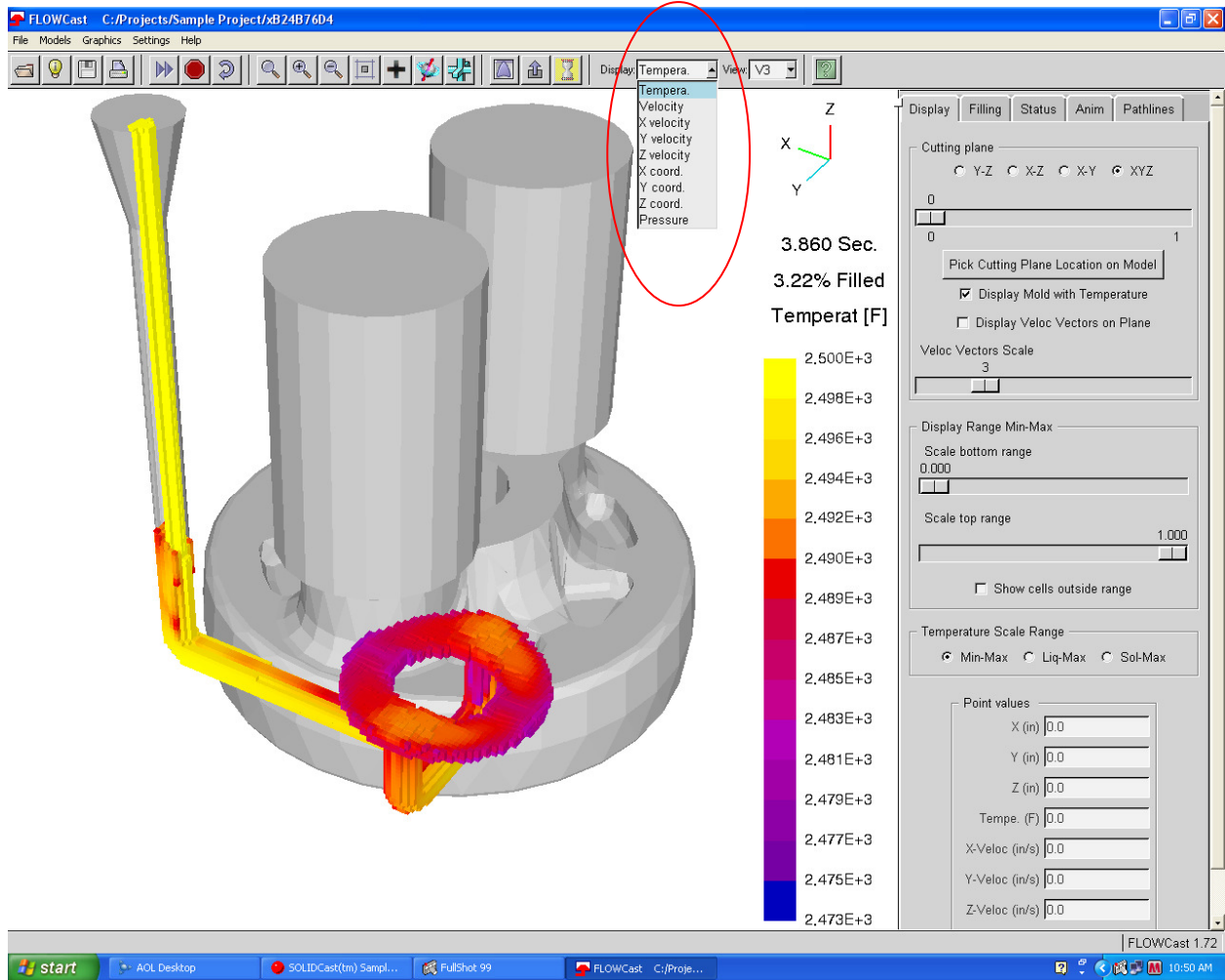
**Q: What will I see when FLOWCast starts running?**

You will see the molten metal beginning to run into the mold cavity. This should look something like the following:



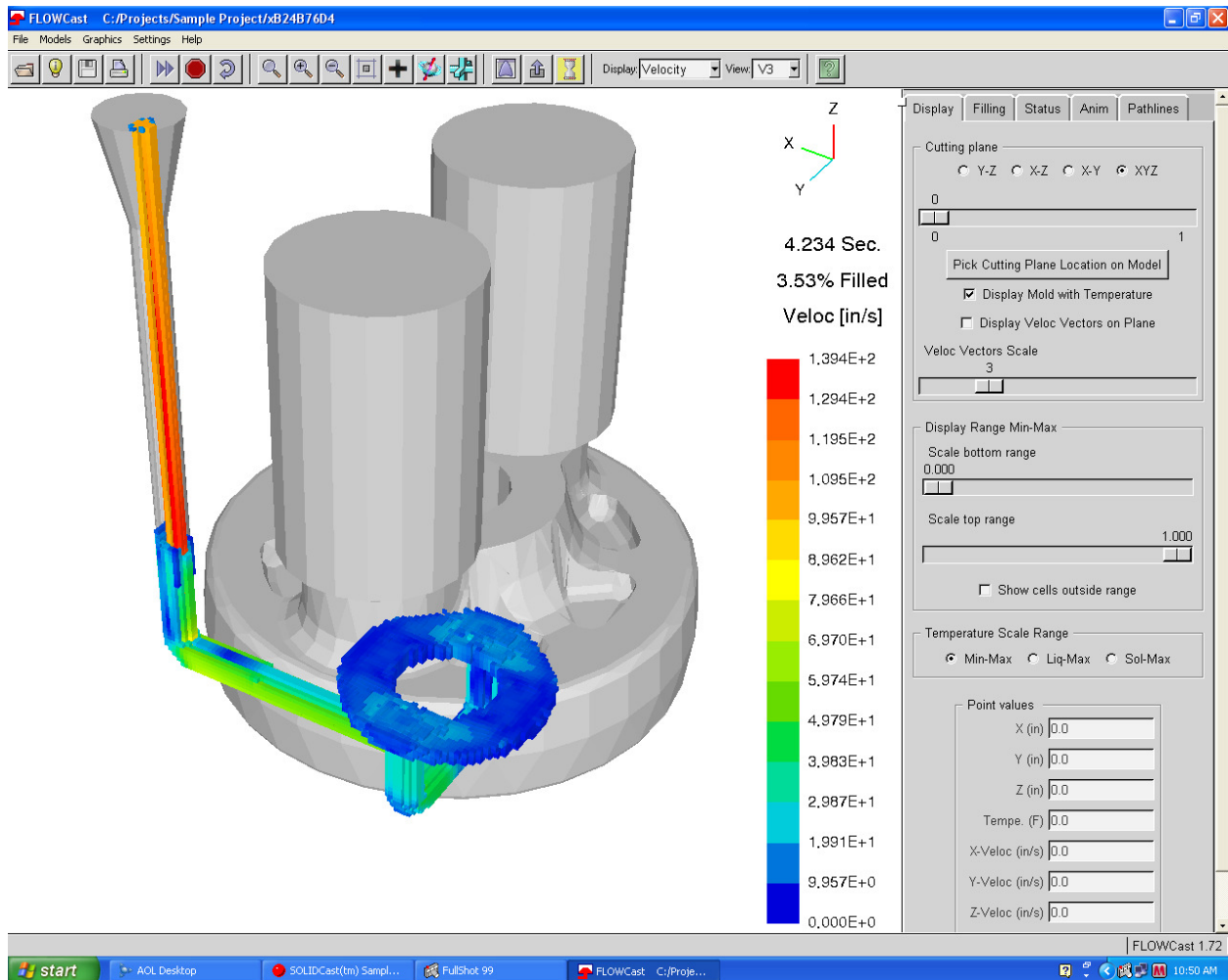
**Q: How do I switch between viewing Temperature, Pressure and Velocity?**

A: Clicking on Display (as shown below) allows you to select between Temperature, Pressure and Velocity displays (as well as some other display choices):



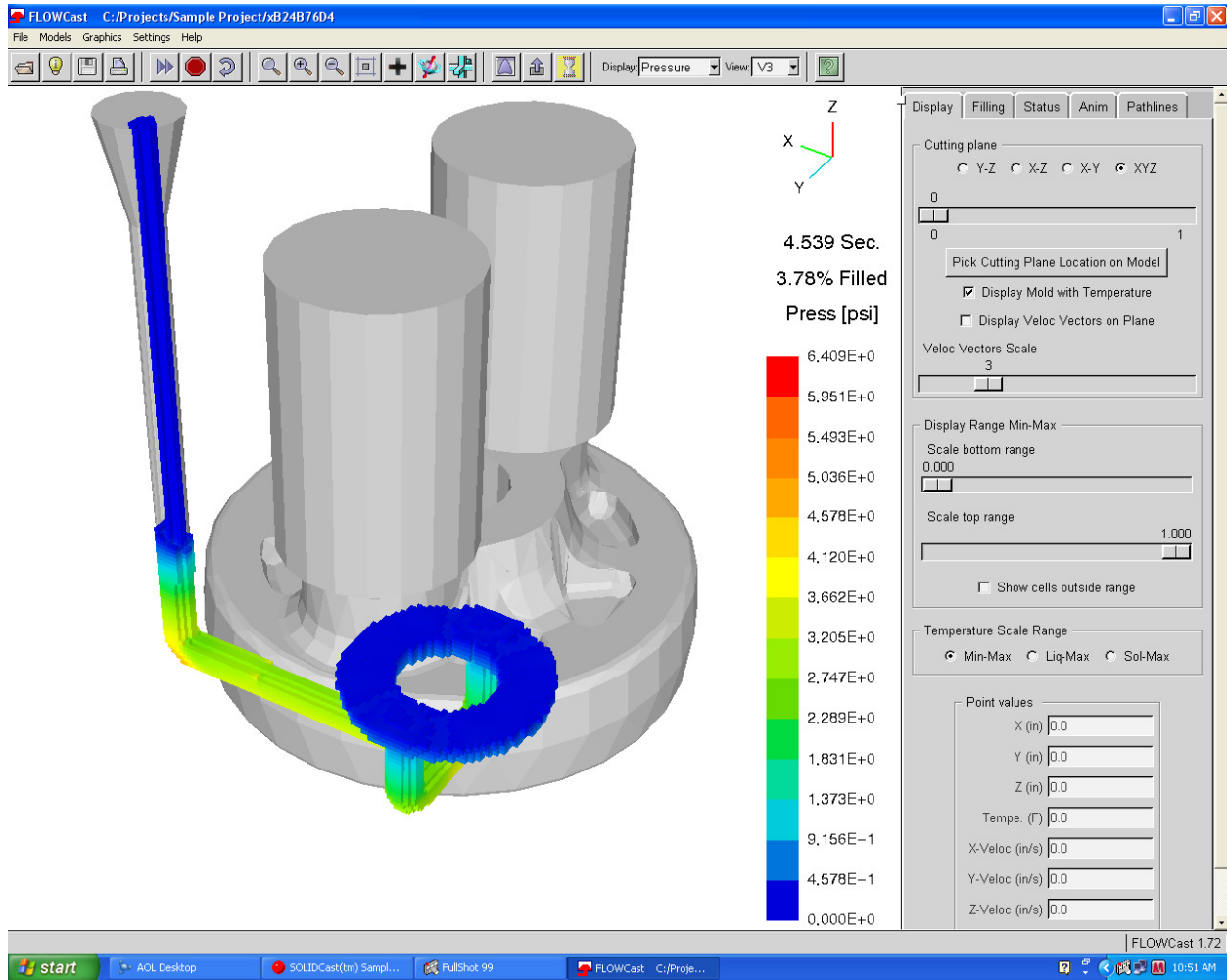


Let's see what the display looks like if we click on Velocity...



Here, the brighter colors represent areas where the velocity of the liquid metal is higher. We can see that the metal pouring down the sprue is obviously the highest velocity. The liquid in the runner between the sprue and gate, and the metal “falling” down the gate into the casting are all areas of higher velocity.

Selecting the Pressure display shows the following:

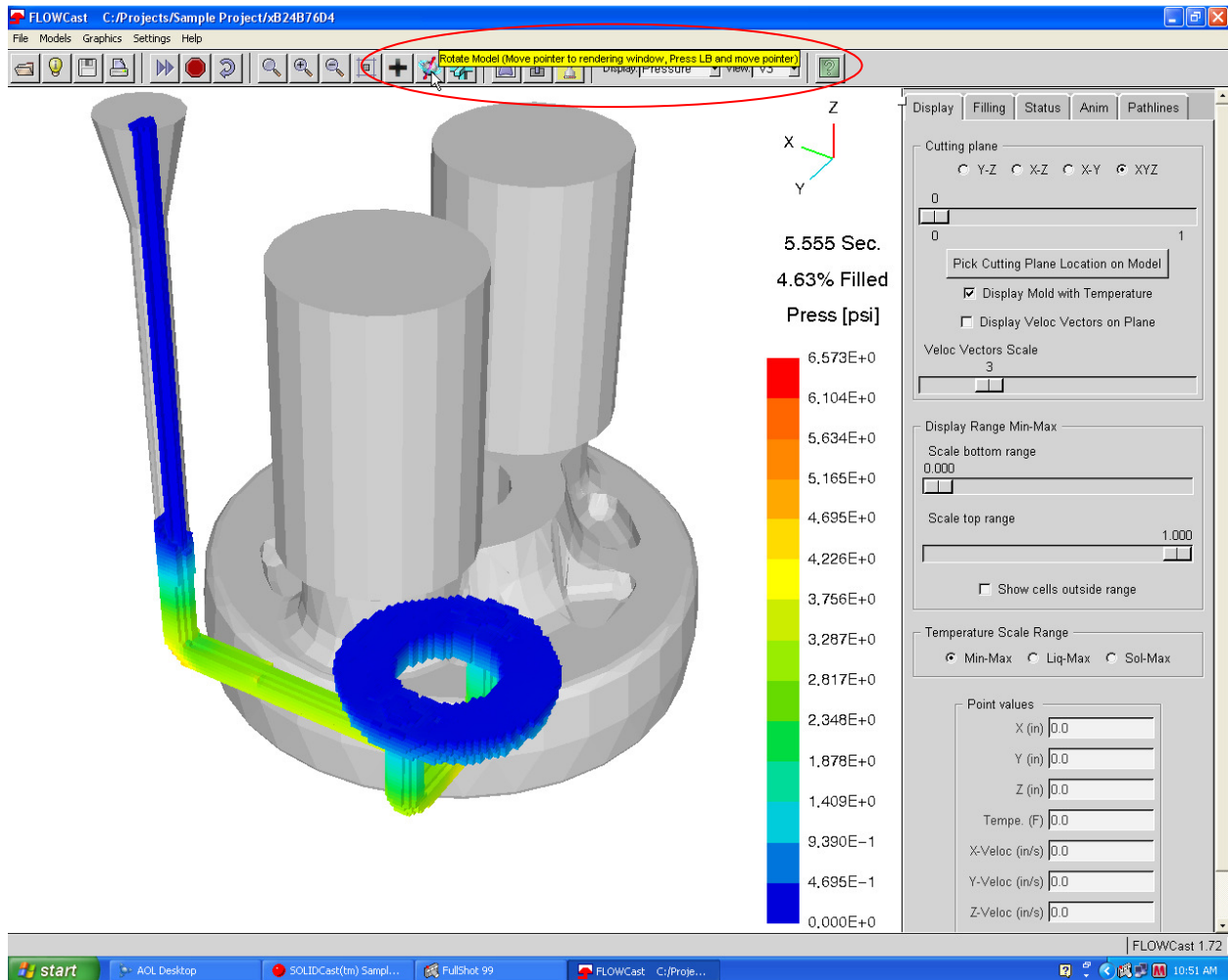


Here we can see relatively high pressure in the runner. Anywhere that the liquid is free-flowing and not under metallostatic pressure is a low-pressure area.

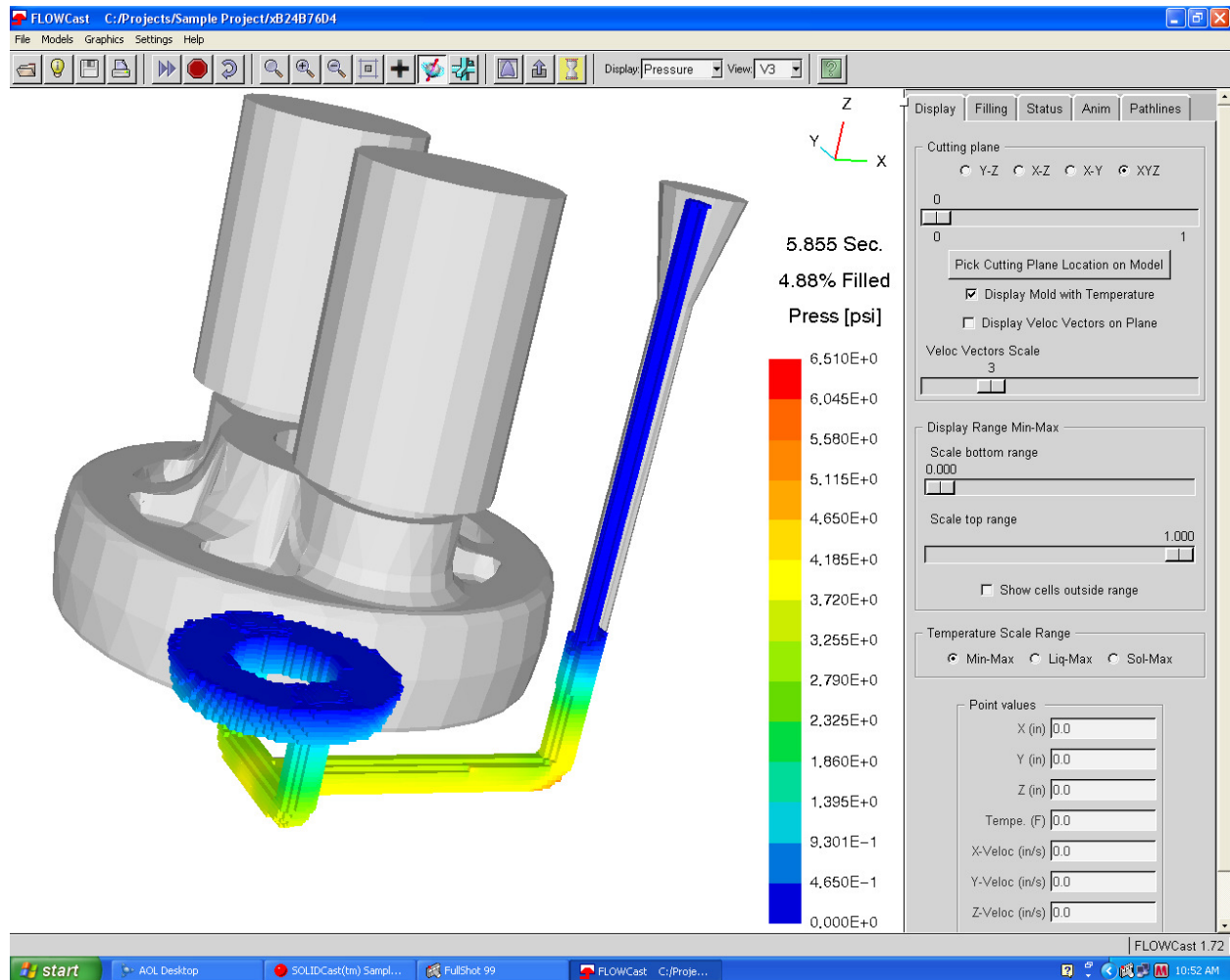
Note: You can change the display at any time, even while the simulation is running.

**Q: How do I rotate the FLOWCast model to see it from different angles?**

A: FLOWCast allows you to rotate the model in real time, as you can do in SOLIDCast. To rotate the model, click on the Rotate Model button as shown below, and then press and hold the left mouse button as you drag the cursor across the screen until the model is in the desired orientation.



This screen shows the model view after a rotation has taken place:



You can change the viewing angle at any time, even while the simulation is running.

There are a number of preset viewing angles that can be accessed from the View Control. Details are discussed in Unit 10 of this workbook.

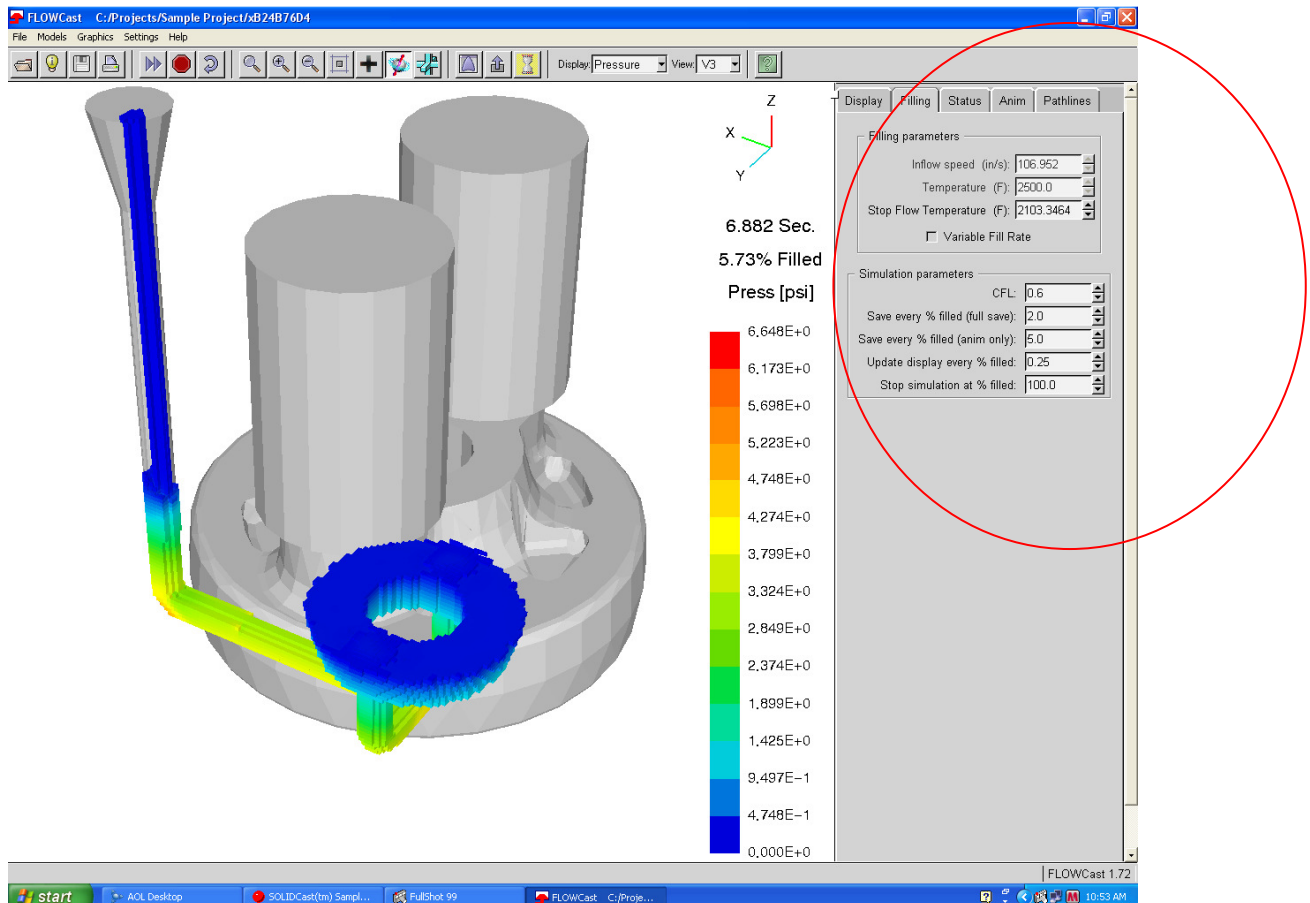
**Q: How do I set the percent time steps for saving flow simulation results?**

A: This should be done **BEFORE STARTING THE FLOW SIMULATION**. First click on the tab labeled **Filling**. Then, enter a percent in the field labeled **Save every % filled**. For example, if you want to save the filling data every 5%, then enter 5 in this field as shown below. This means that, after the fill simulation is complete, you can display the data at each of these increments. If you create an AVI file from the FLOWCast simulation, the individual frames of the AVI video will be at these increments.

Since the most interesting flow data usually occurs near the beginning of filling, you can set the save increment to, say, 1% initially. Then, when filling has evened out, stop the simulation, change the increment to a larger value, say 3% or 5%, and then continue the simulation. This will save on disk space.

You can also set a separate save increment for temperature data only, by adjusting the field labeled

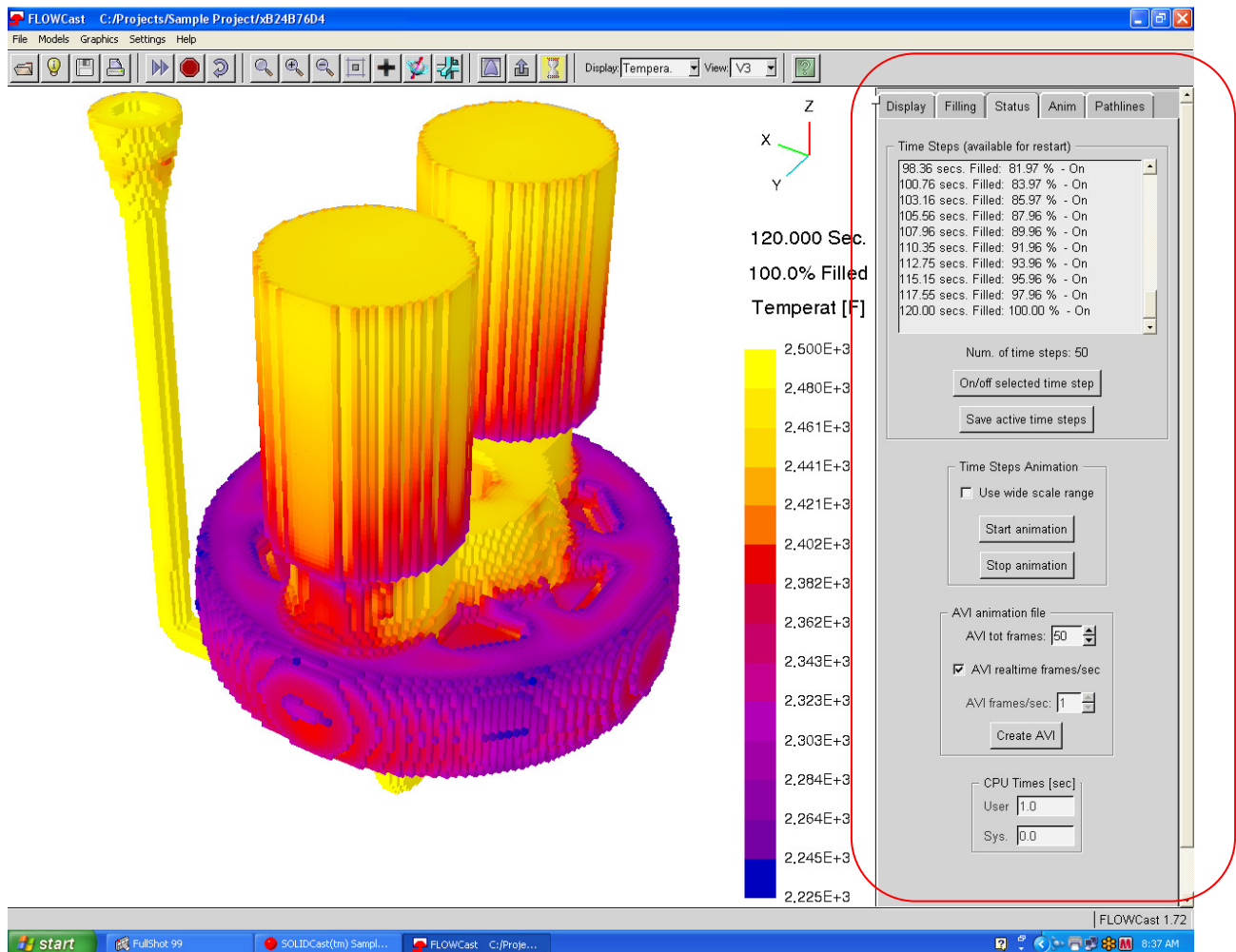
**Save every % filled (anim only)**. Since temperature data is typically used most often, it is a common practice to save temperature data at 0.5% or 1% increments, and other data at 5% increments. This allows for very smooth animations to be made from temperature data, without creating huge data files for all the rest of the data.



**Q: How do I create an AVI (video) file from the FLOWCast simulation?**

A: This is normally done when the FLOWCast simulation is finished. You can, however, interrupt a FLOWCast simulation when it is partially complete and make an AVI file of the simulation up to that point.

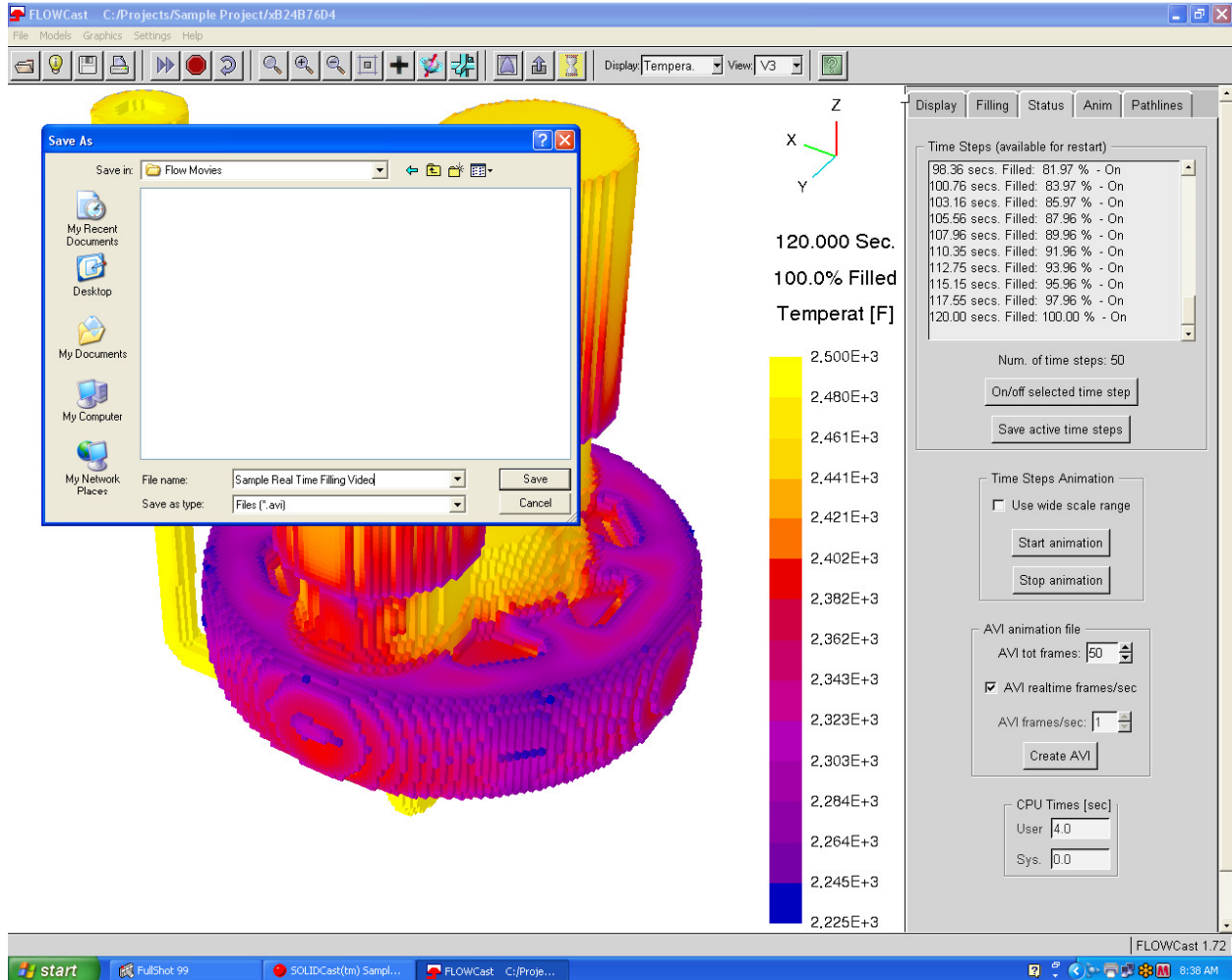
To create an AVI file, first click on the tab labeled **Status**. The individual time step data will be shown in the window. (Note: If you select one of these time steps by clicking it with the mouse, the simulation at that point will be displayed.) Now, click the button at the bottom of the screen labeled **Create AVI**.



You have the choice of creating a 'realtime' AVI, which runs at the actual pouring speed, or at a fixed number of frames per second.

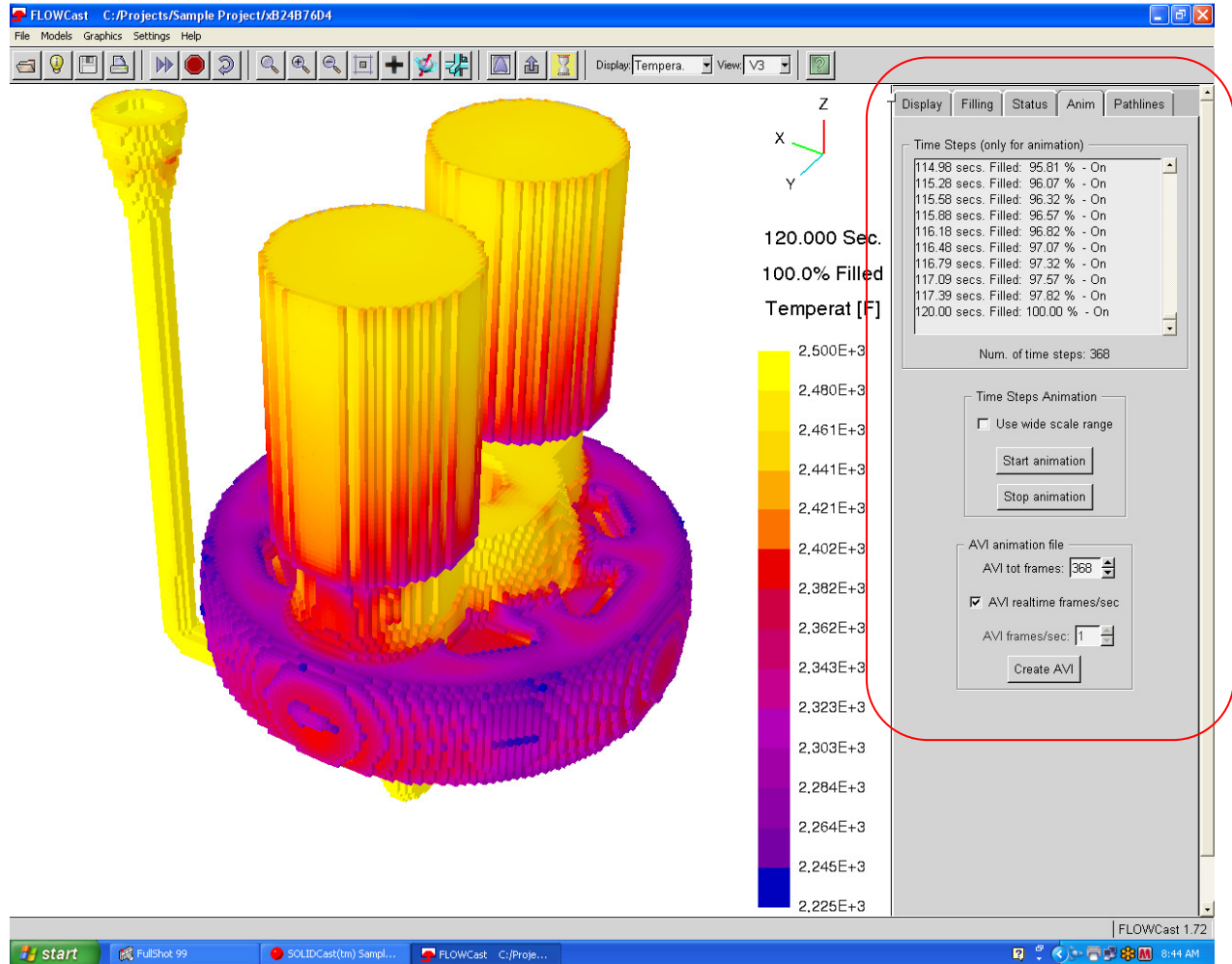


The system will ask you to enter a name for the AVI file, and will also allow you to specify where the AVI file is to be placed. Once you have entered the file name, click **Save**.



The AVI file will be created at this point.

Note that the AVI file is created using the display type and viewing angle that are currently selected. You may typically want to create three AVI files, one showing temperature, one showing pressure and the other velocity. All you have to do to create these different files is to select the appropriate Display type next to the Tool Bar before creating the AVI file.



Temperature information can also be animated using the Anim Tab. Typically, you would save temperature data at a smaller increment, and so more data is available to make movies with. By saving, say, 200 time steps, a very smooth animation can be created. An example of what the Anim Tab looks like is shown above. The Create AVI function works in exactly the same way on the Anim Tab as it does on the Status Tab, but it only considers temperature information. If your display is set to any other data, when you click on Create AVI the system will switch over to temperature before creating the movie.

### Q: How do I get back to SOLIDCast?

Just exit from FLOWCast, either by selecting **File... Exit** or closing the FLOWCast window with the X button in the upper right-hand corner. When FLOWCast closes, SOLIDCast is in a minimized state on the Windows Task Bar. Just click the SOLIDCast button on the Task Bar to restore the SOLIDCast window.



## UNIT 2: System Installation and System Considerations

---

### Hardware

Minimum Requirements (SOLIDCast without OPTICast or FLOWCast, or for small mesh node counts)

Intel Core 2 Duo E6300 or better, or AMD Athlon II or better

2 GB RAM

500GB Hard Drive

Midrange (\$200-\$300) DirectX 9 or 10 compatible video card

Windows XP Professional or Windows 7 Professional, Enterprise, or Ultimate

Recommended Requirements (SOLIDCast with FLOWCast and/or OPTICast, or very large node counts):

Intel Core i7 900 series or better, or AMD Phenom II

8 GB RAM or larger

1 TB Hard Drive

Midrange (\$200-\$300) DirectX 9 or 10 compatible video card

Windows 7 Professional, Enterprise, or Ultimate, 64-bit edition

A note about video cards: SOLIDCast makes use of standard Microsoft video driver technology to display its images. Any current midrange video card available which advertises compatibility with Direct X 10 will work for the current version and upcoming versions of SOLIDCast. An existing video card which is compatible with DirectX 9 will work with the current version of SOLIDCast.

### System Installation

New systems will come on a CD, DVD or USB Drive with several folders. These folders are typically labeled Software, Software Demos and Training Materials.

Note: Version 8 of SOLIDCast REQUIRES Windows XP, Windows Vista, or Windows 7. If you are running Windows XP, an update to Service Pack 3 (through Windows Update) is required for support of all of SOLIDCast's features.

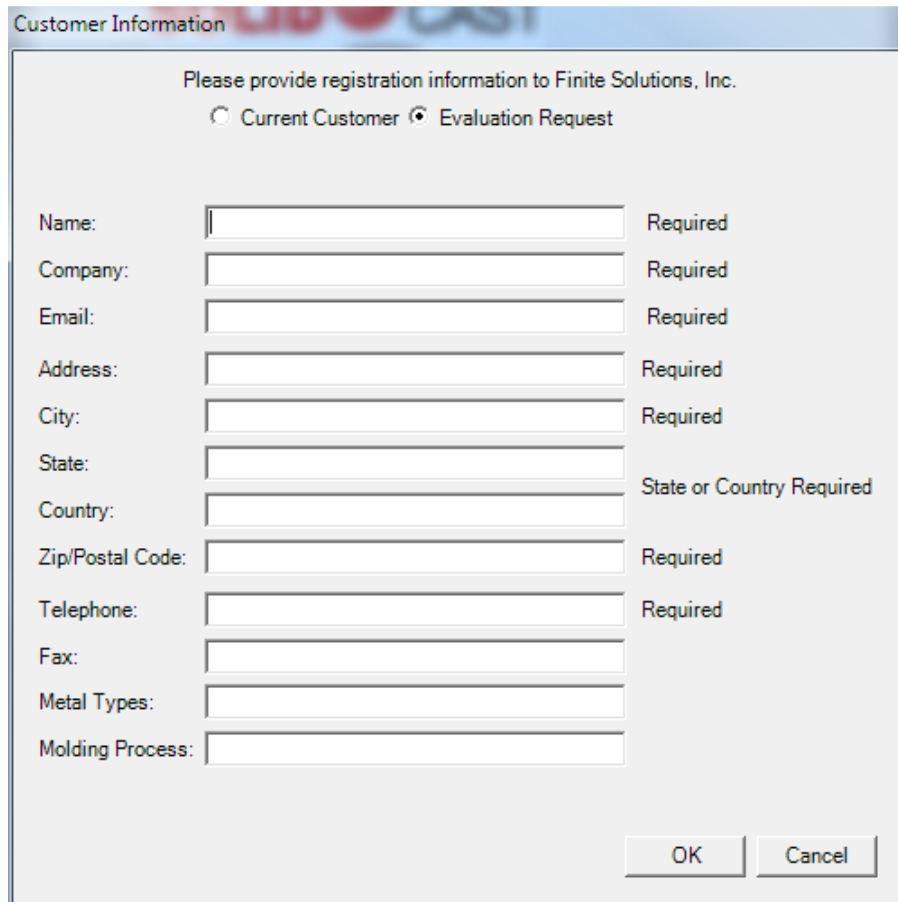
To INSTALL the package:

1 - Insert the CD, DVD or USB Drive.

2 - Go to the Software Folder and double-click the file name SOLIDCastInstall8-1-1.exe to run the installation program.

3 - Follow on-screen instructions. FLOWCast and OPTICast are included in the package installation, and will activate with SOLIDCast if your organization has licensed them.

4 - To activate the system for normal operations, a license file is required, which will be supplied by Finite Solutions Inc. Start SOLIDCast, then click on Help...About SOLIDCast. This will bring up a window with the license agreement, and a "Licensing..." button. Click the "Licensing..." button.



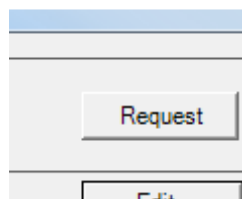
The image shows a dialog box titled "Customer Information" with a blue header. Below the title bar, it says "Please provide registration information to Finite Solutions, Inc." and has two radio buttons: "Current Customer" (unselected) and "Evaluation Request" (selected). Below this are several text input fields with labels and "Required" status indicators:

- Name: [ ] Required
- Company: [ ] Required
- Email: [ ] Required
- Address: [ ] Required
- City: [ ] Required
- State: [ ] State or Country Required
- Country: [ ] State or Country Required
- Zip/Postal Code: [ ] Required
- Telephone: [ ] Required
- Fax: [ ]
- Metal Types: [ ]
- Molding Process: [ ]

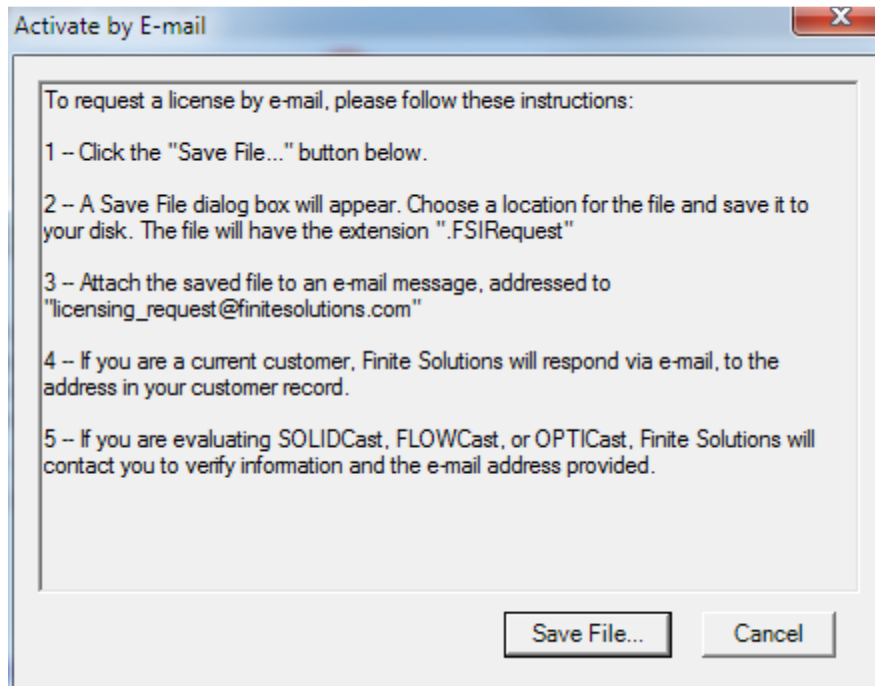
At the bottom right of the dialog box are "OK" and "Cancel" buttons.

You will be prompted to enter registration information. If you are evaluating SOLIDCast, click the "Evaluation Request" button and enter all required fields. If you are a current customer, click the "Current Customer" button and fill out Name, Company, and E-mail.

5a - If your computer is attached to the Internet, click on the button marked "Request". Clicking "Request" will send a license request to Finite Solutions; nothing more is needed from your end.



5b - If your computer is not on the Internet you will see a "Save File" dialog upon clicking "Request" E-mail the file according to the instructions on the prompt.



6 - Close SOLIDCast.

7 - You will receive a license file from Finite Solutions, emailed to the contact name in your customer record, or to the email provided to us in the request. Copy that file to your hard drive and double-click it. This will load the file onto each your machine. One license file may activate one or more machines, but each machine must send in a request in order to be active in the file. Again, be sure that SOLIDCast is NOT running when you activate the license by double-clicking the license file.

8 - SOLIDCast is now activated for your use. Note that a single activation request will cover all modules, including SOLIDCast, FLOWCast and OPTICast.

The media also contains other support files, including the training workbooks, version log file and others. If you have any questions about these files, read the more detailed information in the workbook files for each program, or contact Finite Solutions Inc.

David Schmidt  
Finite Solutions Inc  
Tel 262 644 0785  
Fax 262 364 2537  
Email [Dave@finitesolutions.com](mailto:Dave@finitesolutions.com) or [Finitell@aol.com](mailto:Finitell@aol.com)

## Updates

SOLIDCast is under continuous development, as we add new features and functions to the system. These updates can be downloaded from our web site. To obtain an update, navigate to the following site using your Web browser:

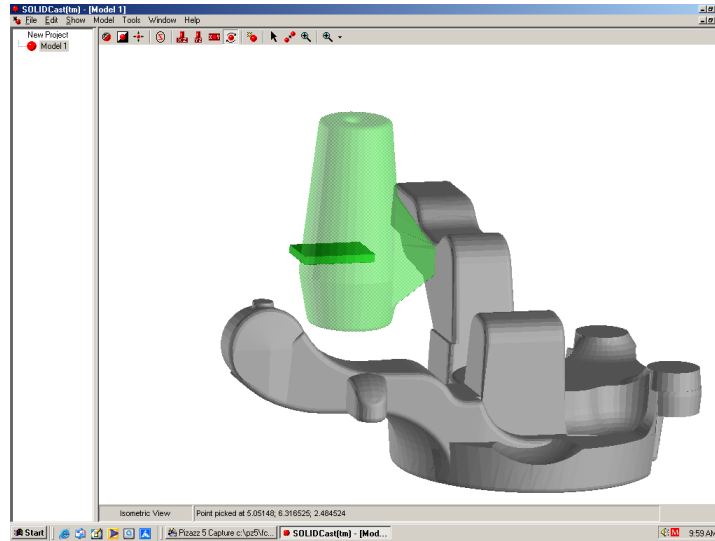
<http://www.finitesolutions.com>

You will see the FSI download page here. There are several useful items here, including the full install set, the latest update, and a copy of this training manual, in compressed format. All training manuals are saved as pdf files, and have been compressed using WinZip to reduce file size. Just click on a link to download the file to your computer.

To automatically be notified of system updates, join the SOLIDCast Users eGroup. Point your web browser to <http://groups.yahoo.com>, and search for SOLIDCast. Ask to join, and we'll put you on the list. We send out regular information, including training schedules, tips and techniques and software update notification. Please join today, as this is our major method of informing our users of program updates.

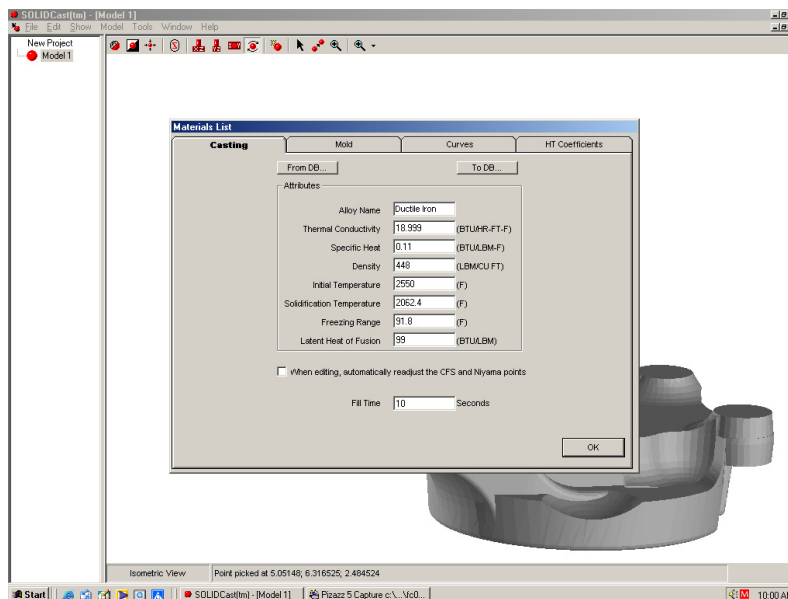
## UNIT 3: Setting Up a Fluid Flow Simulation Run

The first step in running a fluid flow simulation using FLOWCast is to load a casting model in SOLIDCast. An example casting is shown here:



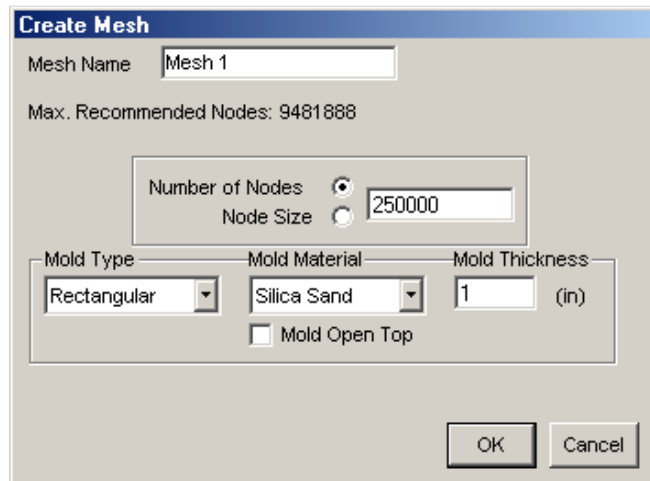
The first thing to note about the model is that you must have at least one piece of Fill Material present in the model. Fill Material tells the system where the hot metal will enter the model as the simulation proceeds. This model has a piece of Fill Material attached to the riser, showing a “hot” riser situation.

The second step is to go to **Model...Materials List...Casting** and enter a Fill Time, in seconds, as shown here:

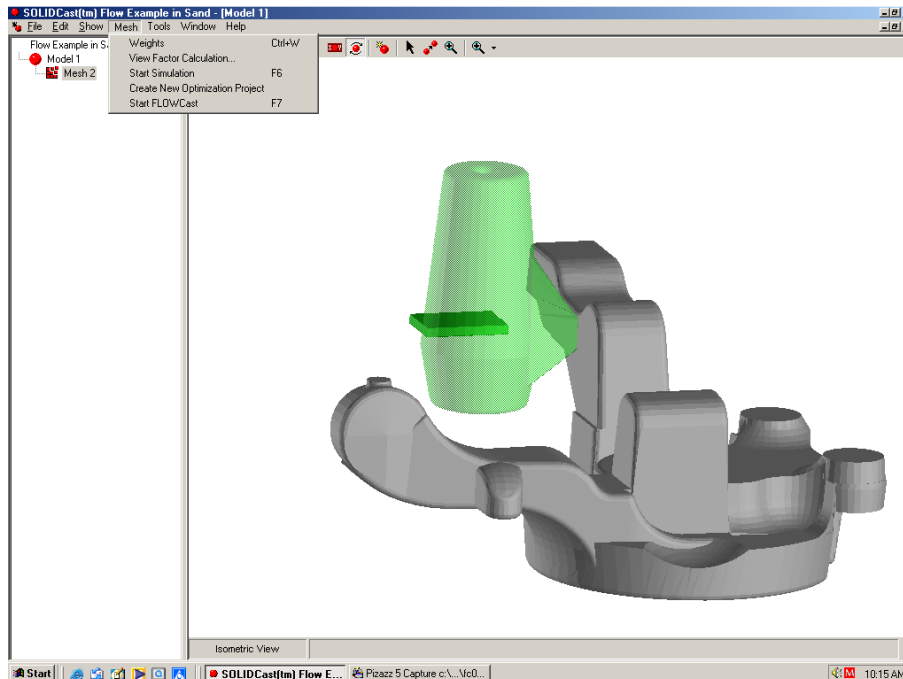


The Fill Time is used by the system to determine the initial filling velocity for the metal.

The third step is to click on **Model...Mesh** and create a suitable meshed model. The screen below shows a typical setup. This mesh will contain approximately 250,000 nodes. The mold shape will be rectangular, with a minimum of 1 inch of Silica Sand mold material around the casting, including on top of the riser.

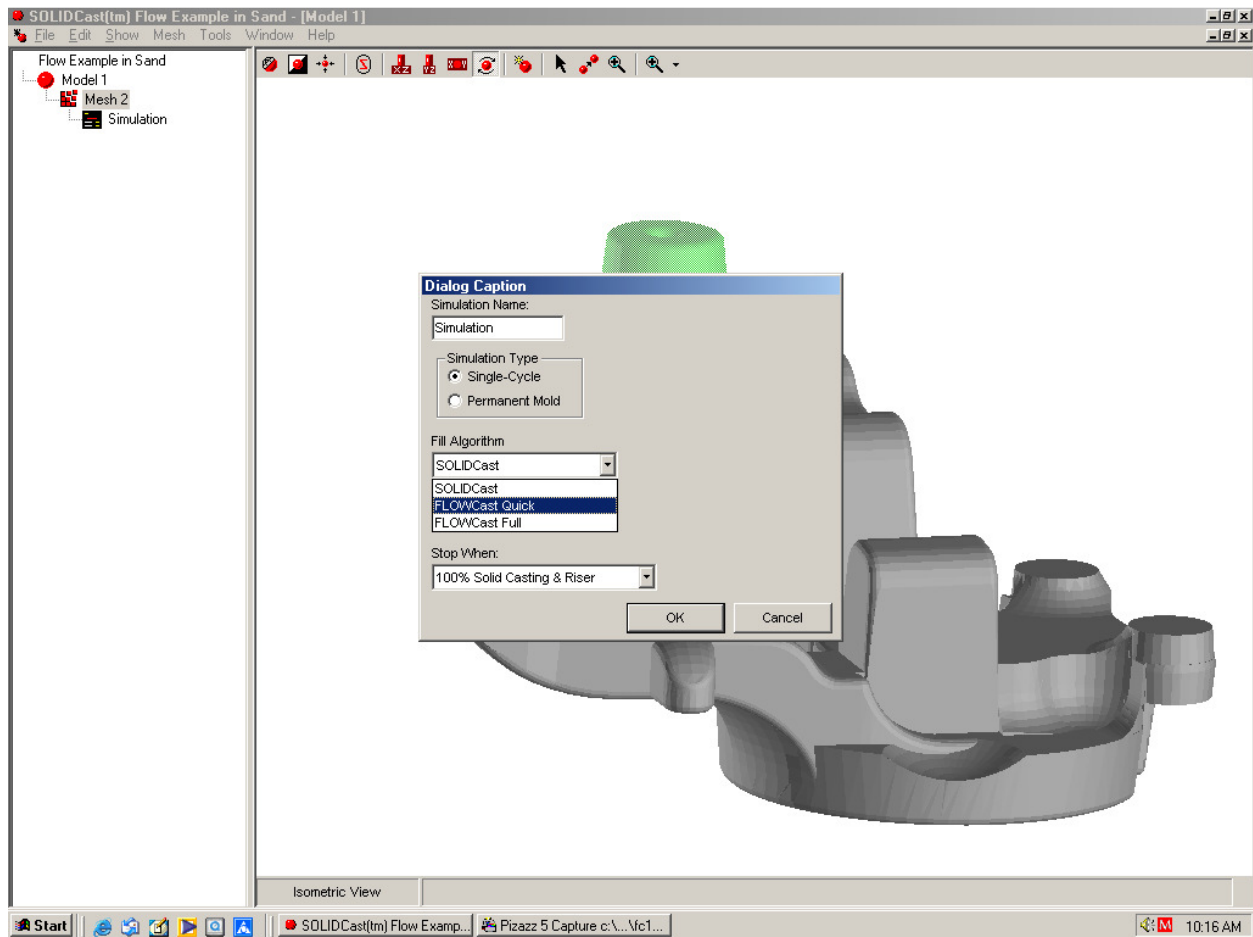


The final step is to start the fluid flow simulation itself, which can be done in one of two ways, depending on if you want a fluid flow simulation only, or want the results of the fluid flow simulation to be piped into a SOLIDCast solidification simulation. Both options are on the **Mesh** menu, as shown here:



If only a fluid flow simulation is required, click on **Start FLOWCast**. This will minimize the SOLIDCast window and load the FLOWCast program and data. From there you can start the simulation.

If you would like the results of the FLOWCast filling simulation to be the basis of a SOLIDCast solidification analysis, then click on **Start Simulation...** You will see a window similar to this:



With FLOWCast installed, you actually have a choice of 3 mold filling algorithms:

**SOLIDCast** – Simple, fast, order-of-fill algorithm supplied with SOLIDCast that gives temperature data.

**FLOWCast Quick** – Relatively fast, cascade method, which gives temperature and velocity data.

**FLOWCast Full** – Full CFD. Provides temperature, velocity and pressure data. Longest calculation time.

If you select either of the FLOWCast algorithms and click on **OK**, the SOLIDCast window will be minimized and the FLOWCast program and data will be loaded and automatically start. Once the filling simulation is complete, the data will automatically be loaded back into SOLIDCast and the solidification simulation will run.





## UNIT 4: The FLOWCast Toolbar

The main options for running FLOWCast are controlled from the FLOWCast Toolbar, as shown here:



This section will describe each of the functions available from the toolbar. You can find the name of each icon by moving the cursor over the top of the icon.

### Open Model File

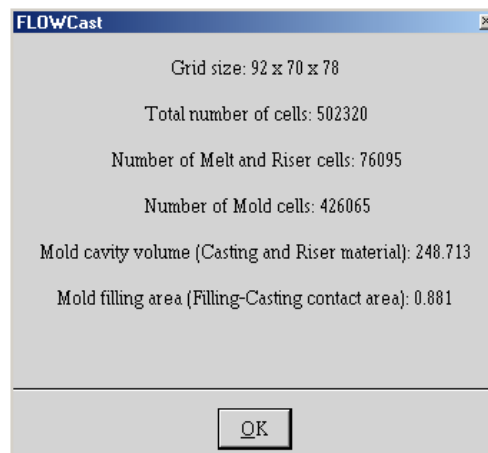


This option is used to load a new meshed model into FLOWCast. If you choose this option the system will display a navigation window, where you can find a different mesh file to load. The mesh file will be named **mymesh.mno**. Normally, this option will not be needed, if you have started FLOWCast from within SOLIDCast. The mesh will be loaded automatically, so you won't need to specifically select it.

### Show current grid parameters



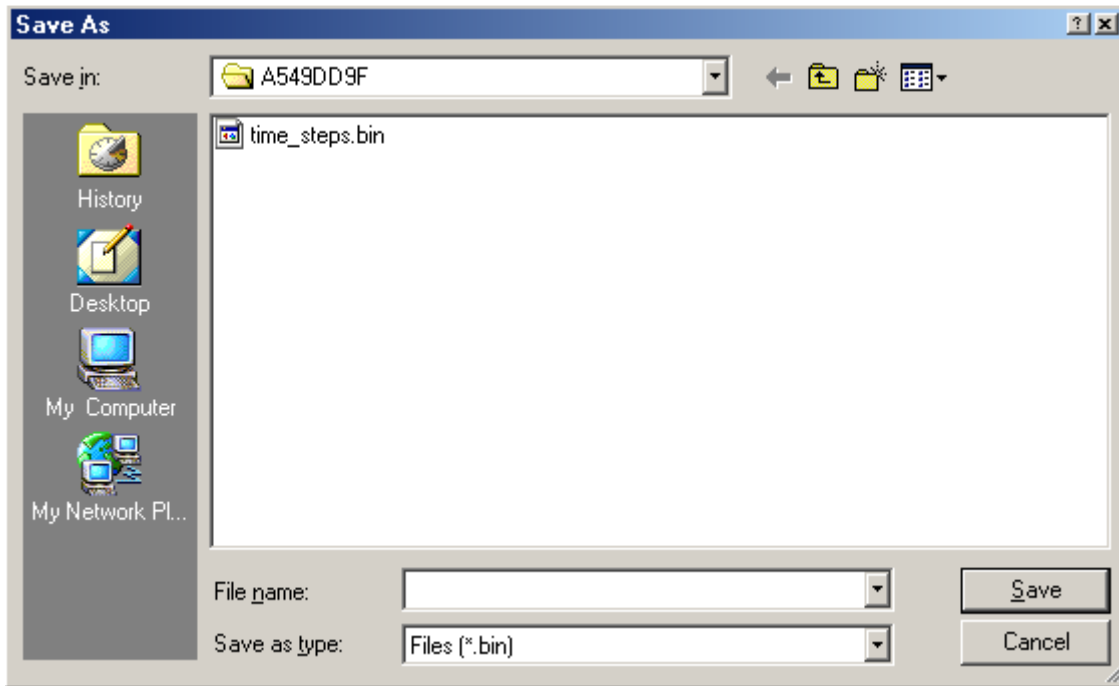
This option will pop up a window, listing the current mesh size, along with the number of casting nodes and riser nodes. An example is shown here:



**Save simulation**



When FLOWCast is running normally, the system will automatically save data at prescribed intervals, which, by default, are at every 5% full. You can, at any time, however, stop the simulation and save the data to a file. When you select this option, you will see a Save As... screen similar to this:



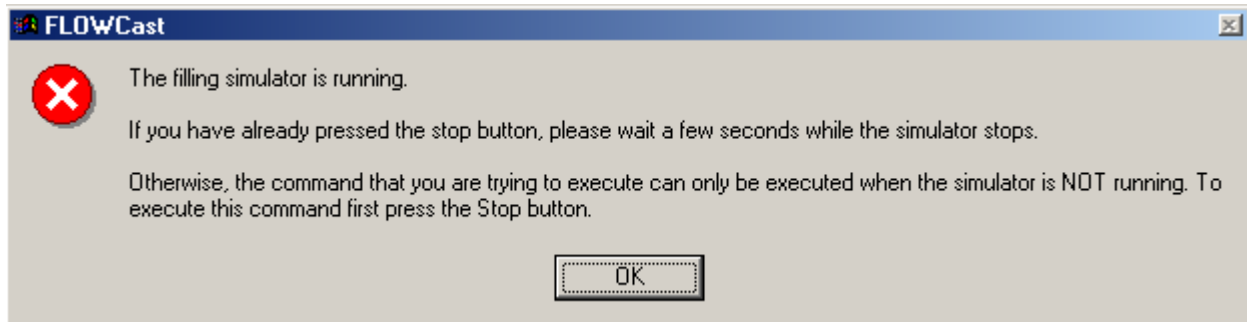
Simply type in a file name and click on Save. You can navigate to any folder you wish.

If no time step data has been created yet, you will see the following message:



This simply means that you are too early in the simulation process for any meaningful data to have been created.

If you try to save the data while the simulation is still running, you will see the following:

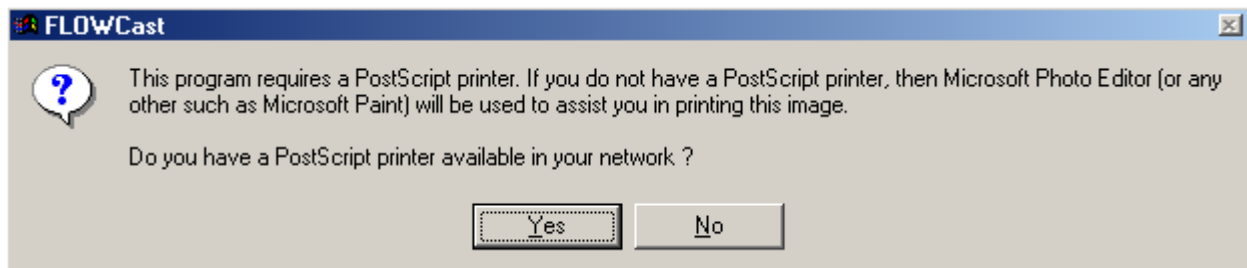


In order to save data manually, the flow simulation must be stopped. To stop the simulation, click on the stop sign icon on the toolbar. You will need to wait a few seconds for the system to organize its' data before you can save it out to a file.

**Print image**



FLOWCast can directly print a copy of the current screen, if you have a PostScript printer. When you click on this icon, you will see the following:



If you say yes, the page will be printed for you. If you do not have a PostScript printer, you can still use the PrtSc key to copy the screen to the clipboard. You can also purchase any number of low cost capture utility programs that will allow you to save one or more screens to files, print directly, etc.

**Start/Continue filling simulation**



Just click the double arrow button to start the simulation running. If you have previously stopped a simulation, this will start the simulation again at the point where you left off.

### Stop filling simulation

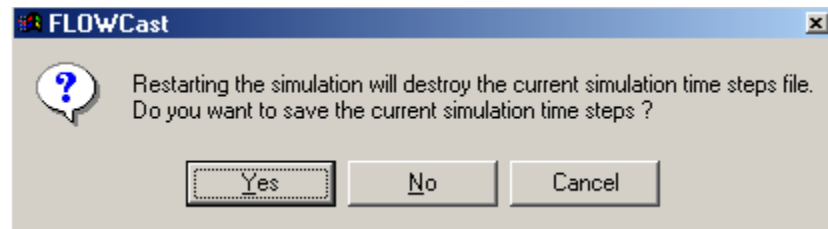


Click the “Stop Sign” button. You may need to wait a few seconds for the simulation to actually stop, as FLOWCast needs a little time to organize its data for the stop.

### Restart filling simulation



Use this function only if you want to restart a filling simulation FROM THE BEGINNING. If you choose this option, the system will show the following message:



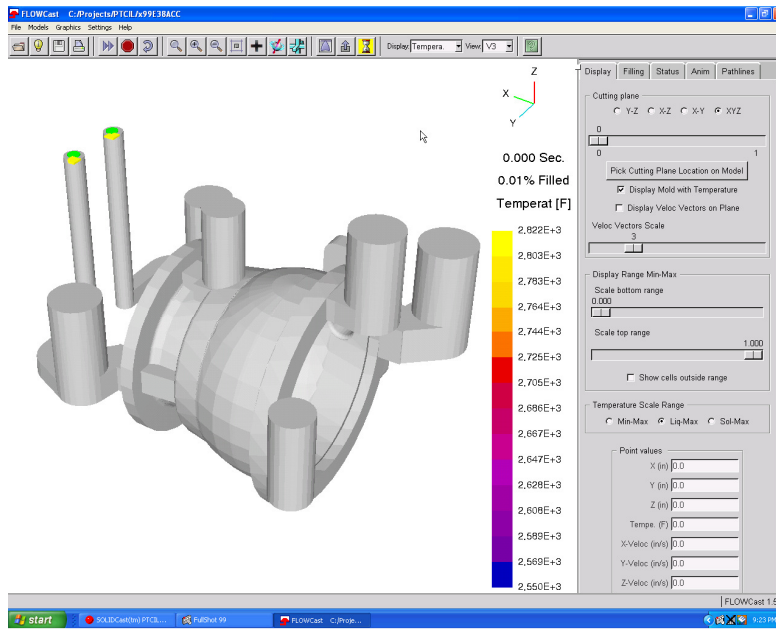
You now have the opportunity to save the time step data produced so far. Once data has been saved, the system will erase the data in memory, and restart the simulation from the beginning.

### Reset zoom



When FLOWCast is first activated, the model display is automatically reset to full size. That is, the system sizes the picture so that the whole model can be viewed. There are a series of three zoom functions that can be used to adjust this view, the zoom in and out functions increase or decrease the picture by 10% each time their icon is clicked. The Rubber Band Zoom allows you to drag the mouse across the section you want viewed. The Reset Zoom function will restore the zoom distance and center the model, but will not change the current orientation.

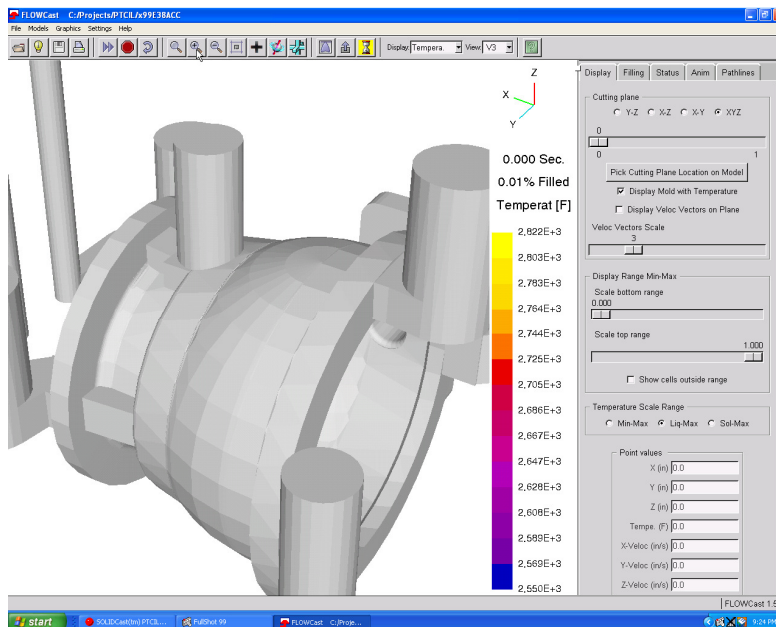
A typical full size view is shown here:



**Zoom In**



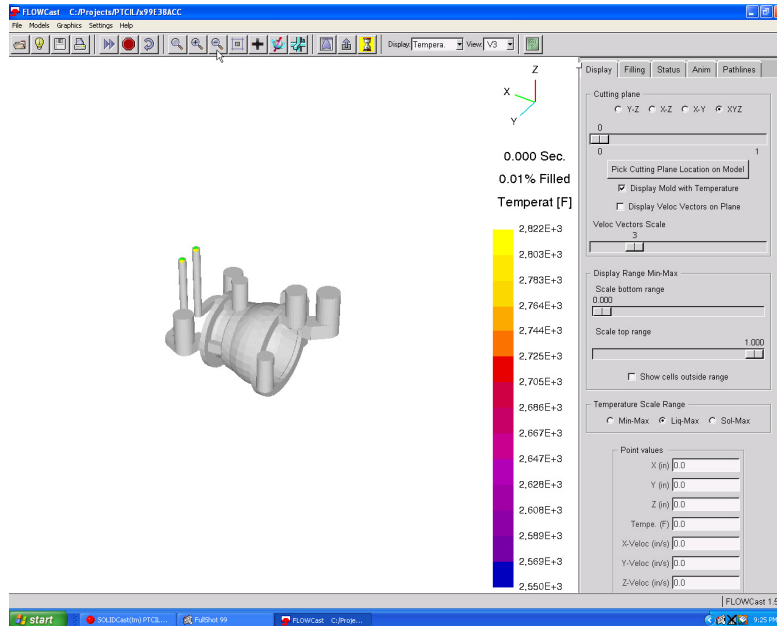
Each time you click on the Zoom In icon, the screen display will move closer by 10%, as shown here:



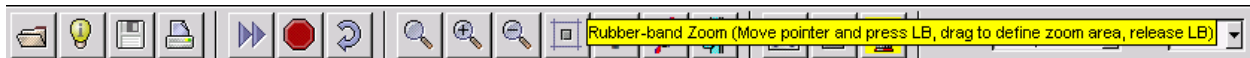
### Zoom Out



For each click on the Zoom Out icon, the screen view will move back by 10%, as shown here:



### Rubber-band Zoom



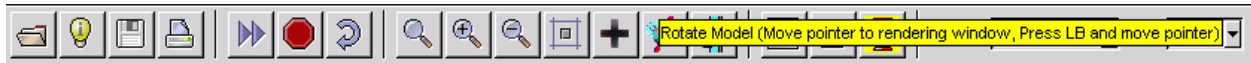
The Rubber-band Zoom function allows you to zoom up on a specific part of the displayed view. Move the mouse cursor to the lower left section of the box you'd like to display, then click and hold the left mouse button. Drag the box until it surrounds the area you want displayed, and then release the mouse button. The system will size the display accordingly.

### Set Zoom and Rotation Centers



This function allows you to choose the point on your model where zooming and rotating take place. To use Set Zoom and Rotation Centers, first click the icon so that it is depressed. Then move the cursor onto the model view window and click the left mouse button at the desired point. This will now become your new point for zooming and rotating.

### Rotate Model



The Rotate Model function allows you to drag the model to any arbitrary orientation, similar to what you do in SOLIDCast. To activate Rotate Model, click on the Rotate Model icon so that the icon is depressed. Then move the cursor to any point on the model display window, press and hold the left mouse button and drag the cursor across the screen. Dragging the cursor will rotate the model. Once you have the desired orientation, you can click the Rotate Model icon once again, to lock the model orientation in place.

### Pan/Translate Model

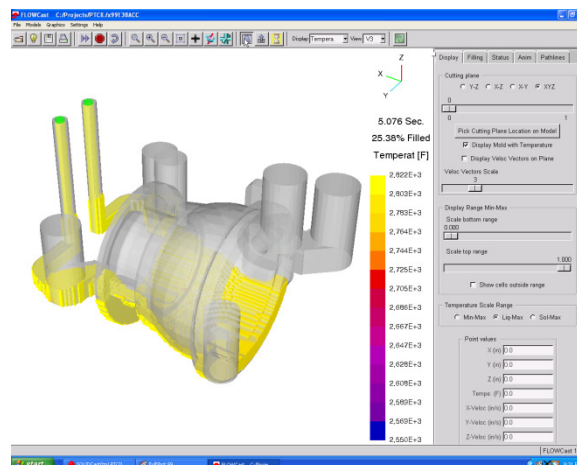


The Pan/Translate Model function lets you drag the model back and forth within the existing view window. This will not change the part orientation, only the position in the window. To activate Pan/Translate Model, click on the Pan/Translate Model icon. Then move the cursor into the model view window, press and hold the left mouse button, and drag the cursor in the window. This action will drag the model itself back and forth in the window. Once the model is in the desired location, click the Pan/Translate Model icon once again to lock the model view in place.

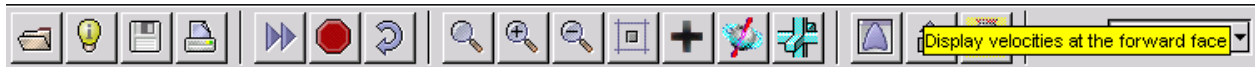
### Display mold cavity using transparency



The normal display during a filling simulation shows the model in opaque colors, and the filling material is drawn over the top. By selecting this function, you can turn the outside of the casting and risers translucent, and can watch the filling sequence from the outside in. In some cases this may make the colors a bit more difficult to see. However, it gives a more realistic feeling to the filling of the mold. An example is shown here:



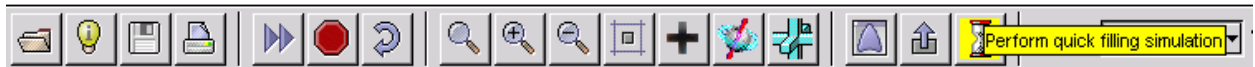
**Display velocities at the forward face**



When displaying velocity during a simulation, each node is colored, and that color represents the velocity. Actually, the velocity is calculated at the interface between the nodes, so there are two options when displaying velocity; the backward face and the forward face. The default is the backward face.

For example, if a node has a forward face on the boundary of the casting, then that velocity will be zero, whereas the backward face velocity will be non-zero if the interface is inside the casting. By clicking on this icon you can toggle the display back and forth between the forward face display and the backward face display.

**Perform quick filling simulation**



The hourglass button is used to toggle between Quick and Full filling simulations. If this button is depressed, you are in Quick Mode. If it is “popped out”, you are in Full Mode. When FLOWCast is first started in Interactive Mode, it is in Full Mode.

The Quick Fill Simulation ignores momentum effects, so it can be run much more quickly than a Full CFD Fill Simulation. However, it is not as accurate a simulation, and you will not get any Pressure information from a Quick Fill Simulation.

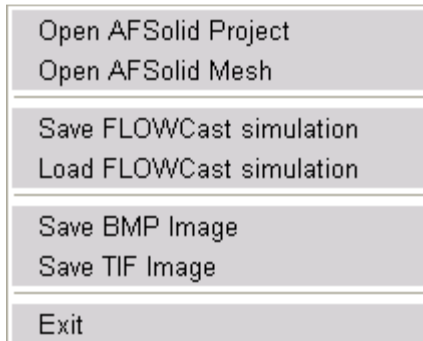
If you are only looking for a simple, fast filling simulation, it is probably better to use the SOLIDCast filling algorithm. If you do this, make sure you go into the Set Simulation Parameters Utility and check a box to fine tune the filling for either horizontal or vertical filling.



## UNIT 5: The File Menu

---

The File menu has the following seven options:



These functions will be explained in the remainder of this unit.

### Open AFSolid Project

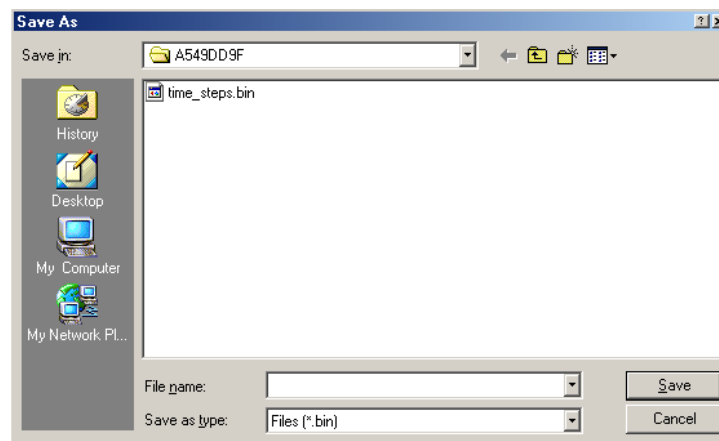
This function was originally used to select a project to run a flow simulation on. Since FLOWCast is now launched automatically, this function is no longer needed.

### Open AFSolid Mesh

This function was originally used to choose which mesh you wanted to use for a flow simulation, if a given project contained more than one mesh. Since FLOWCast is now launched automatically from a mesh file, this function is now longer needed.

### Save FLOWCast simulation

When FLOWCast is running normally, the system will automatically save data at prescribed intervals, which, by default, are at every 5% full. You can, at any time, however, stop the simulation and save the data to a file. When you select this option, you will see a Save As... screen similar to this:



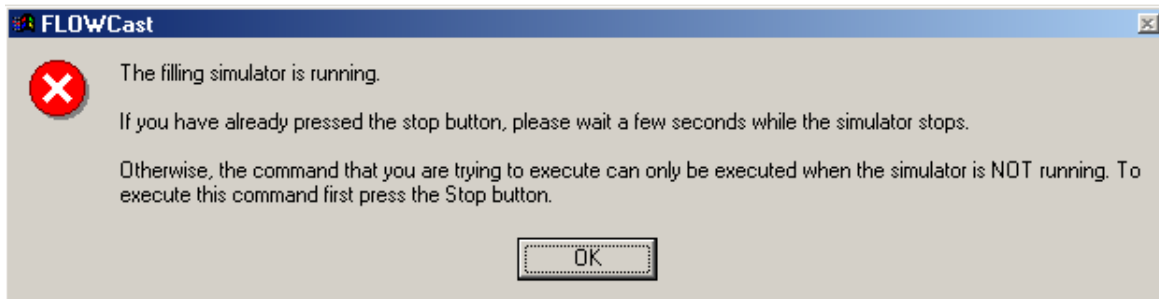
Simply type in a file name and click on Save. You can navigate to any folder you wish.

If no time step data has been created yet, you will see the following message:



This simply means that you are too early in the simulation process for any meaningful data to have been created.

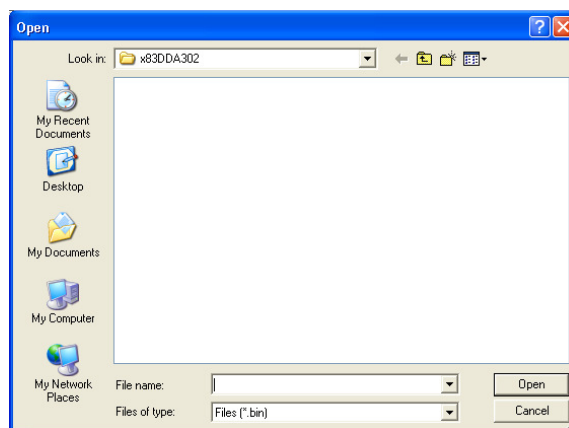
If you try to save the data while the simulation is still running, you will see the following:



In order to save data manually, the flow simulation must be stopped. To stop the simulation, click on the stop sign icon on the toolbar. You will need to wait a few seconds for the system to organize its' data before you can save it out to a file.

### Load FLOWCast simulation

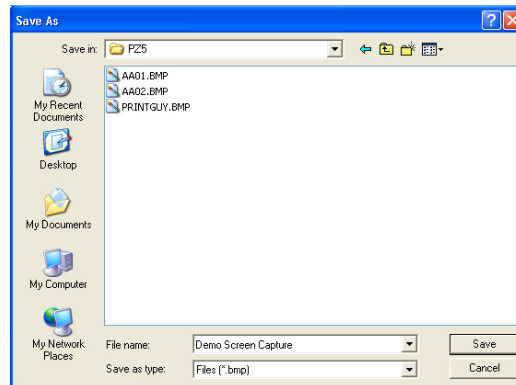
You may want to load existing time step data into FLOWCast. If you make this selection, you will see a window similar to this:



Simulation data is normally saved in a file called time\_steps.bin. In most cases this file will be loaded if you are starting or continuing a flow simulation.

### Save BMP Image

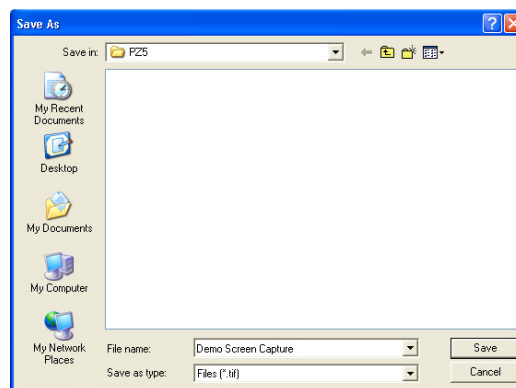
At any point, you may want to capture and save the screen image. This can be saved in either BMP or TIF format. If you select BMP format, you should see a window similar to this:



Type in a name for the file and click on the save button. You may wish to change the screen background from black to white before saving the image. This can be done from the Graphics Menu.

### Save TIF Image

At any point, you may want to capture and save the screen image. This can be saved in either BMP or TIF format. If you select TIF format, you should see a window similar to this:



Type in a name for the file and click on the save button. You may wish to change the screen background from black to white before saving the image. This is done from the Graphics Menu.

### Exit

Select this function to end a flow simulation and exit the FLOWCast program. The SOLIDCast window will be minimized, so click on the button on the toolbar to restore SOLIDCast.



## UNIT 6: The Models Menu

---

This menu is not active at the current time.



## UNIT 7: The Graphics Menu

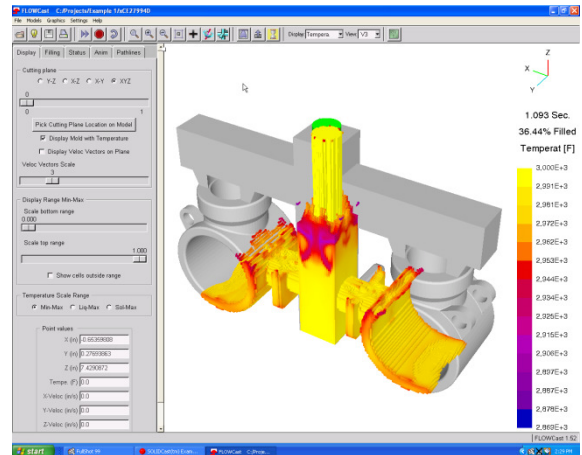
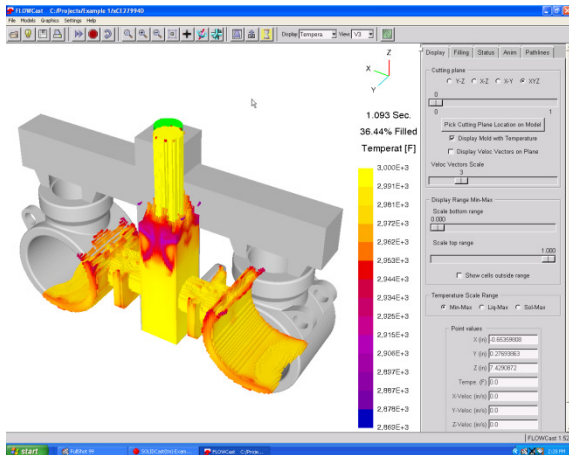
This menu contains the following six options:

- Drawing window position ▶
- ..White (Black) background
- ..Draw reference axis
- ..Draw scale/palette
- ..Perspective
- Graphics card ▶

These options will be explained in the rest of this unit.

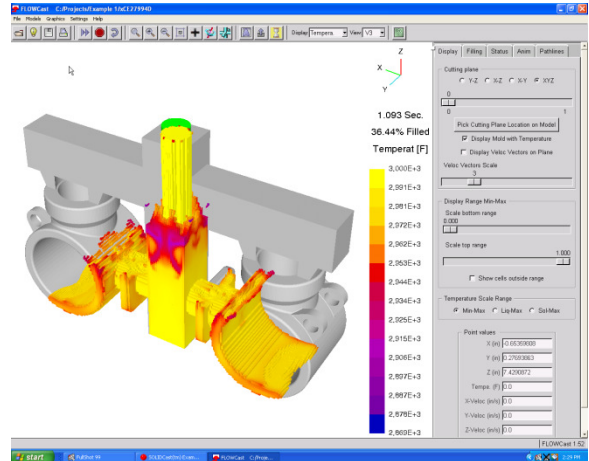
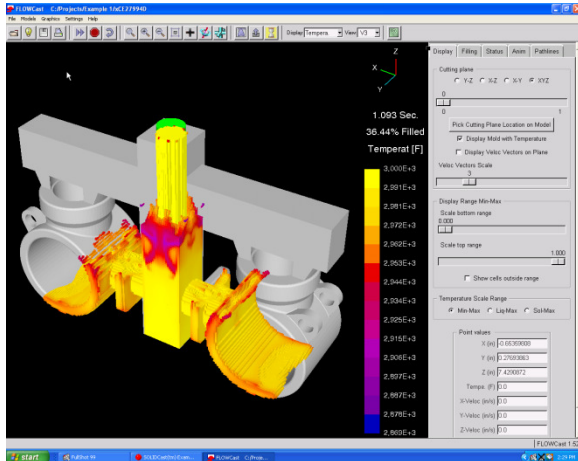
### Drawing window position

This function has two options, On Left and On Right. This specifies which side of the screen the Drawing Window is positioned. The Drawing Window contains the picture of the filling casting during a flow simulation. The following two figures show the Drawing Window on the Left and on the Right:



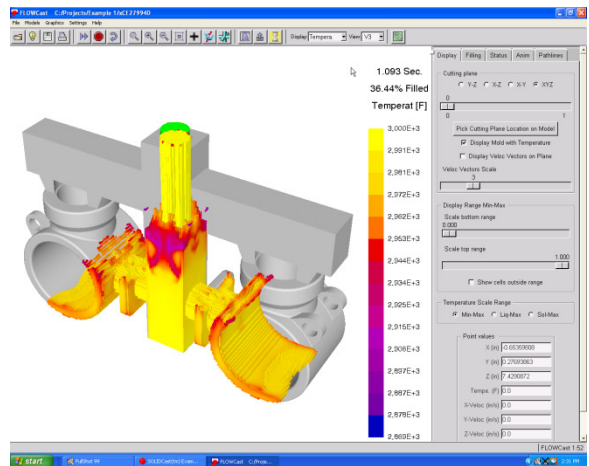
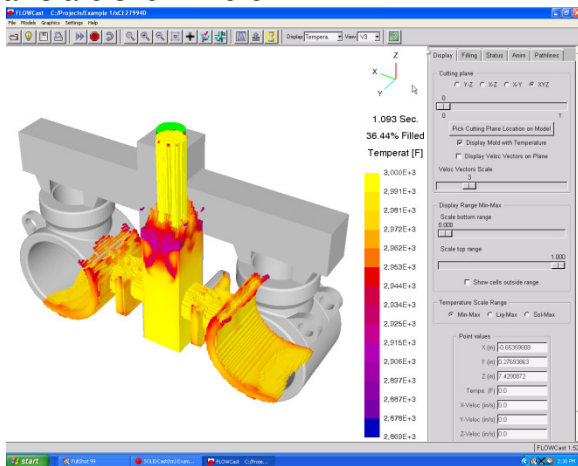
### White(Black) background

This function allows you to toggle the background color of the Drawing Window between black and white. A white background is usually better for screen capture and/or printing. The following two figures show toggled background colors:



### Draw reference axis

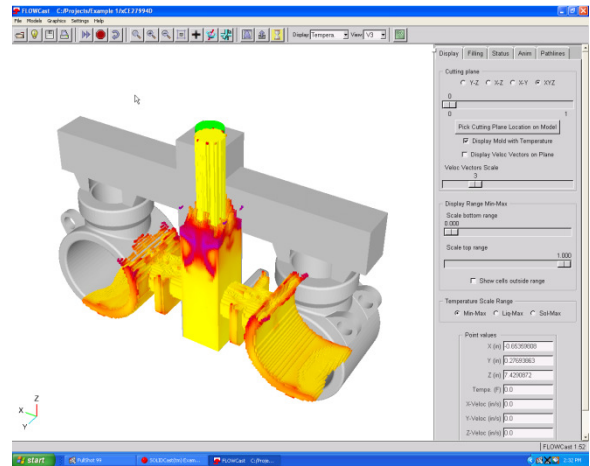
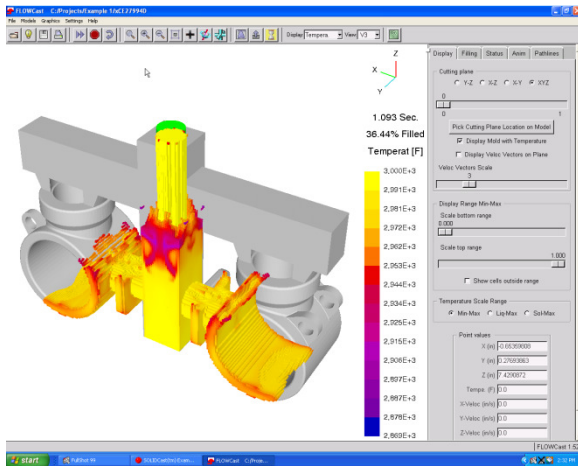
This option is a toggle, which determines whether or not the reference X, Y, Z axis is displayed in the upper right corner of the Drawing Window. Example views with and without the reference axis are shown here:





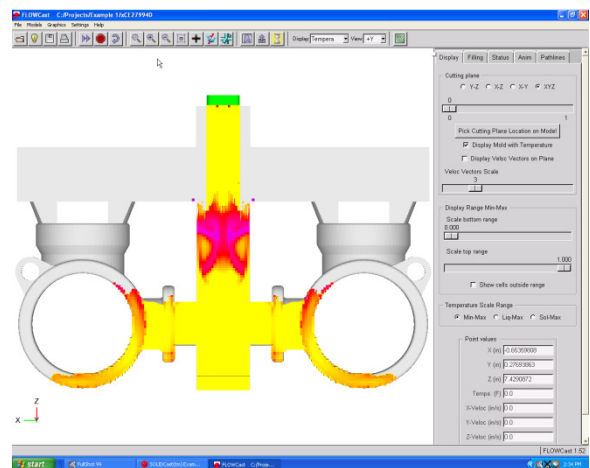
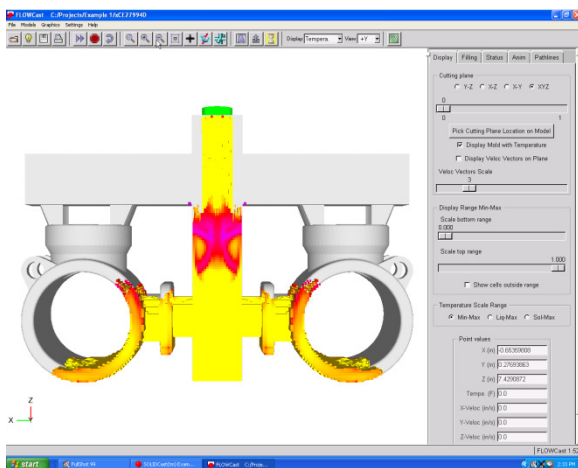
## Draw scale/palette

This option is a toggle, which determines whether or not the color palette and scale are shown on the right side of the plot or not. By toggling the display off, you can reduce the size of the picture, and AVI movies created from the data. Example screens showing the scale/palette on and off are given here:



## Perspective

Perspective Rendering of the Model View can be toggled on and off. This can sometimes be useful when displaying a complicated model. The default is Perspective Rendering on. The figures below show the same view with Perspective Rendering on and off:



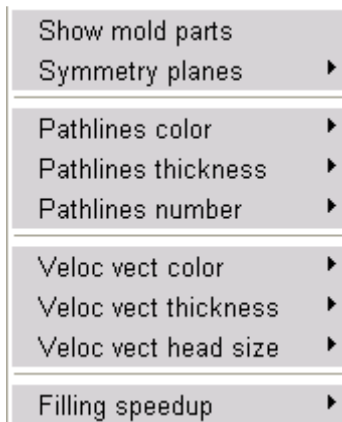
## Graphics card

This option lets you to see the current graphics card settings, view other graphics modes available, and modify the graphics card settings. Since Windows lets you modify these settings, it is normally not necessary to use these options directly. If you have difficulties in making animations or in rendering a picture, use these functions to help troubleshoot the problem.



## UNIT 8: The Settings Menu

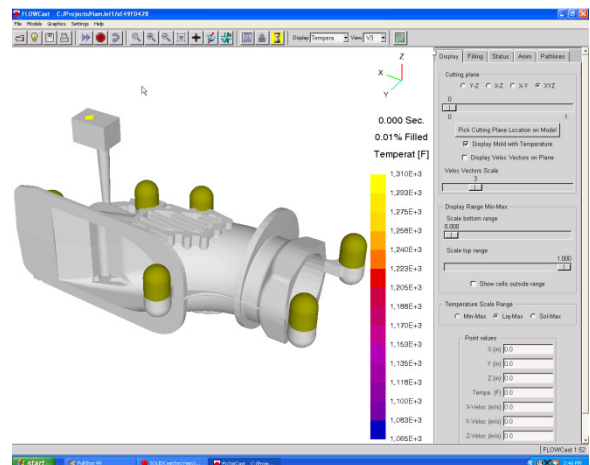
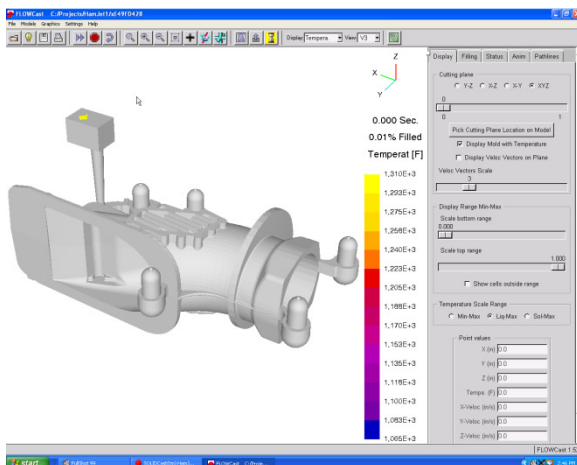
This menu is used to control the color, thickness and number of pathlines displayed when using the Display Particle Tracks Group function. This function is toggled on or off using the Pathlines Tab. It also controls the Filling Speedup parameter. Options available are:



These options will be explained in the rest of this unit.

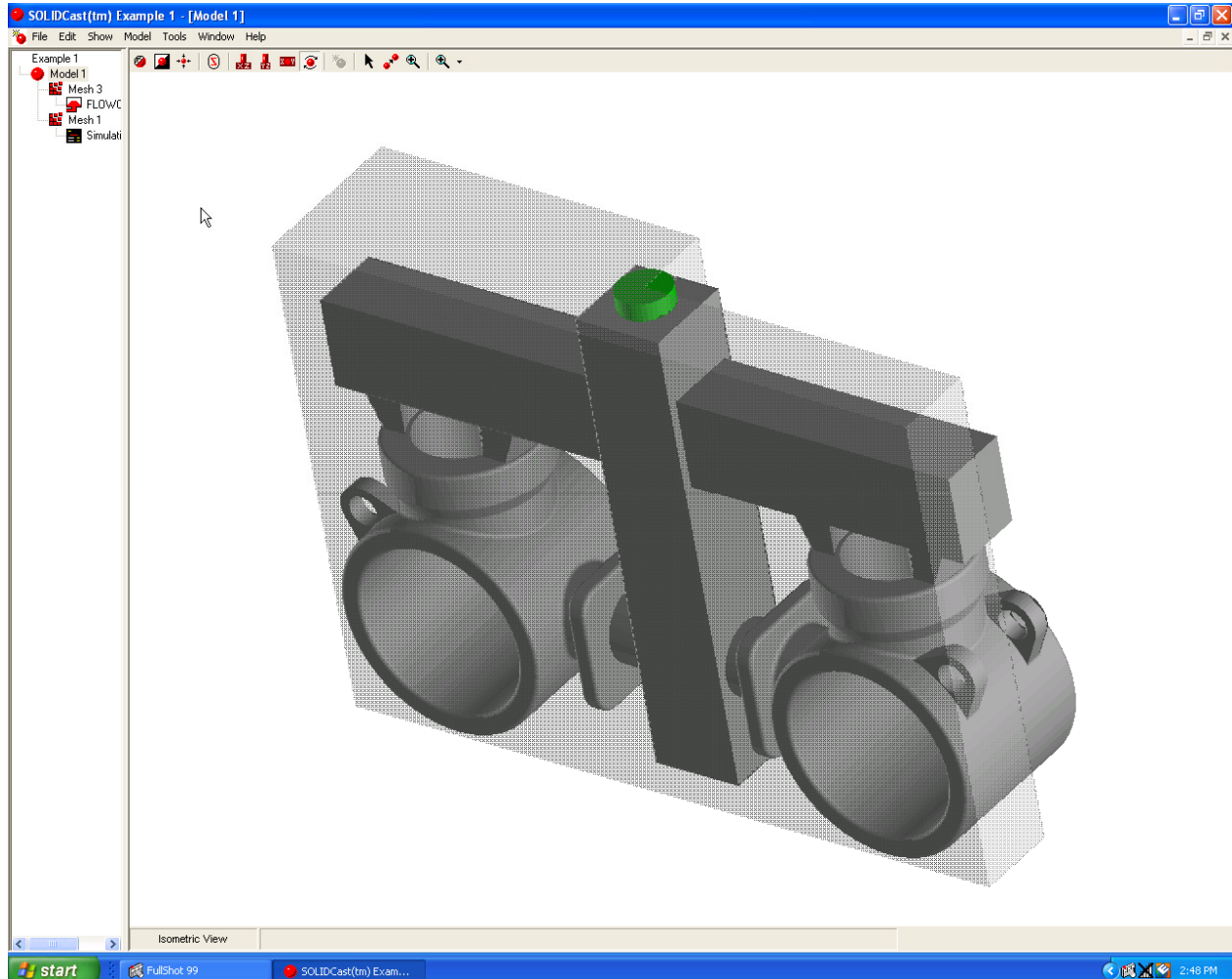
### Show mold parts

In most cases you only look at the metal portion of the model, that is, the casting, gating and risering. There may be times, however, where you would also like to look at items in the mold, such as chills, sleeves, etc. Show mold parts allows you to toggle the display of these extra pieces on and off. The two figures below show a model with mold parts off and with mold parts on.



## Symmetry planes

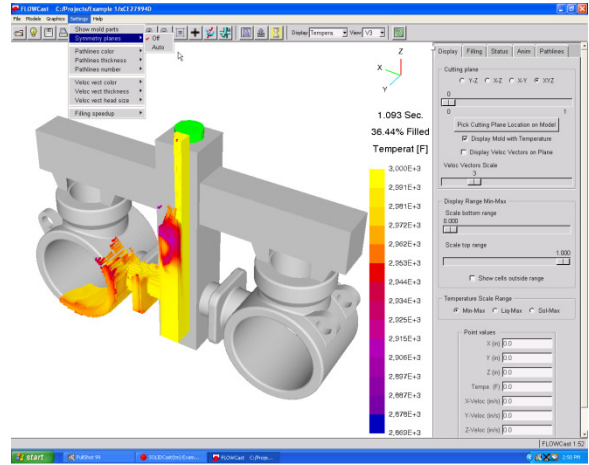
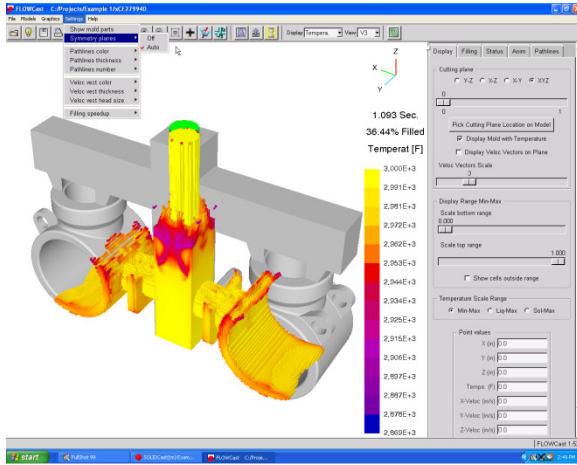
A common method of speeding up simulations is to use one or more planes of symmetry, when the model has mirror images. For example, the model below has two planes of symmetry which are represented by the grayed-out areas:



In this particular case, using planes of symmetry can reduce the model size to one-quarter, without sacrificing simulation quality at all. This also means that results come four times faster!

When a SOLIDCast model is meshed using planes of symmetry, FLOWCast has the ability to display the filling process on the entire model, by mirroring those results on the display. The Symmetry planes function is a toggle switch, which controls whether the display data is mirrored or not. The default state is Auto. That is, the data is automatically displayed on the full model, so that the complete model fills. If Symmetry planes are turned off, then the data will be displayed only on the meshed portion of the model, which may be only one half or one quarter of the full model.

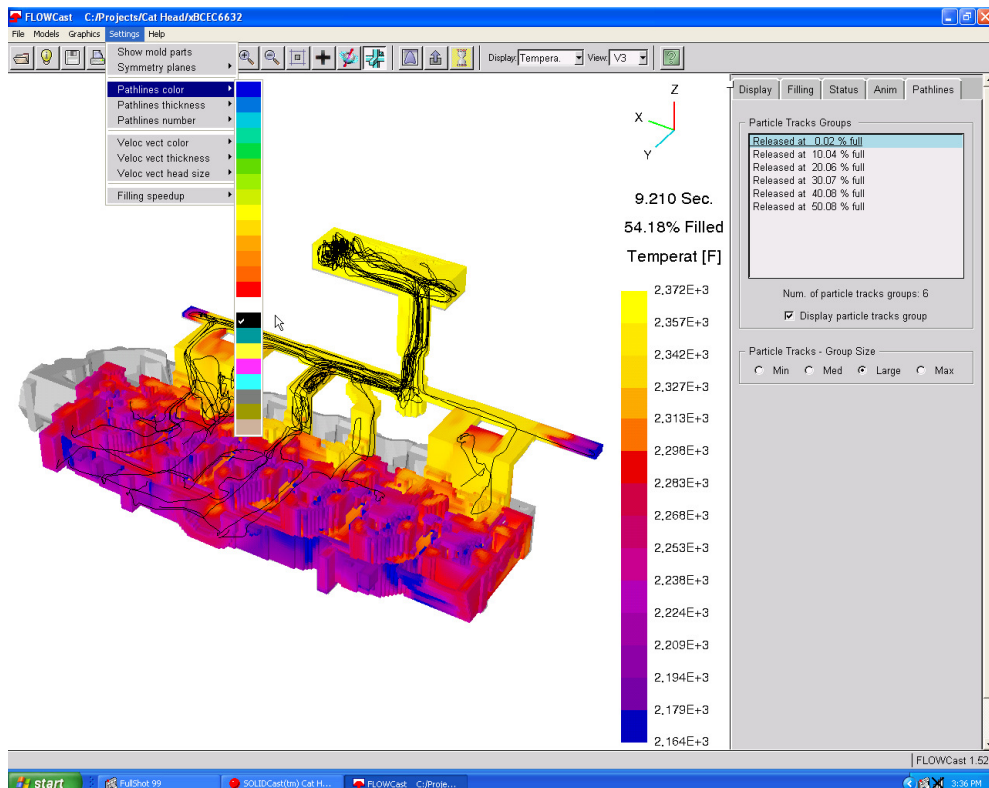
The two figures below show the effect of using the Auto and Off settings for the Symmetry planes function:



There are three functions that control the display quality of metal flow pathlines. They are described here.

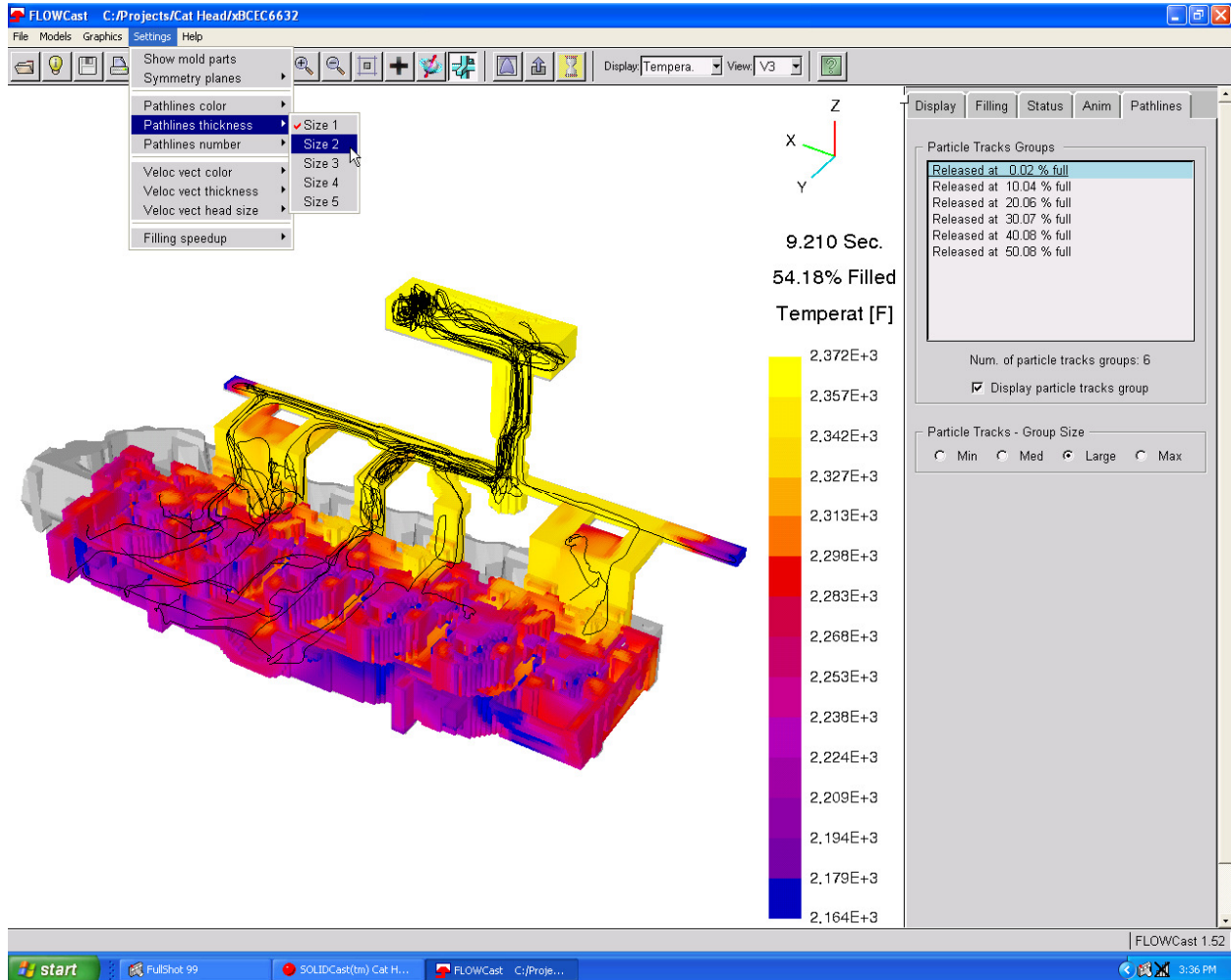
### Pathlines color

Select Settings...Pathlines color, to choose the color of the flow lines from the following palette:



### Pathlines thickness

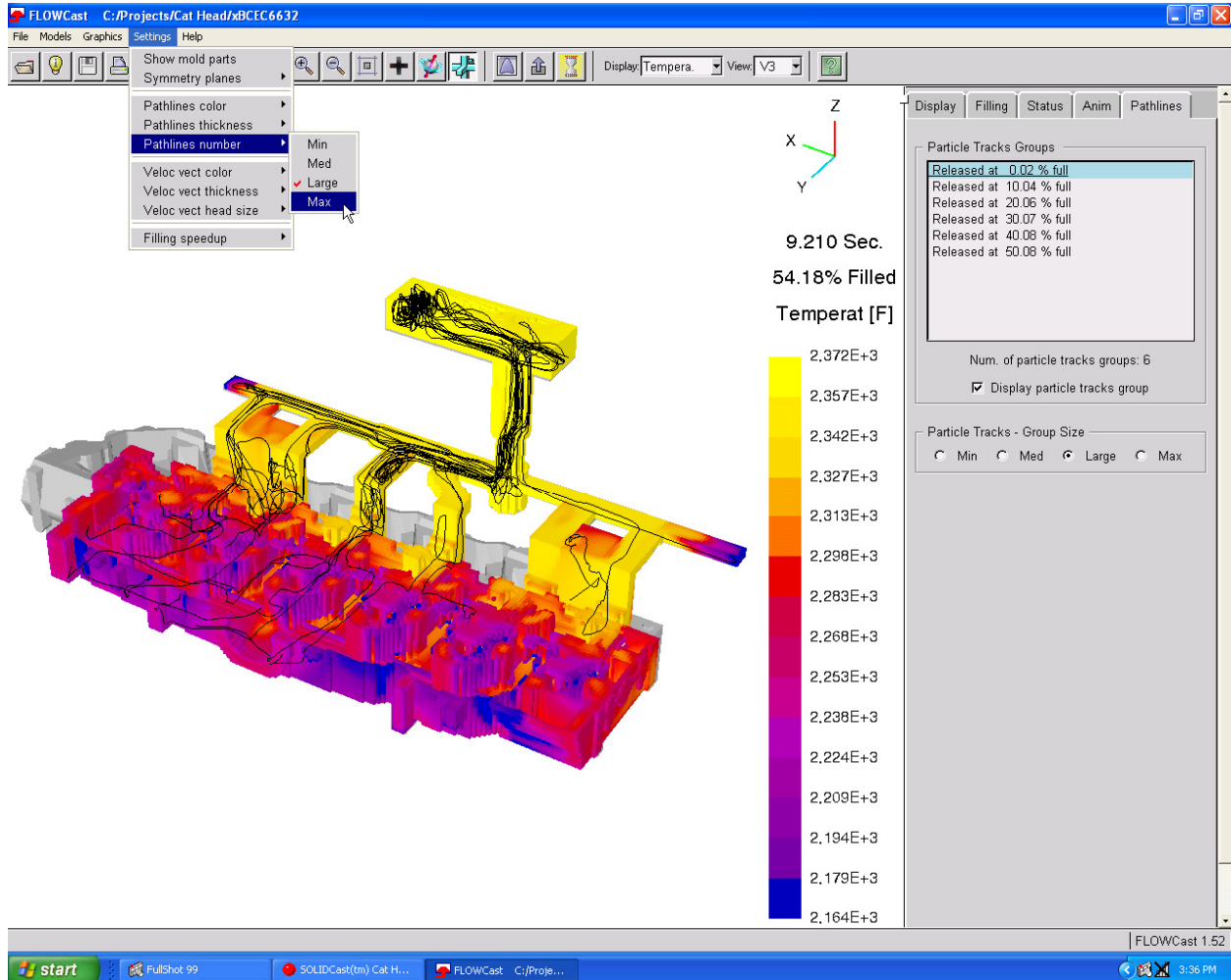
If you select Settings...Pathlines thickness you have the opportunity to change how heavy the pathlines are drawn on the model, as shown here:





### Pathlines number

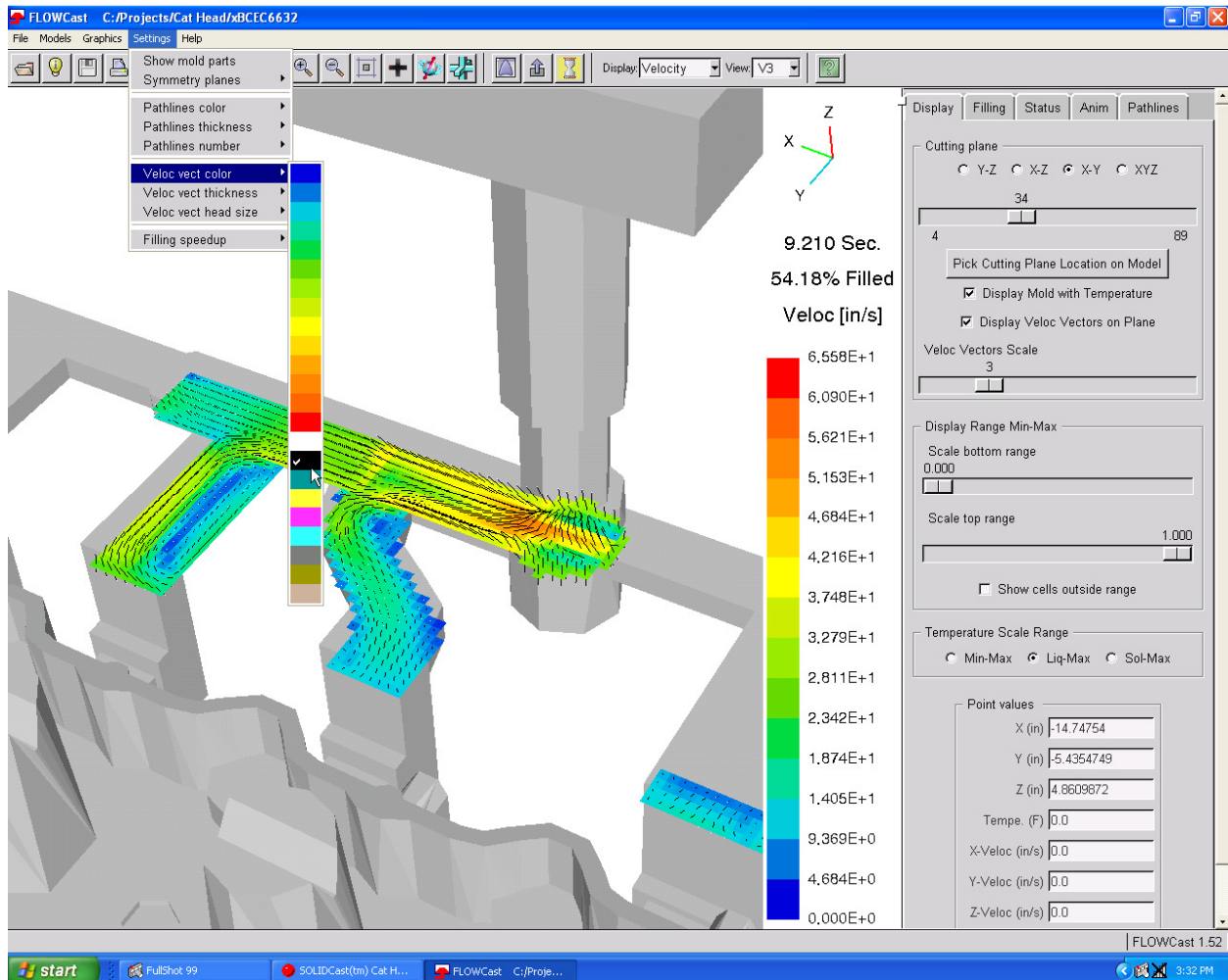
If you select Settings...Pathlines number you can choose the relative number of pathlines that are displayed, from Min, Med, Large and Max. This option is displayed below and is also available on the Pathlines Tab:



As with the pathlines function, there are three controls for display of velocity vectors.

**Veloc vect color**

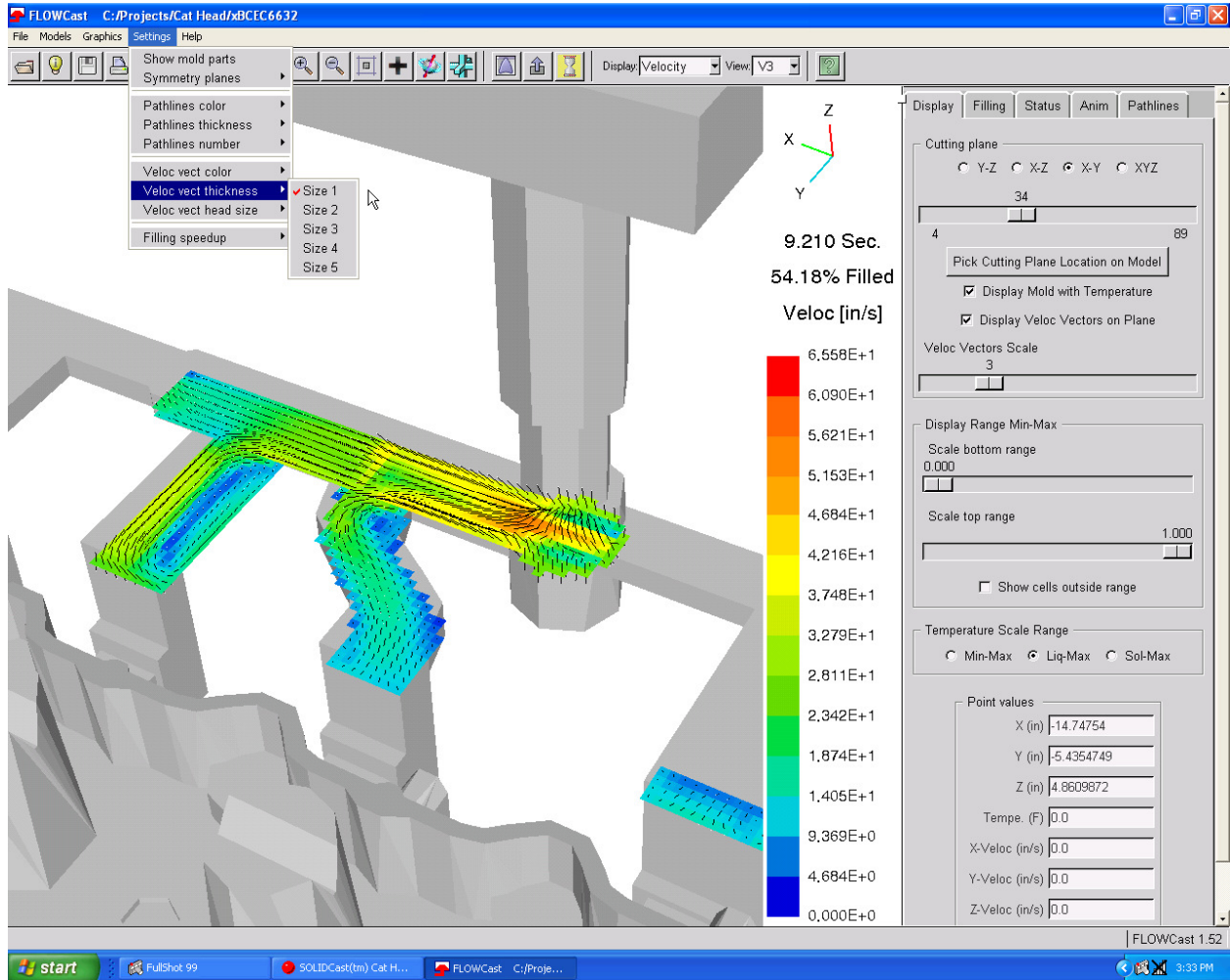
Select Settings...Veloc vect color, to choose the color of the flow lines from the following palette:





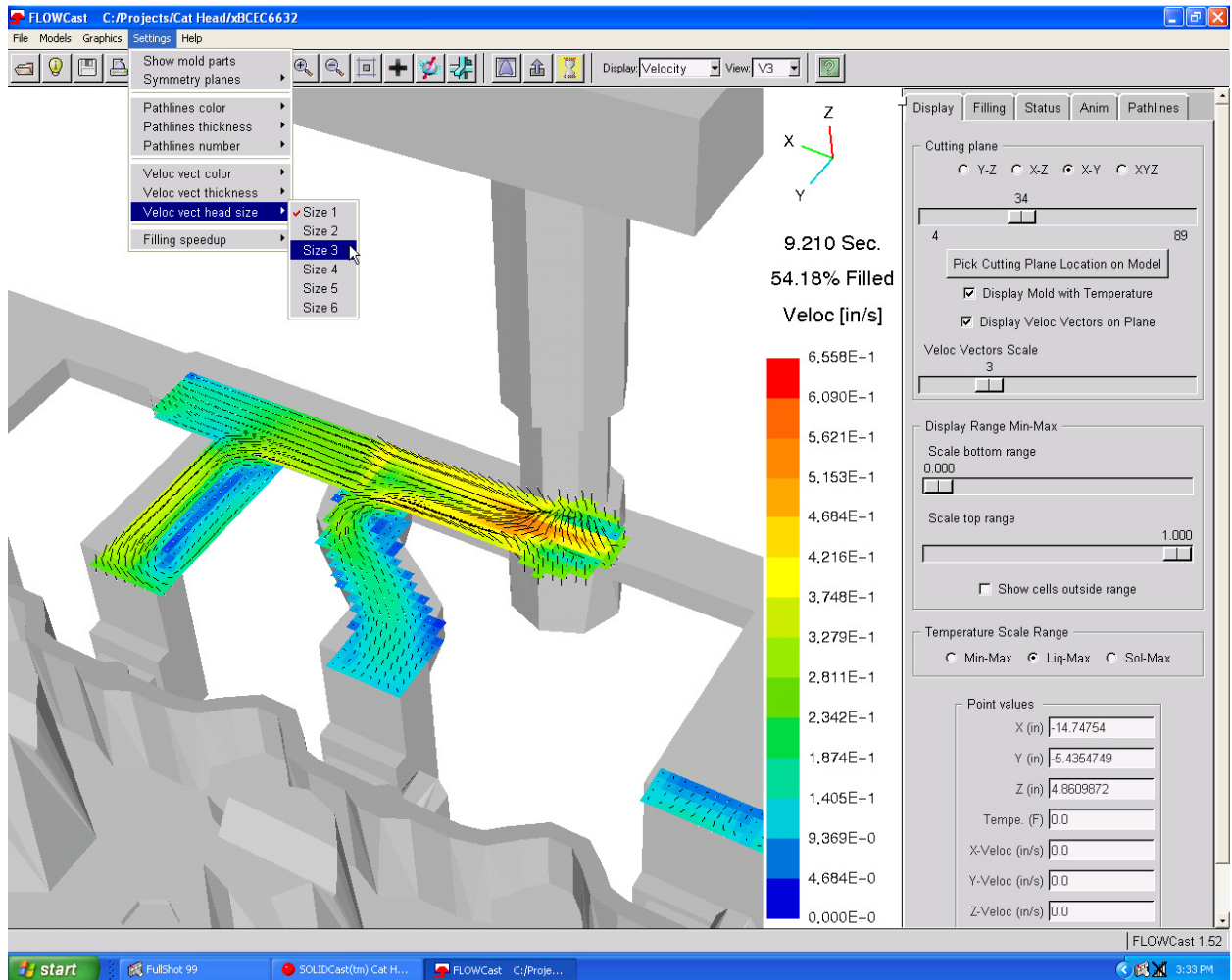
**Veloc vect thickness**

If you select Settings...Veloc vect thickness you have the opportunity to change how heavy the velocity vector tails are drawn on the model, as shown here:



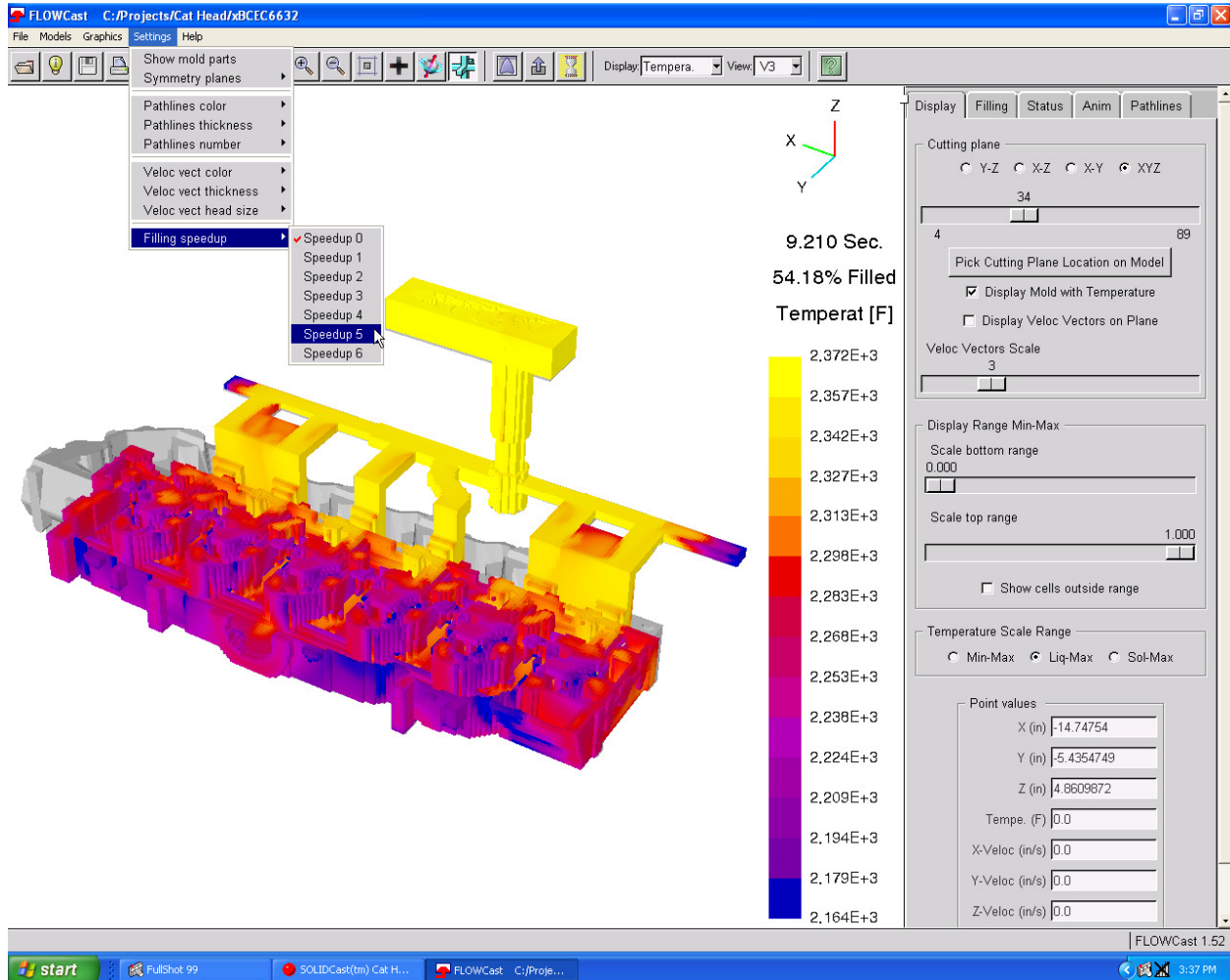
**Veloc vect head size**

This function controls how large the white head of the velocity vector is.



### Filling speedup

If you select Settings...Filling speedup, you have the option of adjusting how fast the FLOWCast Full Simulation runs. Options are a speedup of 0, 1, 2, 3, 4, 5 or 6.



The speedup number refers to the number of pressure field computations to skip for every liquid/gas interface advance and temperature change due to convection/conduction computations. 0 represents no skipping of calculations, and no additional speedup.

By skipping some of the pressure computations, the simulation will go faster, but at the sacrifice of some simulation accuracy. As the speedup increases, you will tend to notice a 'rougher' liquid/gas interface, with more splashing particles, or 'flotsam'.

Higher speedup values should be used on initial simulations, or with very large models, where computation time becomes a significant factor. Final simulations should be done with a speedup of 0 to insure the highest accuracy. This is the default condition.



## UNIT 9: The Help Menu

---

This menu is not active at the current time. Please use the information provided in this training manual to answer your questions, or direct your questions to Finite Solutions Inc. or your local agent. You can contact Finite Solutions Inc. at the following addresses:

David Schmidt  
Finite Solutions Inc.  
4769 Highland Park Dr  
Slinger WI 53086-9441  
Tel 262 644 0785  
Fax 262 364 2537  
Email [Dave@finitesolutions.com](mailto:Dave@finitesolutions.com)

Larry Smiley  
Finite Solutions Inc.  
2931 Hamilton New London Rd  
Hamilton OH 45013  
Tel 513 737 7300  
Fax 513 737 7302  
email [Larry@finitesolutions.com](mailto:Larry@finitesolutions.com)

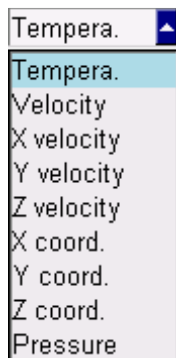


## UNIT 10: Display and View Controls

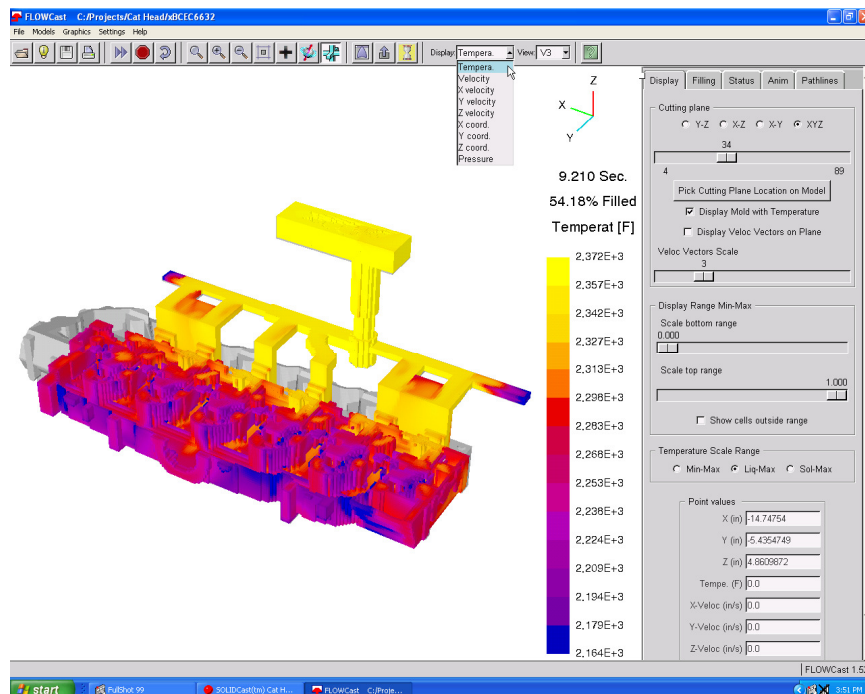
The Display and View Controls are on the right side of the FLOWCast Toolbar. These controls give you great flexibility in what data is displayed and how the data is viewed during a simulation.

### Display Control

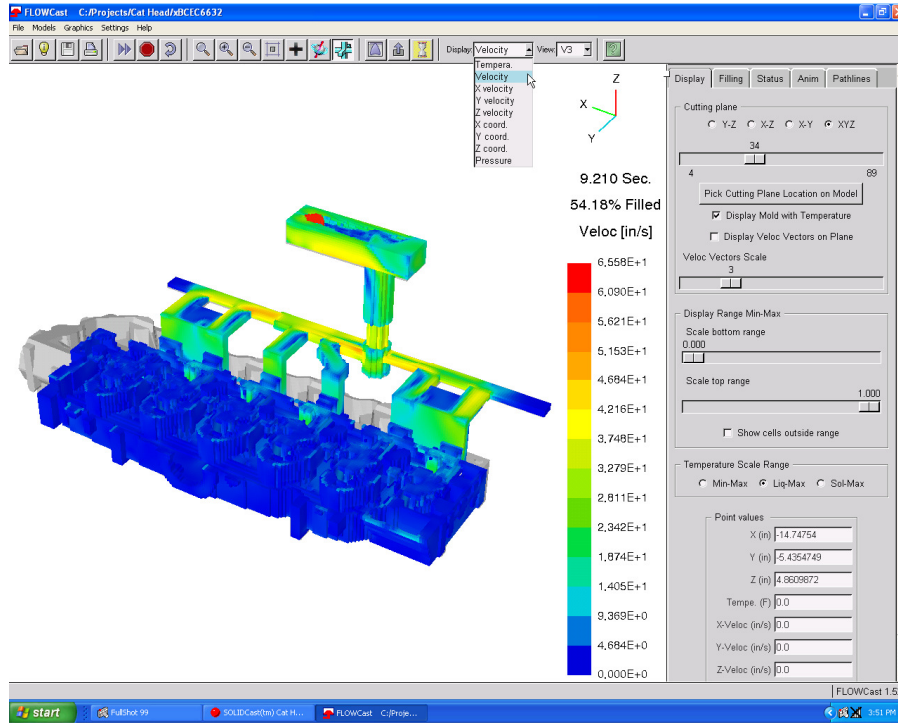
The Display Control allows you to switch between nine different types of data, as follows:



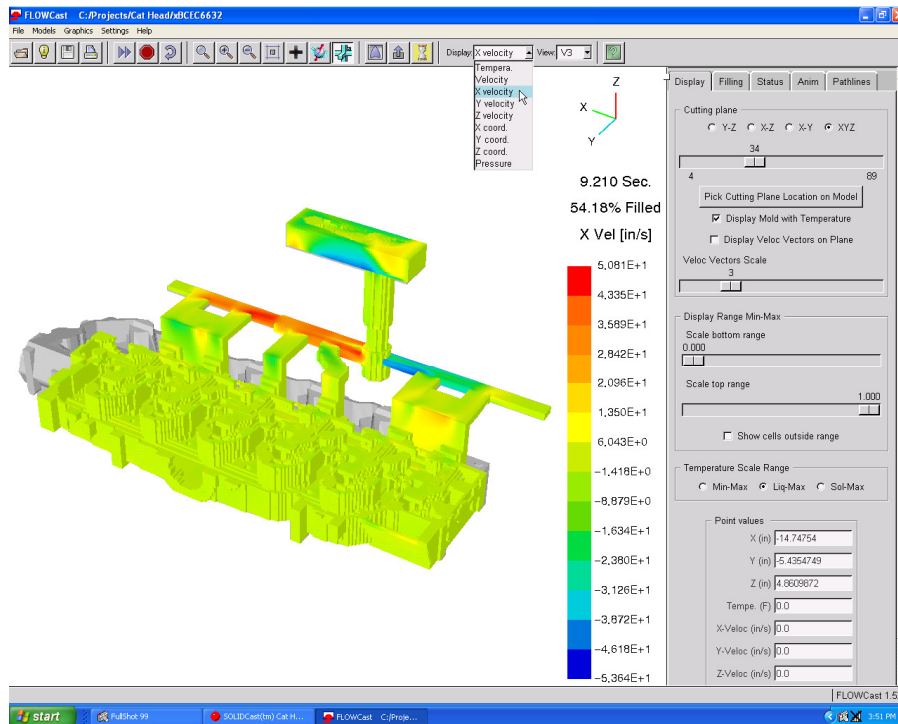
The default data displayed is Temperature. Examples of each of these data types are shown in the following figures.



### Temperature Display

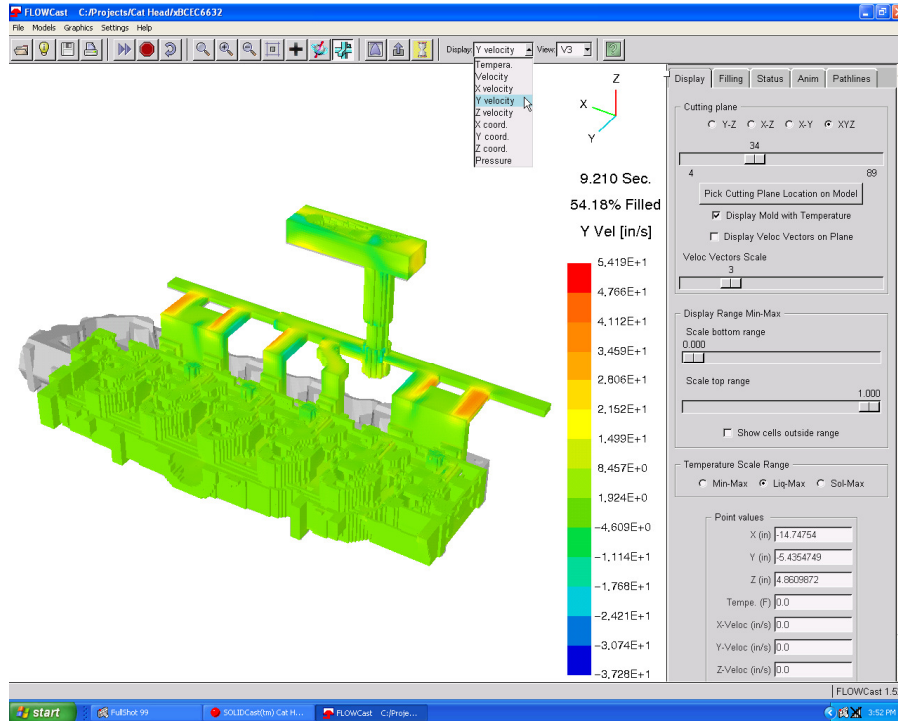


Velocity Display

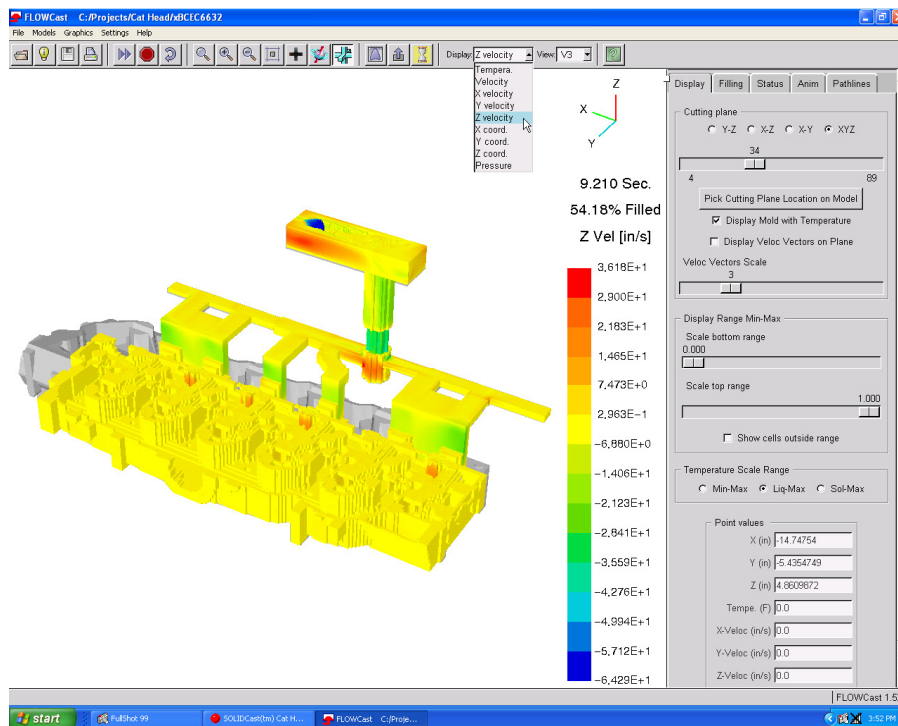


X Velocity Component Display

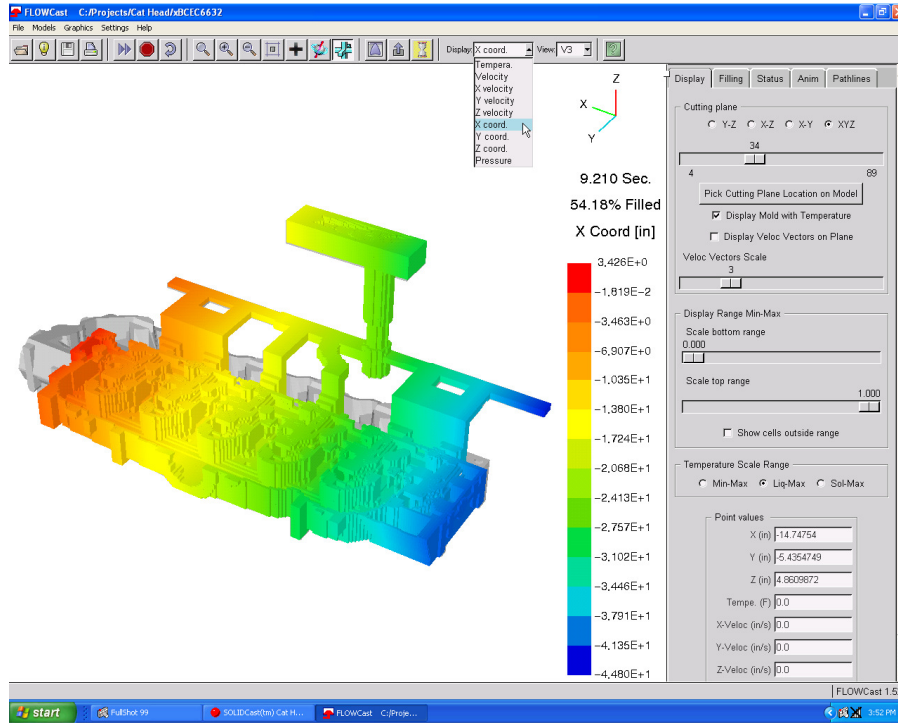




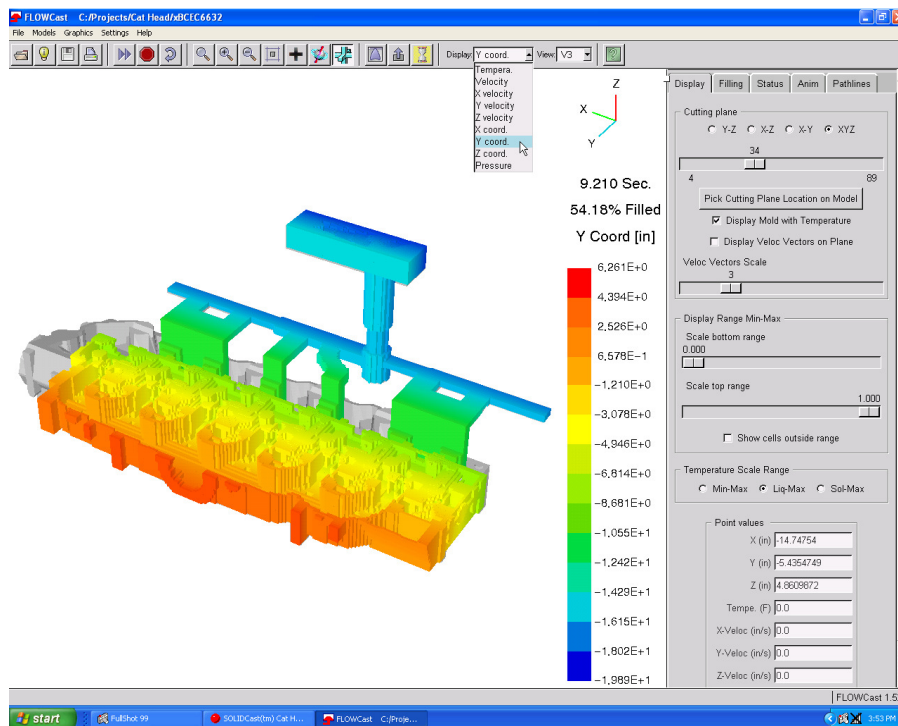
**Y Velocity Component Display**



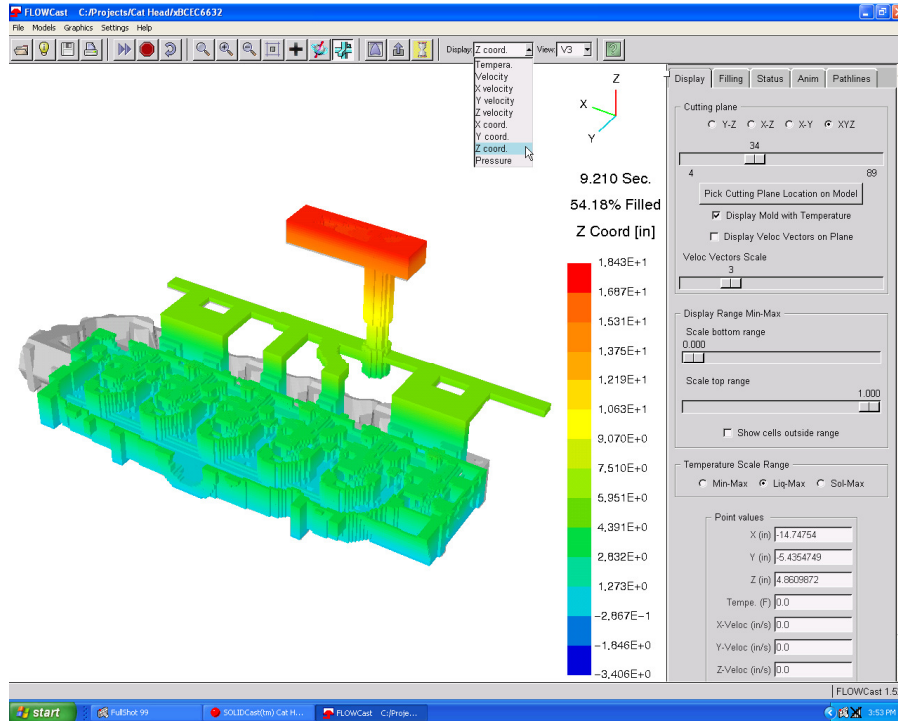
**Z Velocity Component Display**



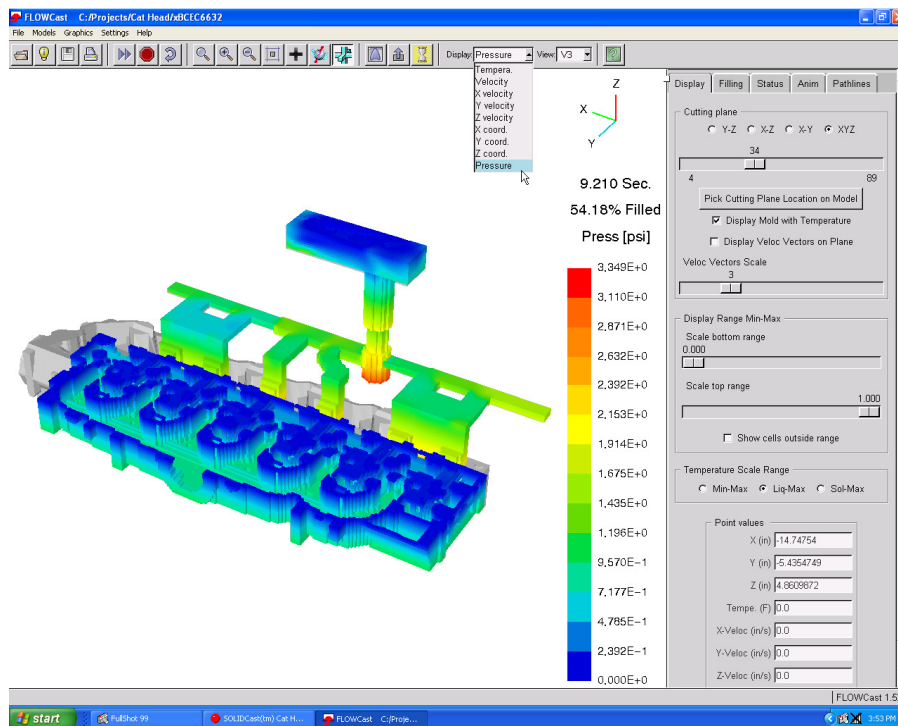
**X Coordinate Display**



**Y Coordinate Display**



Z Coordinate Display

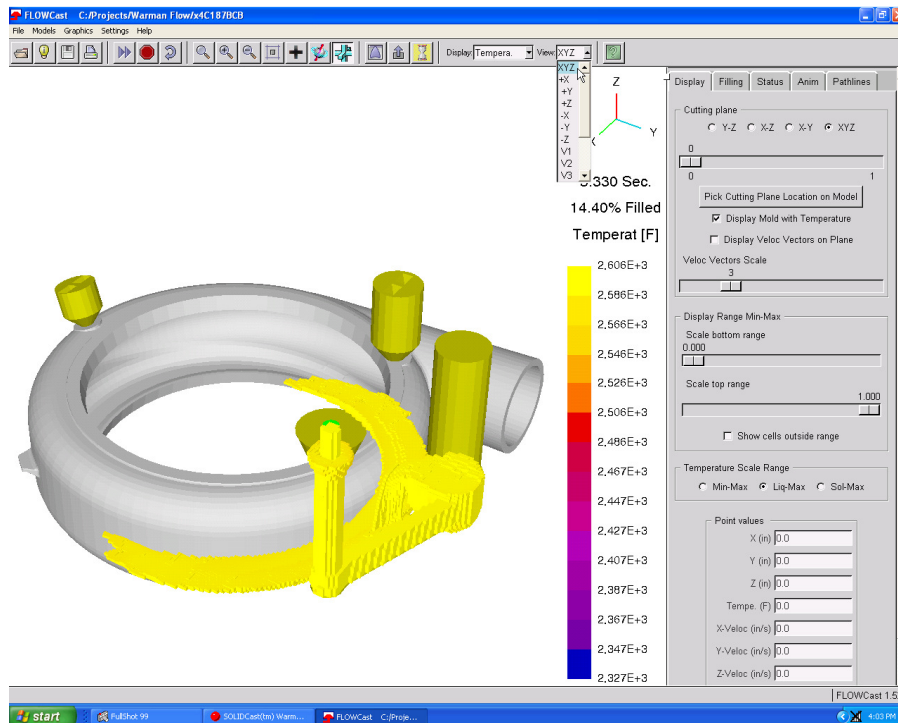


Pressure Display

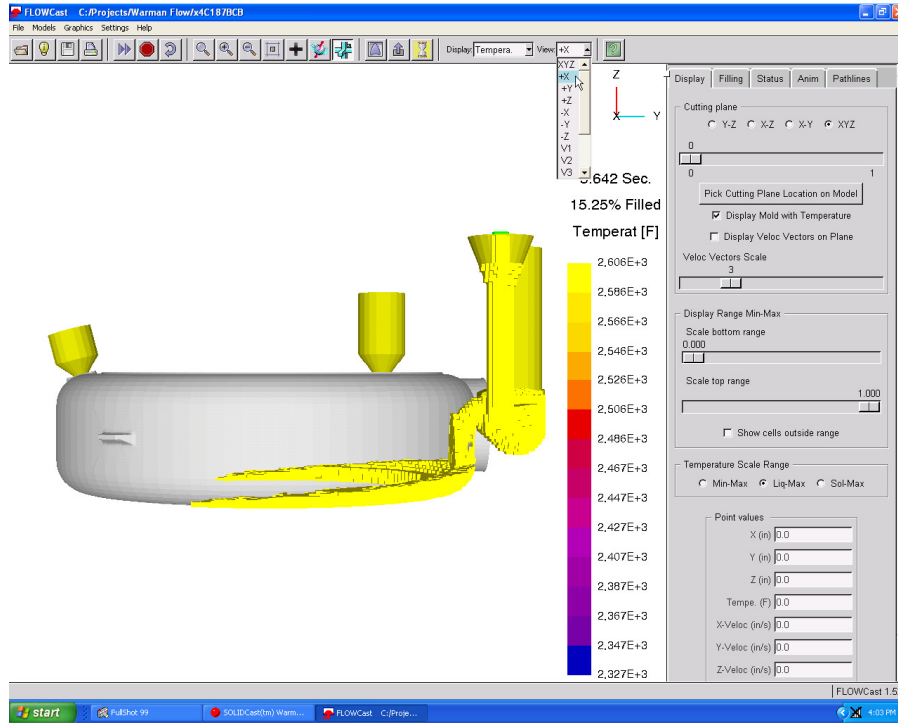
### View Control

The View Control allows you to change the angle from which the model is viewed during a simulation. You can view the model in the positive or negative X, Y or Z directions, plus nine rotated isometric views. The V3 view is the default view. After a simulation is run, you can change the view and then create a new AVI movie of the data from a different angle. The view can be changed at any time, even while a simulation is running.

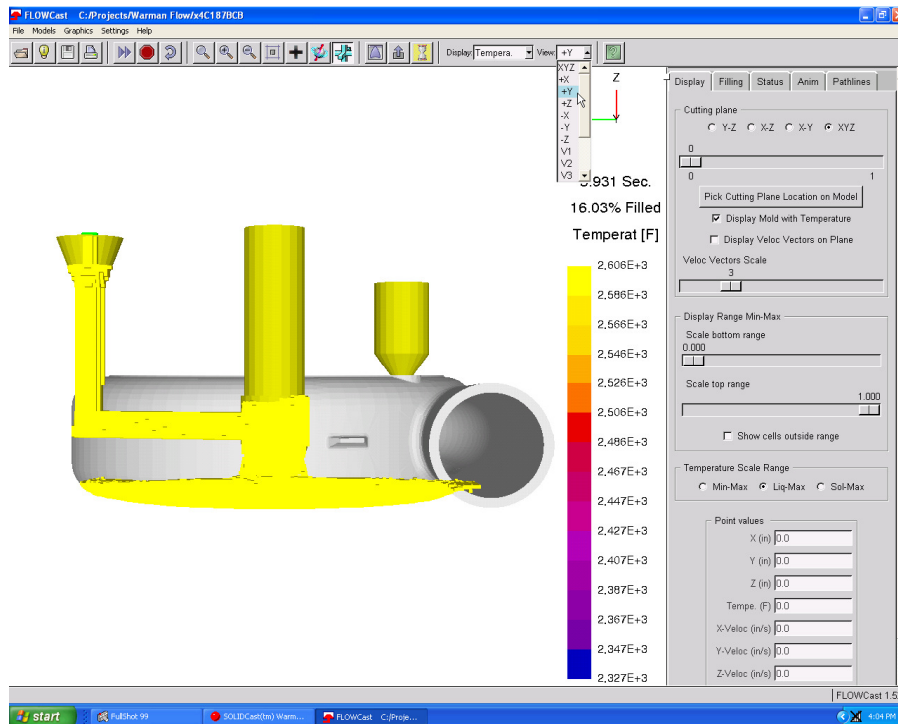
Samples of each of the view choices are shown in the following figures:



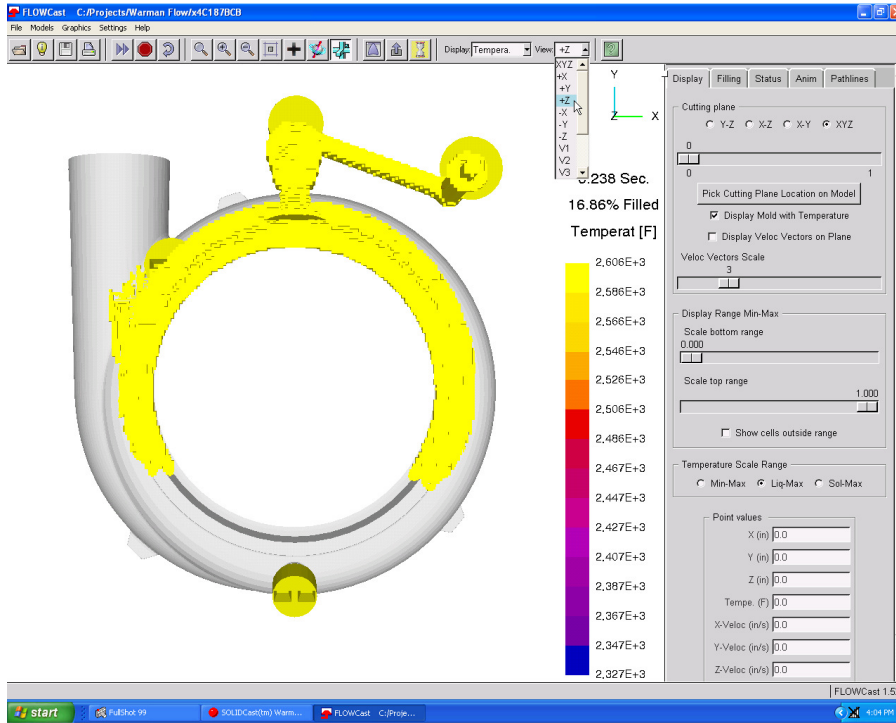
### XYZ View



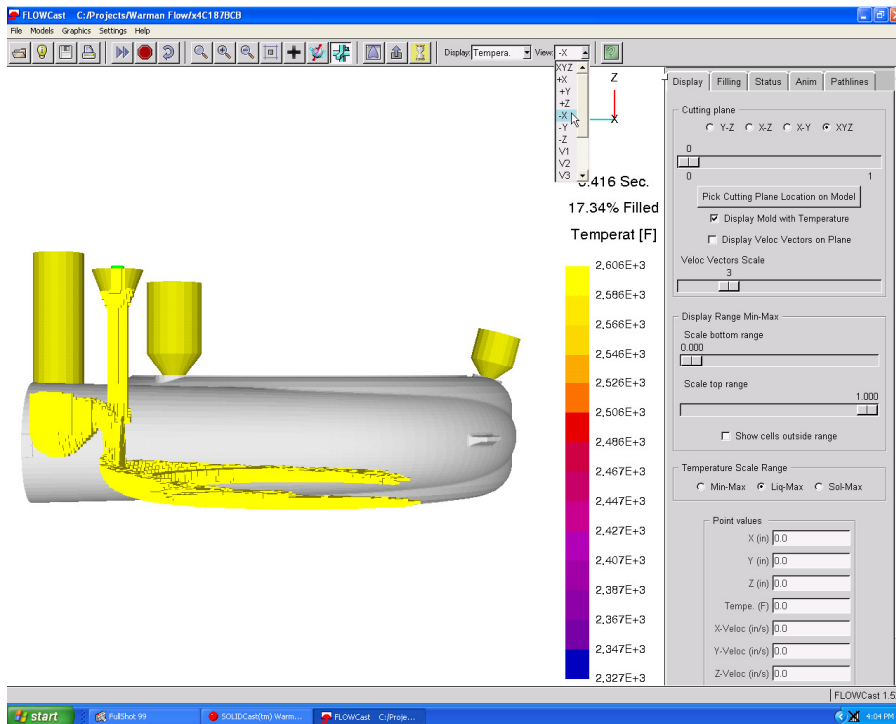
**+X View**



**+Y View**

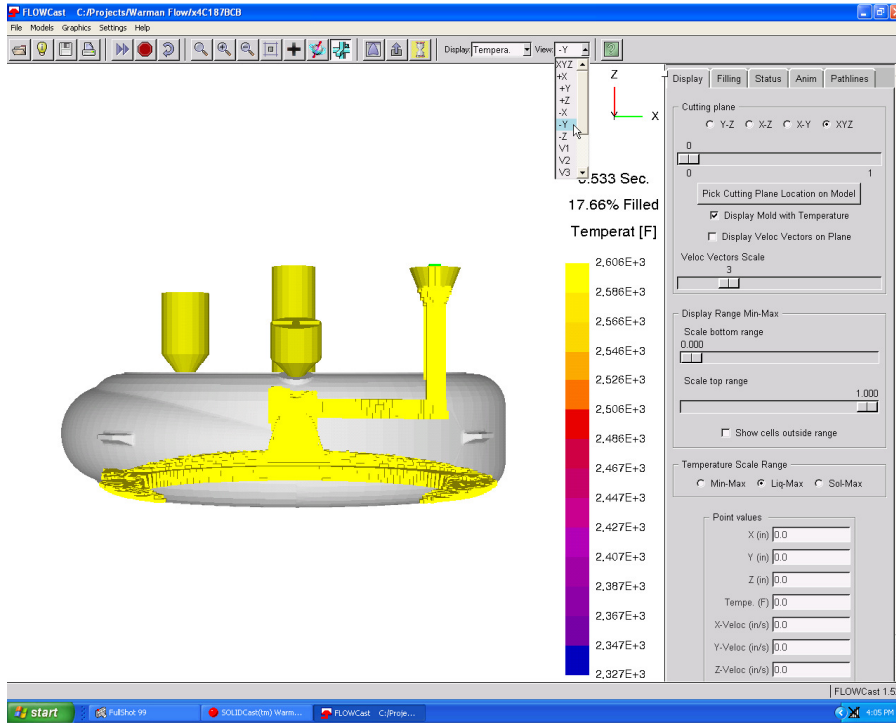


**+Z View**

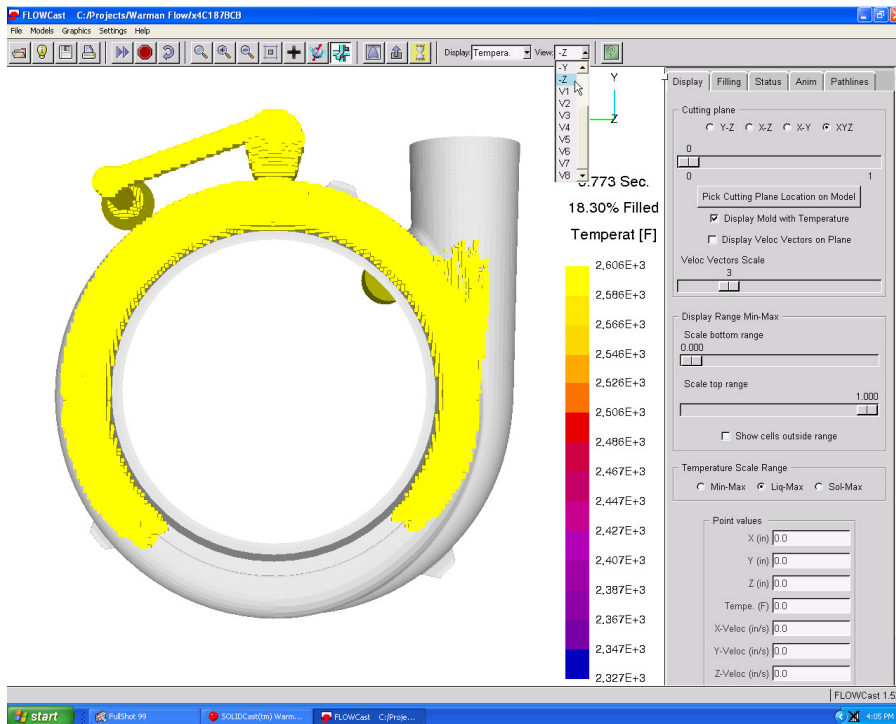


**-X View**

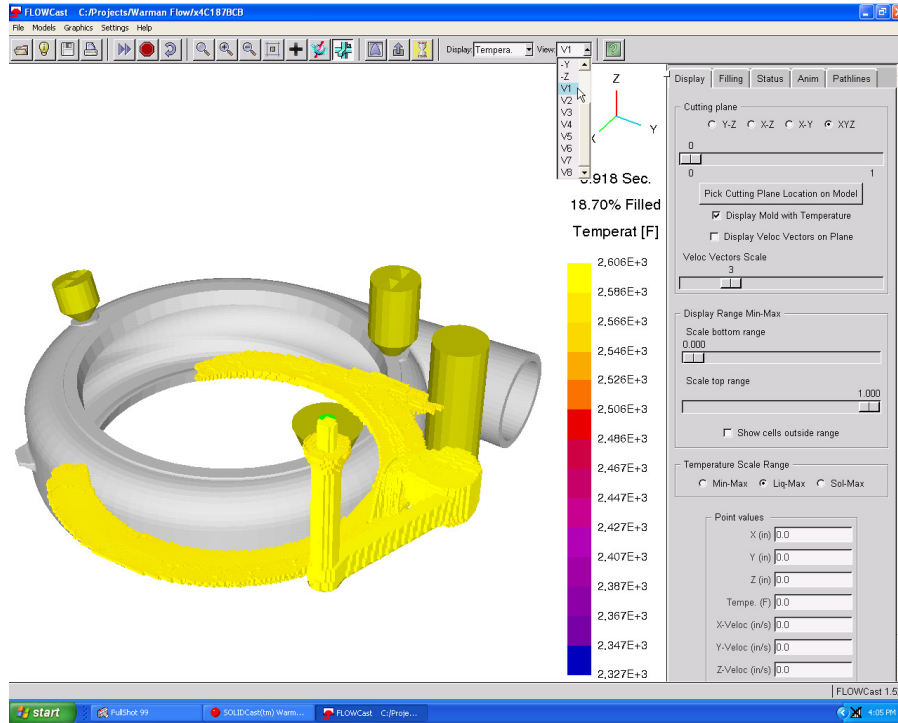




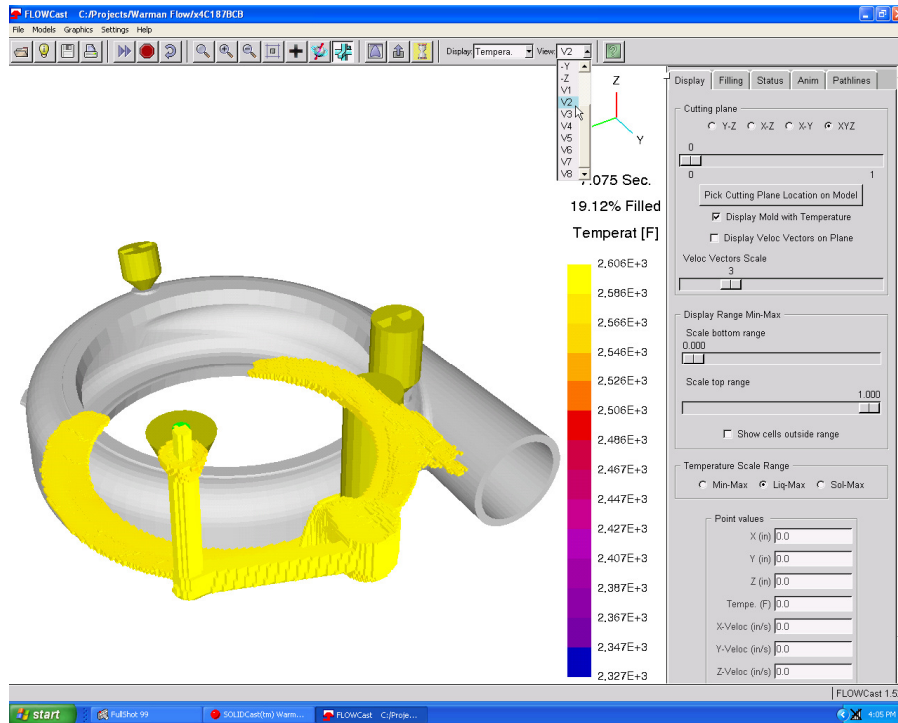
-Y View



-Z View

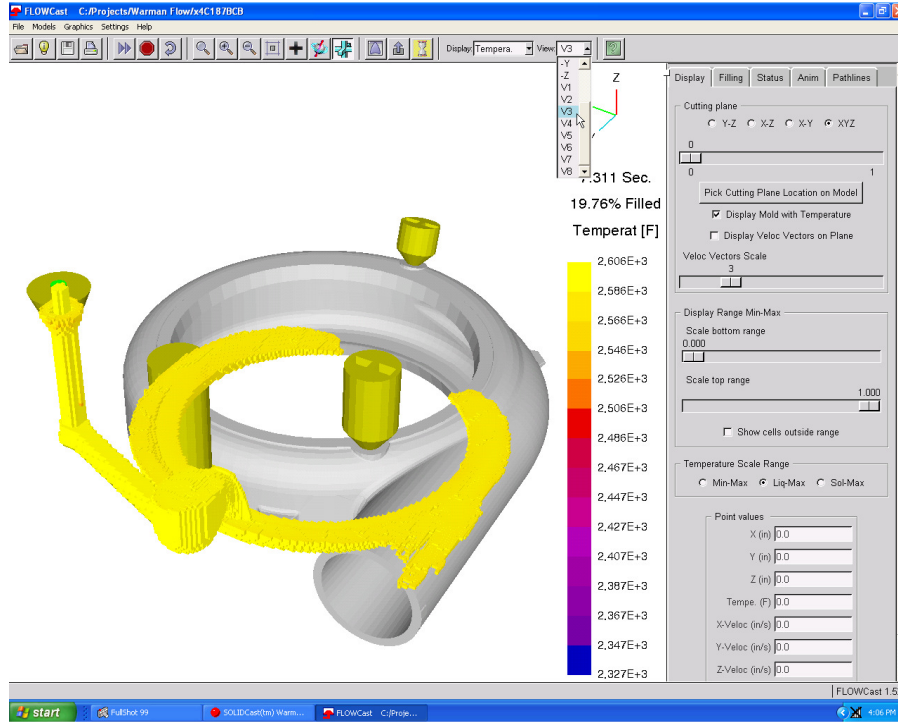


V1 View

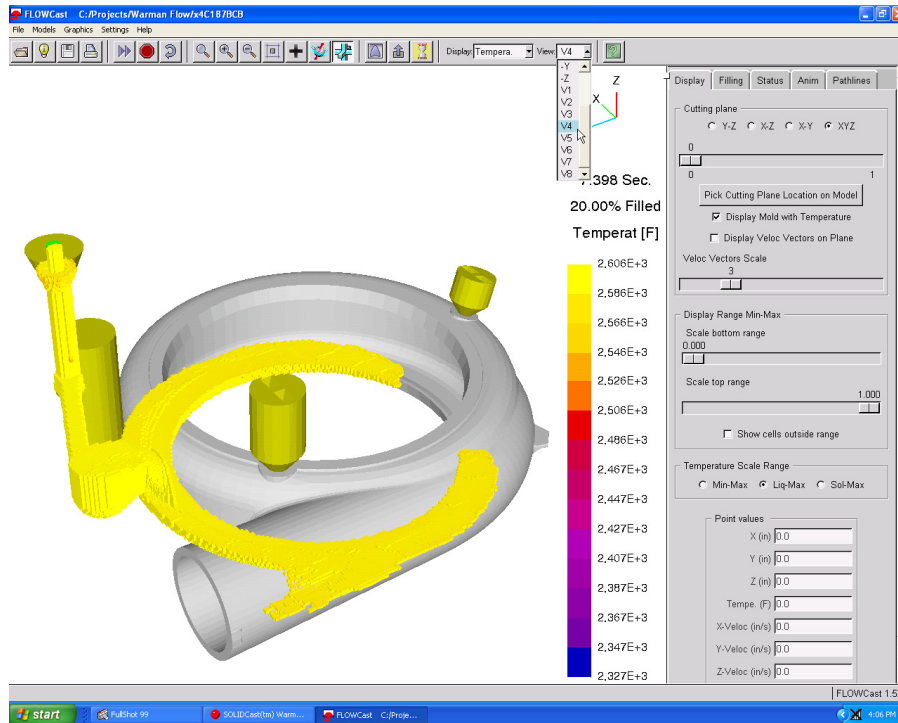


V2 View

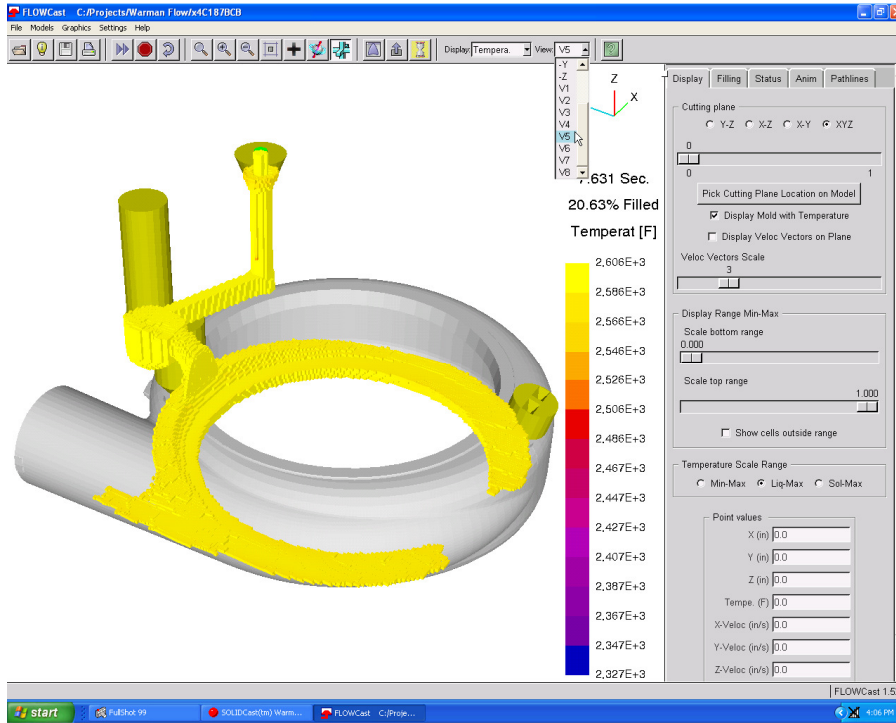




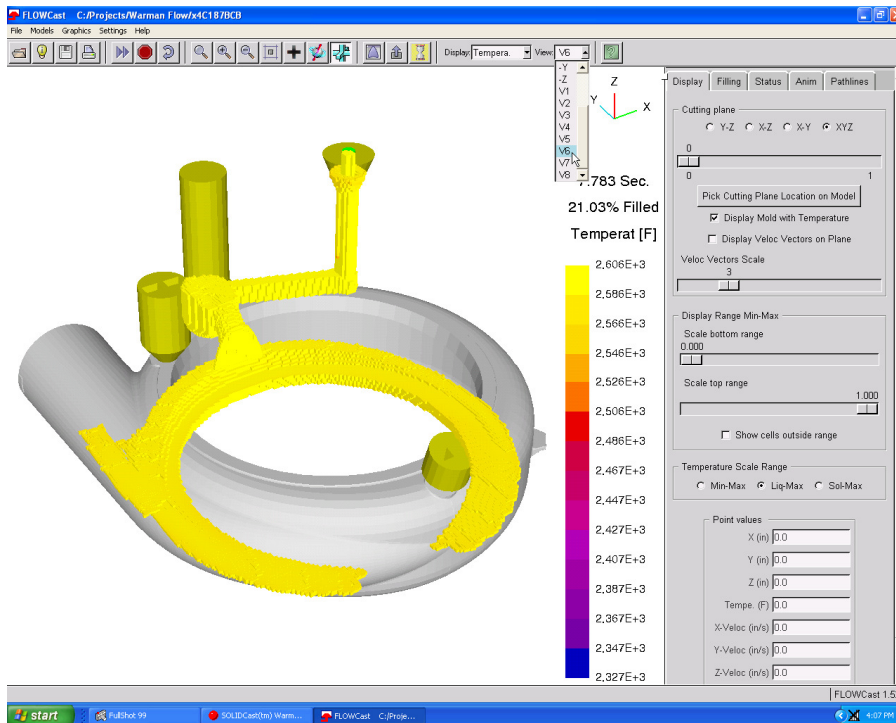
**V3 View**



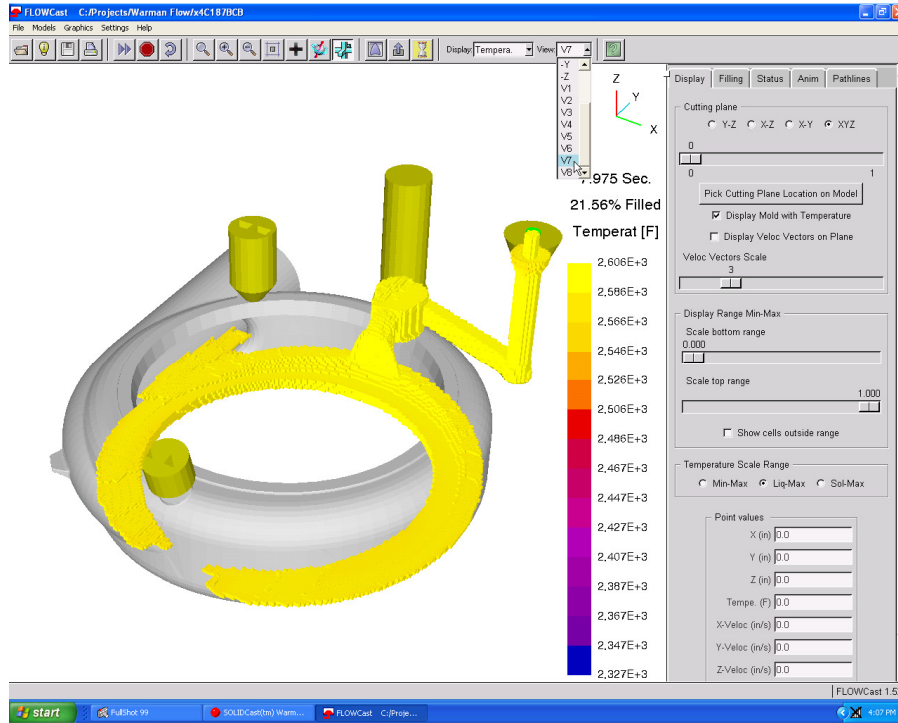
**V4 View**



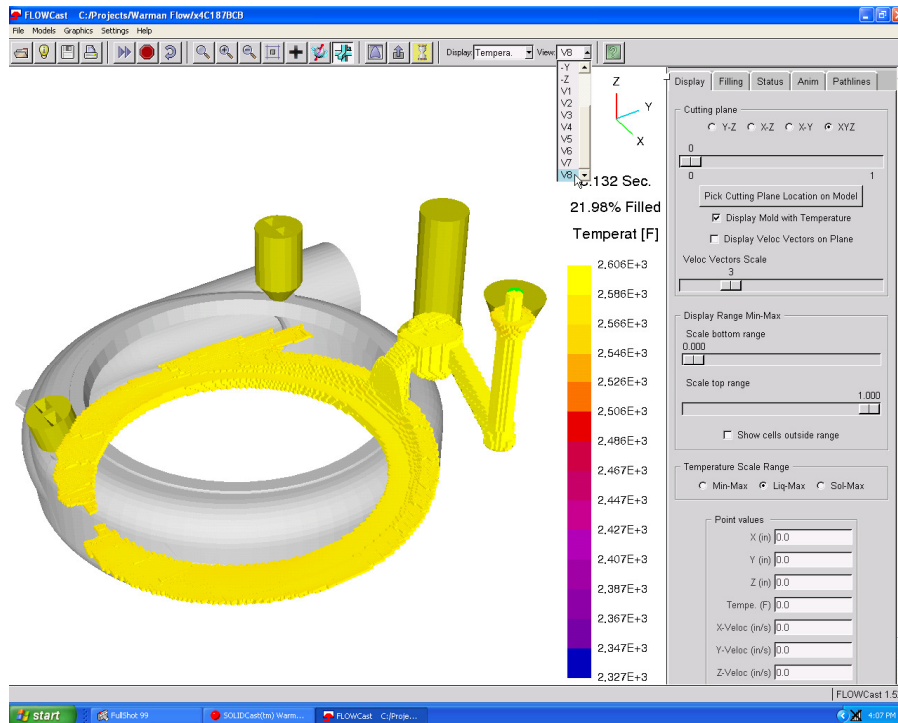
V5 View



V6 View



V7 View



V8 View



## UNIT 11: The Display Tab

The Display Tab gives you even more control of the visual screen during a simulation run. The Display Tab has the following options:

- Cutting plane**
- Display Range Min-Max**
- Temperature Scale Range**
- Point values**

These options are described in the rest of this unit.

### Cutting Plane

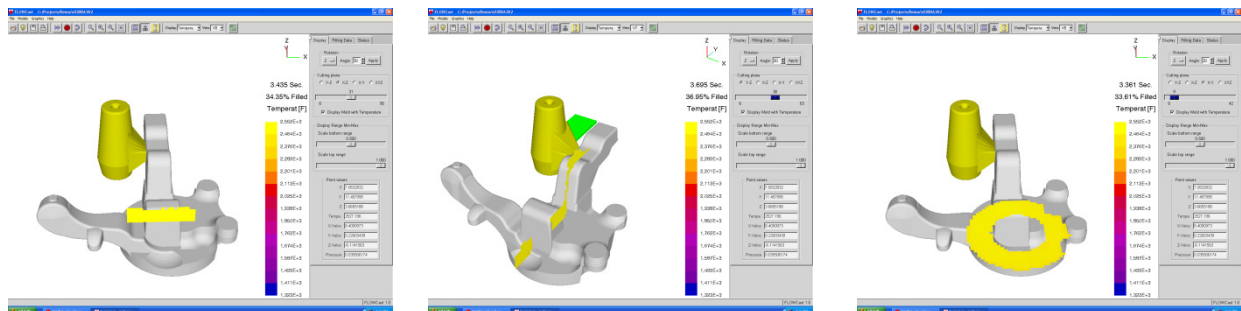
This function is used to create a 2D plot on a moveable plane cut through the casting model.

The default setting for this function is XYZ, which disables the 2D cut and displays the full 3D model. You can choose a cut plane in the Y-Z, X-Z or X-Y planes, which will display results at a constant X, Y or Z value, respectively.

Once you select a plane the slider bar becomes active, and you can use the mouse to drag the plane to any location in the model.

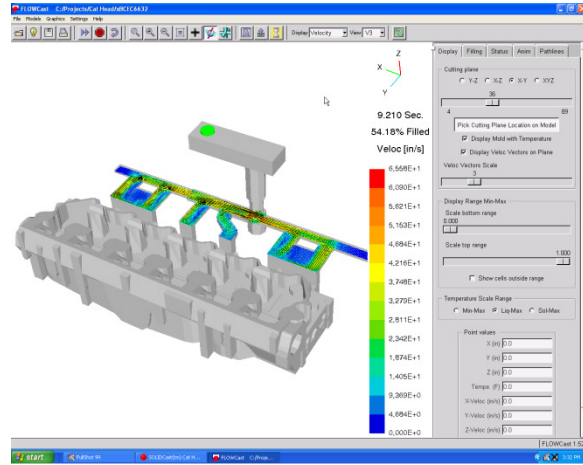
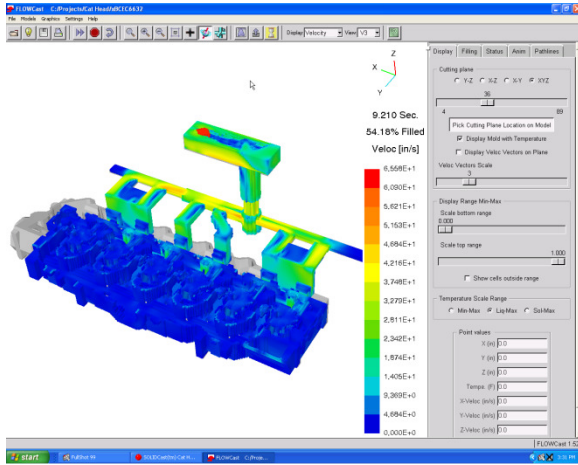
If you check the button labeled **Display Mold with Temperature**, the system will display the temperature of mold nodes as well as casting nodes. This selection also has the effect of extending the temperature scale to include the minimum mold temperature.

The following figures show the selection of Y-Z, X-Z and X-Y planes:



If you check the button labeled **Display Veloc Vectors on Plane**, then the system will display a set of velocity vectors on the plane, showing the relative speed and direction of the metal movement at that point. If Velocity Vectors are active, you will also have an active **Veloc Vectors Scale** slider bar. This bar varies from one to ten, and changes how long the vectors are on the cut plane.

The following figures show an XYZ display of metal velocity, and velocity vectors displayed on the X-Y plane:

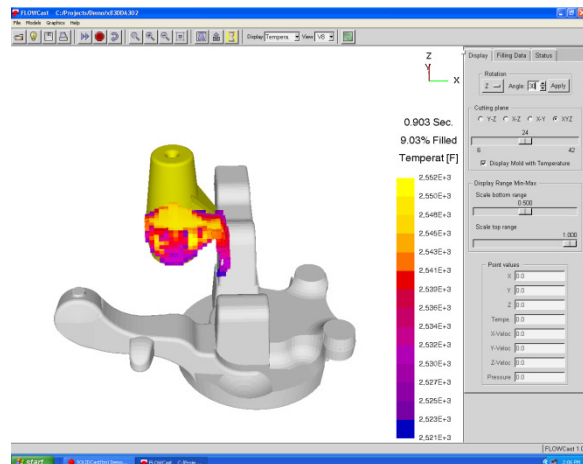
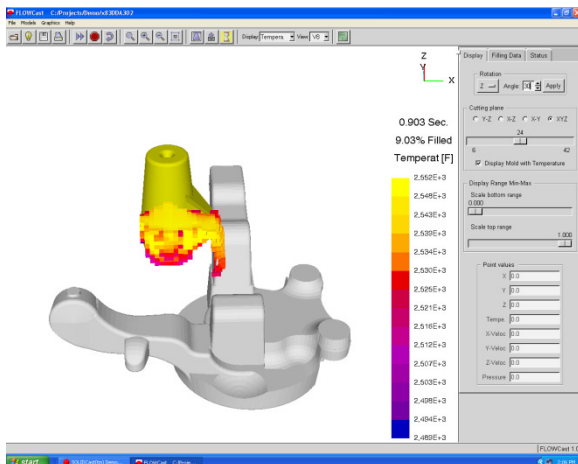


## Display Range Min-Max

When a simulation is running, the palette of plotting colors remains constant, but the range of data plotted may change, to show the full range of data at that point in the simulation. For example, if the display is showing temperature, as the casting cools during pouring, the range of temperatures will normally get larger.

You have the option of only showing a portion of the full range of data. This is adjusted by moving the Display Range Min-Max slider bars in the center of the Display Tab.

The example shown below shows the full range of temperature on the left, and only the upper half range on the right. Each shot was taken at the same time during the simulation.



**Temperature Scale Range**

When working with a temperature scale, many times it is more convenient to use a fixed scale range during the simulation or during movie creation. You have the following three options:

**Min-Max****Liq-Max****Sol-Max**

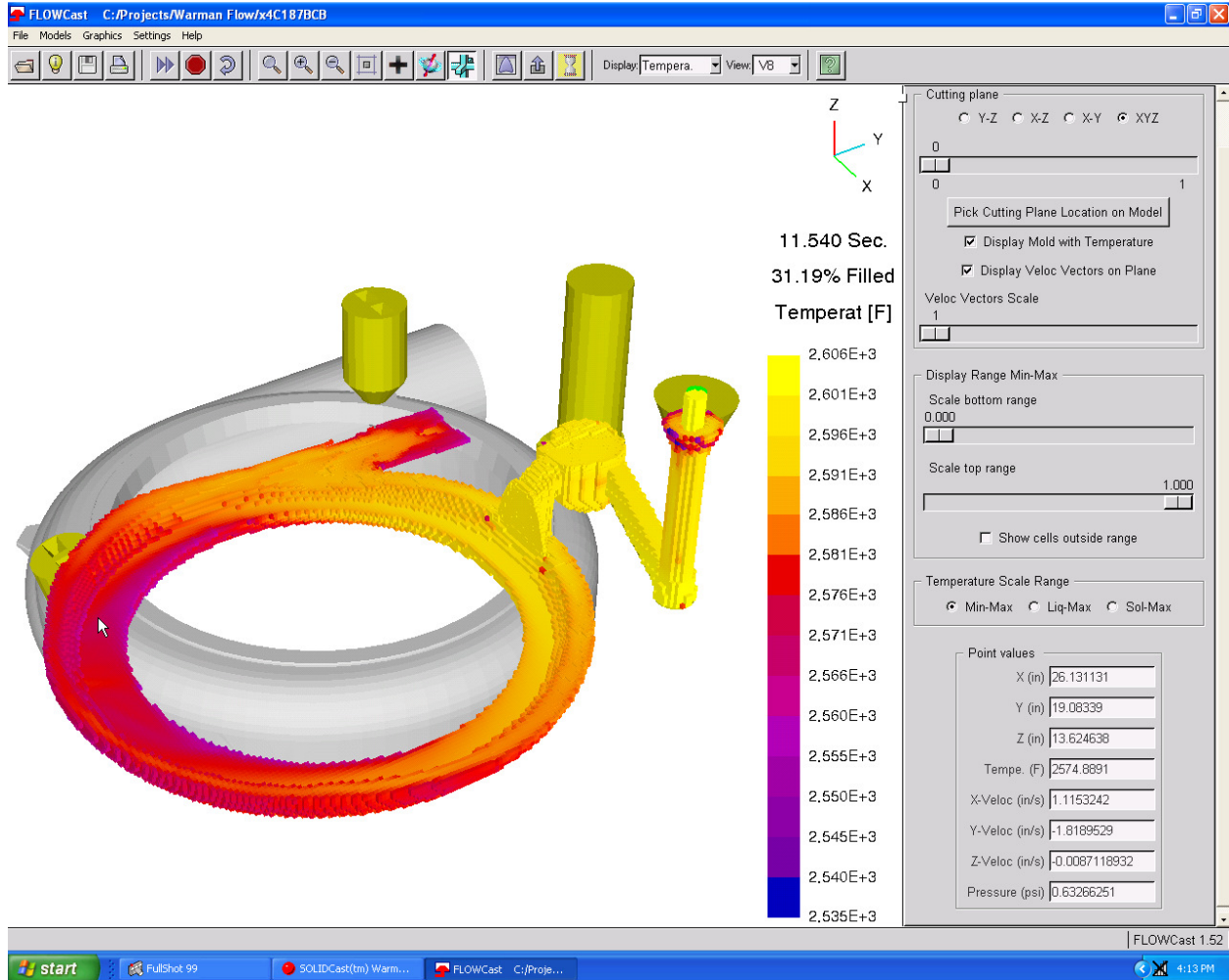
Min-Max is a variable display setup, where the values change as the range itself changes. Liq-Max is a fixed display, where the displayed range is from the Maximum temperature to the Liquidus temperature. This is the system default. Sol-Max is a fixed display, where the displayed range is from the Maximum temperature to the Solidus temperature.

By selecting a specific range, colors represent the same temperatures throughout the simulation/movie, and one screen can be directly compared with another.



**Point values**

The Point values function allows you to see all the data at a particular point in a model, simply by placing the cursor over a point in the model and clicking the left mouse button. The Point values window will then fill with the data for the current location, including X, Y, Z location, Temperature, X, Y and Z velocity components and Pressure. An example of this output is shown here:

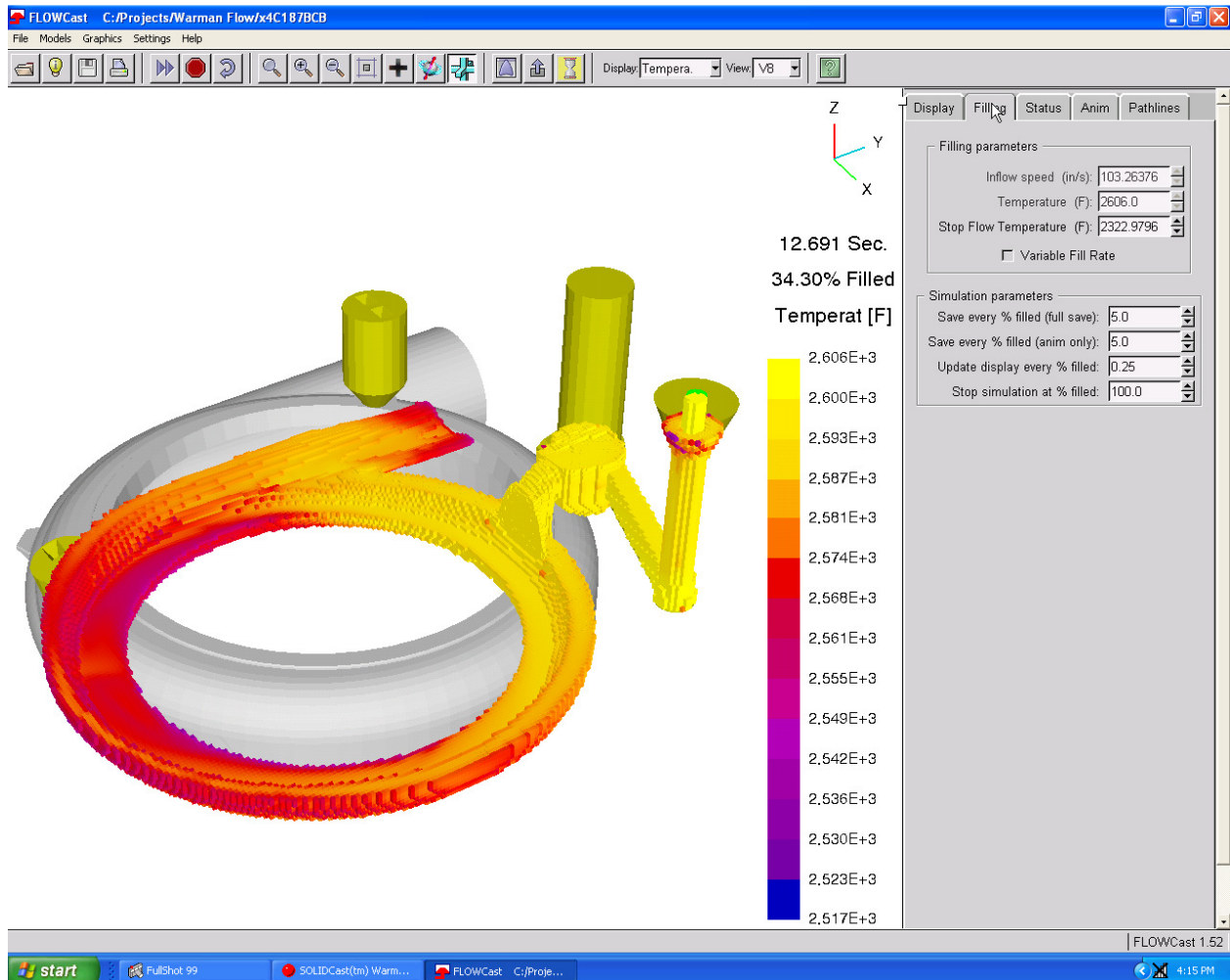


**Simulation data for a given location is shown via the Point values function.**



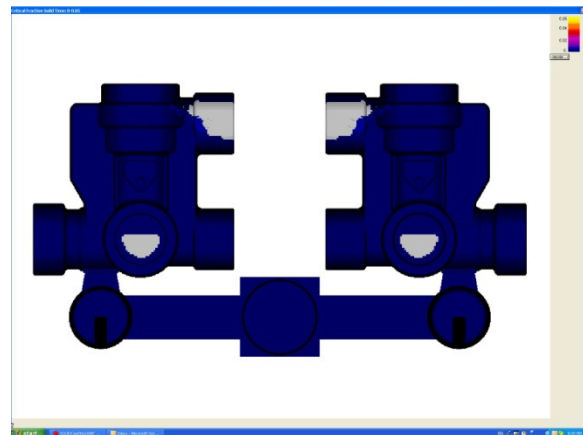
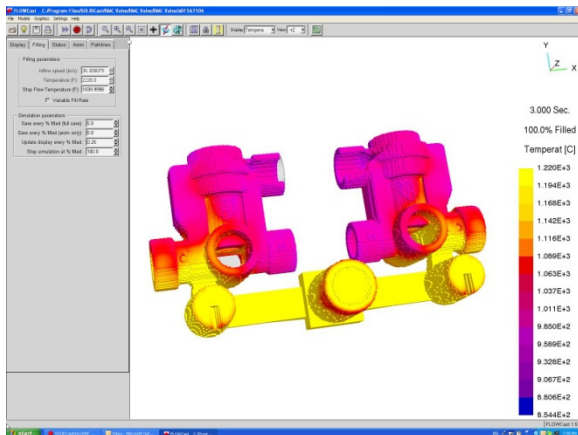
## UNIT 12: The Filling Tab

The Filling Tab contains two main areas, Filling Parameters and Simulation Parameters, as shown below:



Filling Parameters show the inflow speed, pouring temperature and stop flow temperature. The first two parameters are extracted from the meshed model, and are display-only. They cannot be modified by the user. The Stop Flow Temperature, Variable Fill Rate and Simulation Parameters can all be adjusted by the user.

**Stop Flow Temperature** – This is the point at which metal flow stops during a filling simulation. This will default to the Critical Fraction Solid (CFS) point specified in the SOLIDCast Materials List for the given casting alloy. However, it can be modified by the user, to simulate the effects of ‘castability’ and better predict misrun conditions. The following photograph and figures show a misrun condition in a copper-base casting, and two views of the FLOWCast and SOLIDCast predictions.



**Variable Fill Rate** – If this box is checked, then FLOWCast will run a variable fill rate simulation, using data provided from the SOLIDCast Bottom Pour Ladle Utility. This utility creates a file called FlowRates.fil, which contains data as to how the filling rate changes over time. It is also possible for a user to create this file manually, to simulate other types of variable filling rate situations.

**Save every % filled (full save)** – This tells the system how often to save data. The default value is every 5%. Please remember that saving data takes quite a bit of disk space. Normally, the main filling phenomena take place earlier in the filling sequence, so you may want to initially save data at, say, every 1% until 50% full, then reset to every 5%. This data is used after a simulation is complete to create animations of the filling process.

**Save every % filled (anim only)** – This tells the system how often to save temperature data only, for use in making movies on the Anim Tab. Since most people are interested in temperature data, you can specify a smaller number for temperature data, and a larger increment for the full save of velocity and pressure data. This may help to keep file sizes smaller, by not saving all the data in small increments.

**Update display every % filled** – This tells the system how often to redraw the screen during a simulation run. The default value is every 0.25%. As a simulation is run the screen is updated based on this parameter. If you are not actually watching the simulation, the value isn't really important. Since this data is not saved, the value used here will not affect the size of the saved simulation.

**Stop simulation at % filled** – This tells the system when a simulation is considered complete. Most of the interesting phenomena during filling take place fairly early on, so you may wish to stop a simulation before filling is complete. The default value is 100%.

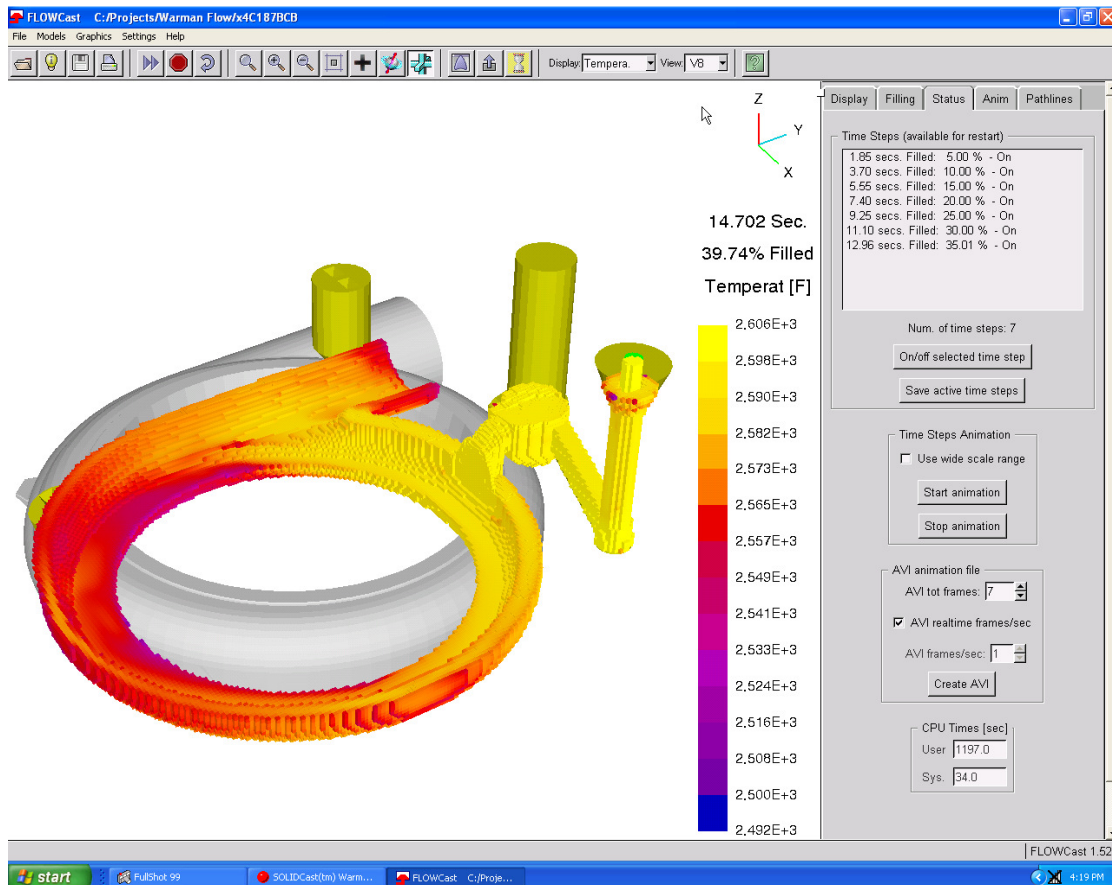


## UNIT 13: The Status Tab

The Status Tab keeps you informed as to how far the simulation has progressed, and allows you to create animation sequences from the saved data. The four sections of the Status Tab are:

- Time Steps (available for restart)**
- Time Steps Animation**
- AVI animation file**
- CPU Times (sec)**

The Status Tab looks like this:



### Time Steps (available for restart)

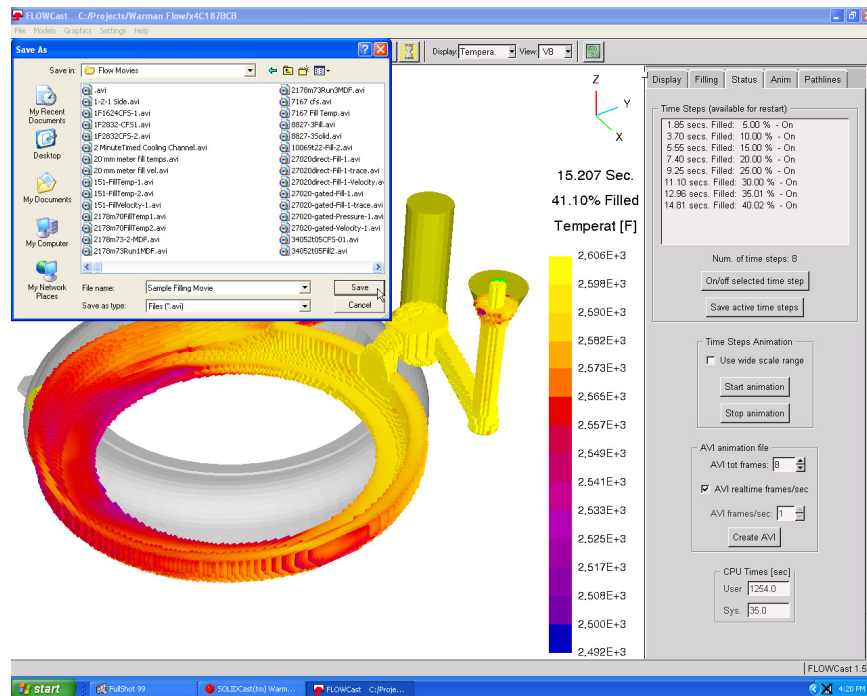
The Time Steps window lists the time intervals for which simulation data has been saved. For example, if you have instructed the system to save at 5% intervals, the system will list data at approximately 5% intervals as the simulation runs. The default status is On, or active, for use in plotting or animation. You have the options of turning on or off time steps that have been saved so far, and you can save active time steps out to a file at any time. This allows you to save a sub-set of time steps, if desired.

### Time Steps Animation

When a simulation is loaded, but not running, you can animate the available data into a running slide show. To start an animation just for viewing at the moment, click on the Start Animation button. The system will then progressively display the time step data, showing the progression of fill. The animation will show the currently selected data set, so you can choose from Temperature, Velocity, Pressure, etc. If you check the box labeled Use wide scale range, then the scale will stay constant for the full animation, using the maximum and minimum values found in the data.

### AVI animation file

If you want to save the animation as a movie, use the functions in the AVI animation file box. You can choose to create a 'realtime' AVI file, which will play back at the actual pouring speed, or you can specify a given display rate, in frames per second. Then click on the Create AVI button. You will be asked to name the movie, as shown here:



Once you type in a name and click on Save, the system will build the movie file and save it to disk. You can then copy this movie to a floppy disk or CD, or email it to a customer, who can run it simply by double-clicking on the file name.

### CPU Times (sec)

This function lets you know how long a simulation has taken and how much time the system is actually using for its' calculations. The value for User indicates how much time has elapsed since the simulation started (clock time) and the Sys value indicates how much time the CPU has been involved with simulation calculations, I/O, etc.

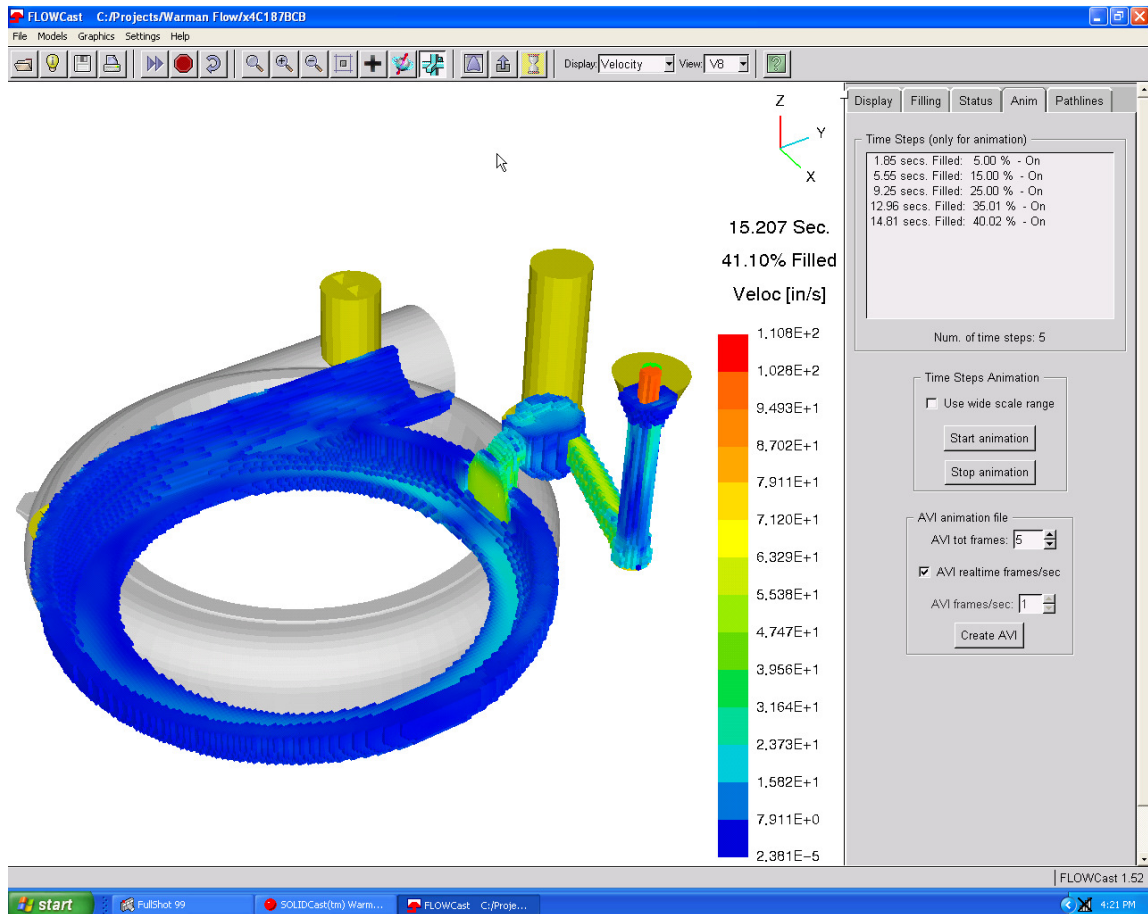


## UNIT 14: The Anim Tab

The Anim Tab keeps you informed as to how far the simulation has progressed, and allows you to create animation sequences from the saved data. The three sections of the Anim Tab are:

- Time Steps (only for animation)**
- Time Steps Animation**
- AVI animation file**

The Anim Tab looks like this:



### Time Steps (only for animation)

The Time Steps window lists the time intervals for which temperature data has been saved. For example, if you have instructed the system to save at 5% intervals, the system will list data at approximately 5% intervals as the simulation runs. The default status is On, or active, for use in plotting or animation. You have the options of turning on or off time steps that have been saved so far, and you can save active time steps out to a file at any time. This allows you to save a sub-set of time steps, if desired.

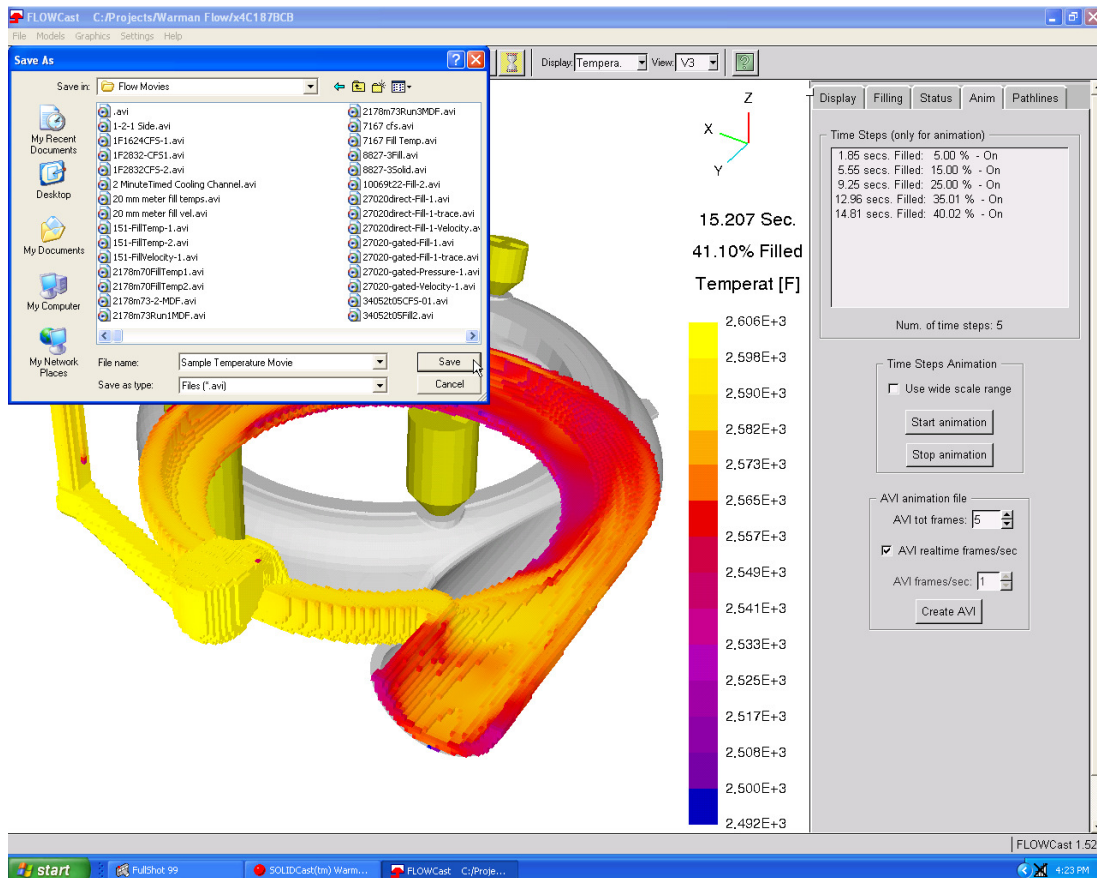
### Time Steps Animation

When a simulation is loaded, but not running, you can animate the available data into a running slide show. To start an animation just for viewing at the moment, click on the Start Animation button. The system will then progressively display the time step data, showing the progression of fill. When using the Anim Tab, only temperature information is available. If another data set is selected when you switch to the Anim Tab, when you start the animation, the system will automatically switch over to temperature.

If you check the box labeled Use wide scale range, then the scale will stay constant for the full animation, using the maximum and minimum values found in the data.

### AVI animation file

If you want to save the animation as a movie, use the functions in the AVI animation file box. You can choose to create a 'realtime' AVI file, which will play back at the actual pouring speed, or you can specify a given display rate, in frames per second. Then click on the Create AVI button. You will be asked to name the movie, as shown here:



Once you type in a name and click on Save, the system will build the movie file and save it to disk. You can then copy this movie to a floppy disk or CD, or email it to a customer, who can run it simply by double-clicking on the file name.

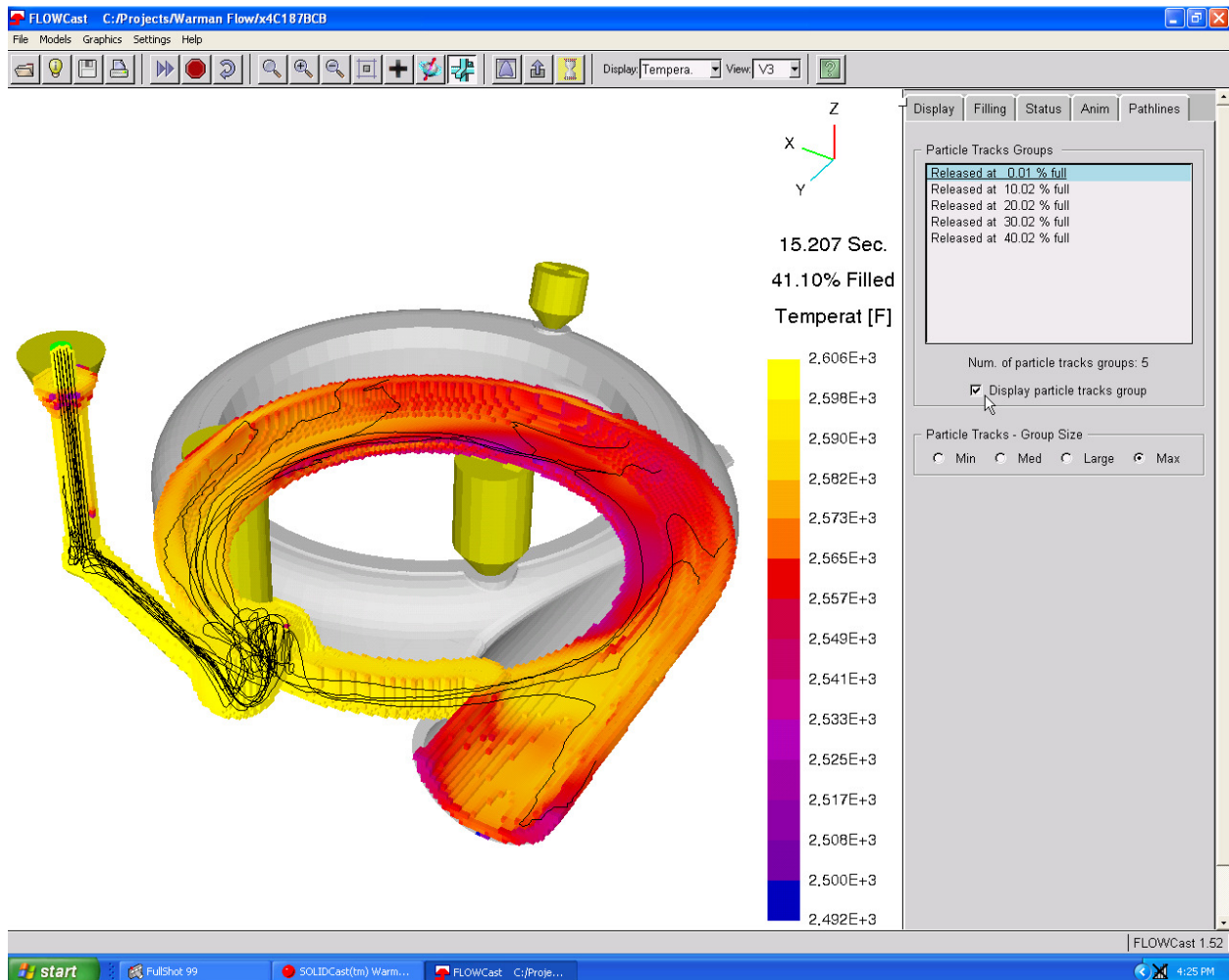


## UNIT 15: The Pathlines Tab

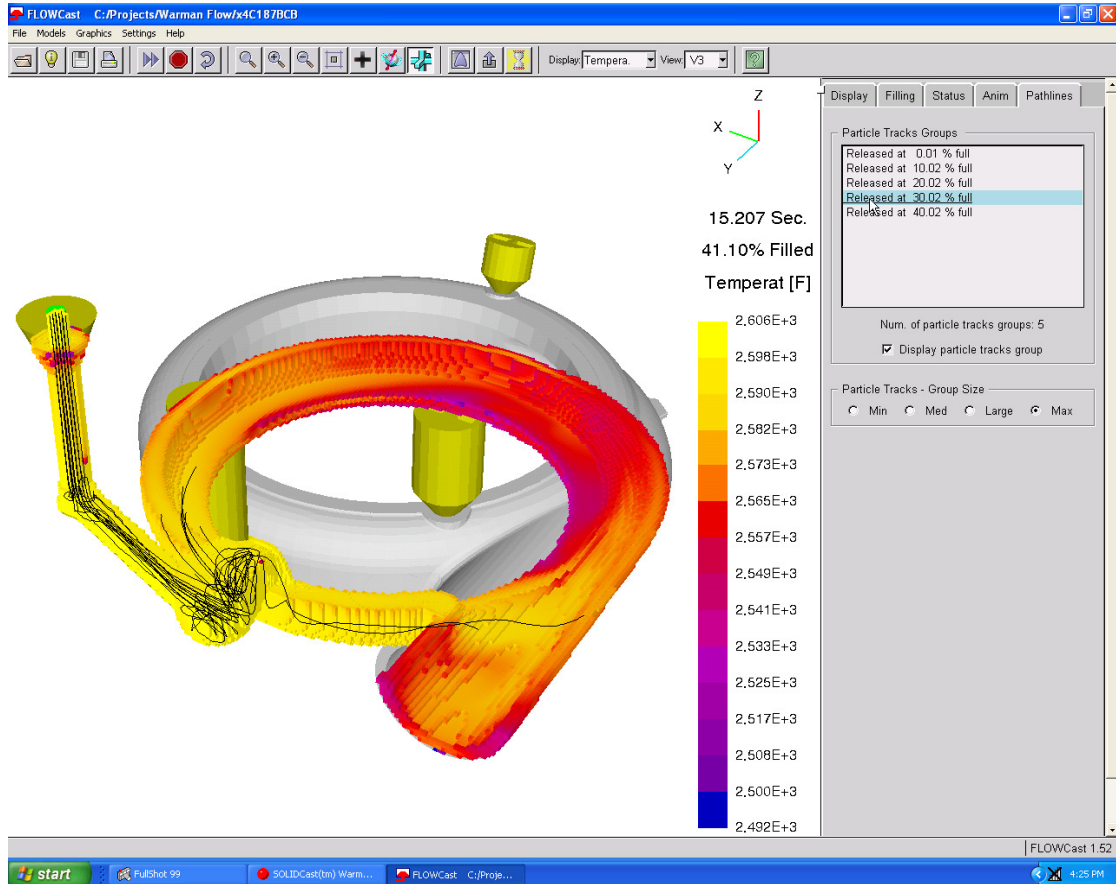
FLOWCast releases a group of particles from the fill material / casting interface cells at the start of the filling simulation, and then at regular intervals during the simulation. The frequency of particle groups release is approximately at 10% increments of casting filling.

Each one of the particles released from each fill material / casting interface cell is tracked in time while the filling simulation is executed. You can watch the particles move during a simulation, and also display the particle movement after a simulation is complete.

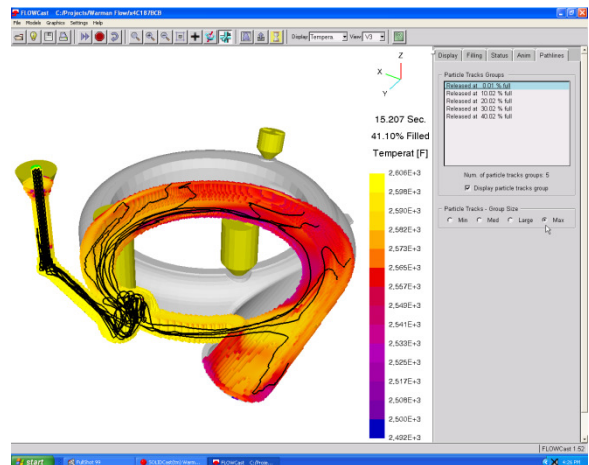
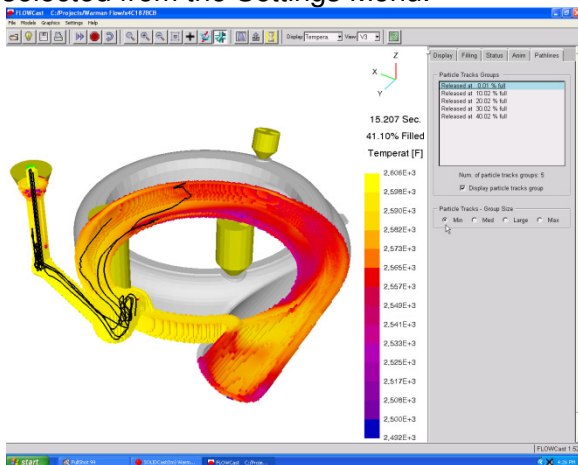
To display pathlines click on the Pathlines Tab and select the checkbox Display particle tracks group as shown here:



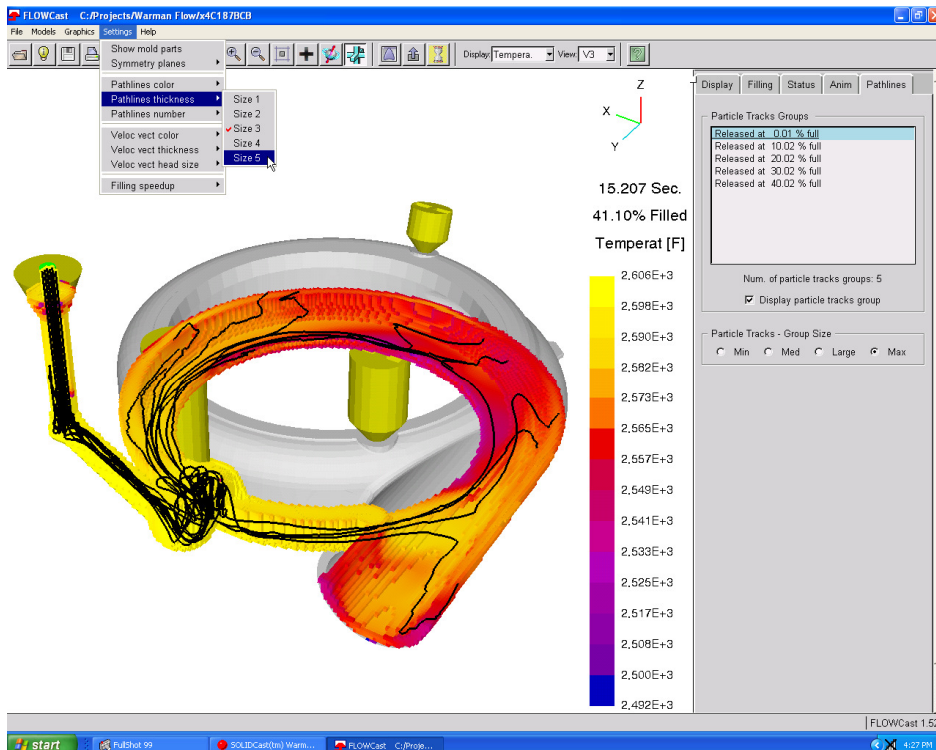
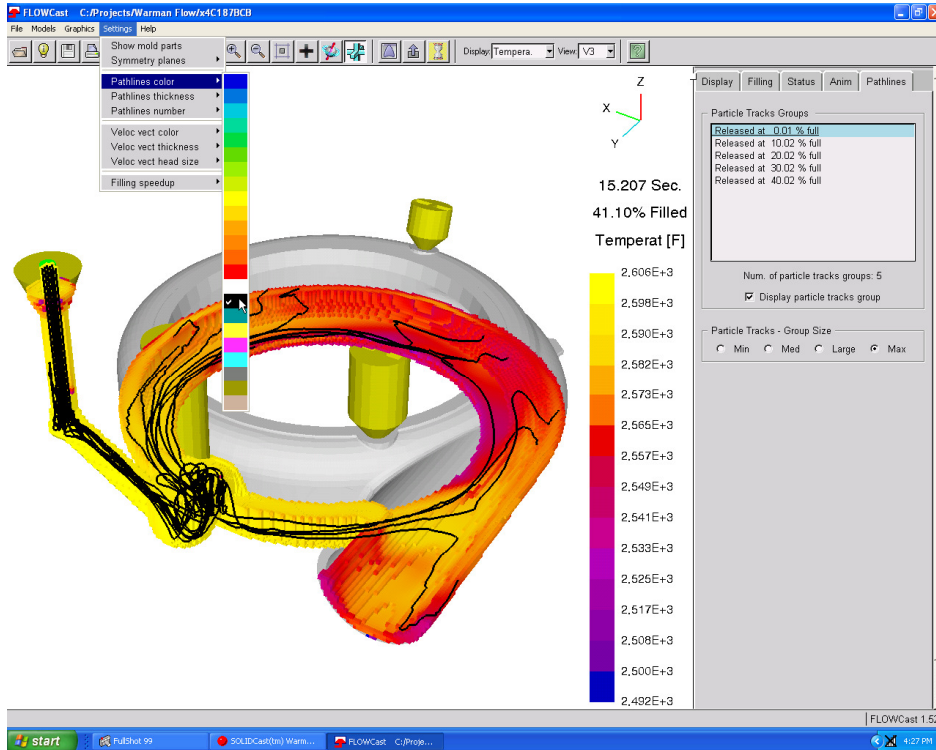
To display a particle group released at another instant simply select the group from the list as shown here:



All particles are tracked in time and their trajectories are available at any time. However, for inspection it may be convenient to use a different number of particles from any particular group. The number of particles whose trajectories are displayed can be selected as shown below. Available choices are Min, Med, Large and Max. The number of particles displayed can also be selected from the Settings Menu.



The color and thickness of the particle trajectories can be selected from the Settings Menu, as shown here:



## Notes:

- Any particle tracks group (loaded from the Pathlines Tab) can be displayed with any available Time Step (loaded from the Status Tab).
- Particle tracks are stored on disk, in the same way as Time Steps. Therefore, they are automatically reloaded when a simulated project is reloaded and they are automatically saved when a simulation is saved.
- If Particle Tracks Group display is selected and you create a Flow Movie, the Pathlines will be created and displayed as a part of the movie.

## UNIT 16: Flow Simulation Strategies

---

FLOWCast is a true fluid flow simulation package and may involve some strategies or methods somewhat different than those used in traditional solidification modeling.

First of all, the “full” flow simulation mode is very calculation-intensive. This means that it takes a lot of computer processing power and memory to be able to run. We recommend that you use the fastest computer processor and the most amount of RAM memory that your budget will allow. We also recommend that you use the largest node size, or the smallest number of nodes, that you can reasonably use for a given model. The amount of processing time increases more than proportionally as the number of nodes increases.

In many cases the “quick” mode of FLOWCast will allow you to do a good job of simulating the filling and progressive heat loss of the metal entering the mold cavity. Even though the quick mode is not a full CFD simulation, it still is very useful in approximating mold filling and the temperature distribution within the liquid metal.

In the full flow mode, FLOWCast has some internal limits (both high and low) on the entering velocity of the liquid metal. FLOWCast determines this velocity based on the Fill Time (in seconds) that you enter for the model. Knowing the volume of the mold cavity, FLOWCast calculates a flow rate from this fill time, and then calculates a velocity based on the area of the entering liquid. The area of the entering liquid is the contact area between the Fill Material and the Casting Material (or Riser Material). FLOWCast will inform you if the velocity is too high or too low. In each case, you can do the following:

1. If the velocity is too low, you can decrease the Fill Time or decrease the contact area of the Fill Material.
2. If the velocity is too high, you can increase the Fill Time or increase the contact area of the Fill Material.

FLOWCast can automatically adjust the Fill Time for you, or you can exit from FLOWCast, make a change and then re-mesh the model.

In future versions we intend to release fill time calculations based on casting size and shape.

If the RAM memory on your computer is limited, you may receive a message when you start FLOWCast saying that the simulation cannot be run for that model due to not having enough available memory. It is possible in this case that you can run a simulation in quick mode (click on the hourglass tool to change to quick mode) but you will not be able to run FLOWCast in full simulation mode unless more memory is made available. Sometimes this may mean shutting down other programs that are operating, or sometimes this may mean installing more RAM memory in the computer.

In using the SOLIDCast mold filling calculation, with complex gating systems (i.e., multiple gates from a common runner system) our recommendation was normally to make the entire gating system of Fill Material so that even distribution of flow from all the gates could occur. This is no longer necessary with FLOWCast (including both the quick and full algorithms). With FLOWCast, you can create the entire gating system using either Casting Material or Riser Material, and place the Fill Material at the point of entry of the molten metal (typically, at the top of the sprue). FLOWCast does a good job of apportioning flow through a gating system on a realistic basis.

For simulations with a high number of nodes, the output files created by FLOWCast can become very large, especially if small steps are specified for saving the results. For example, with a simulation of 1.6 million nodes and time steps set at 1%, we have found result data files up to 2 GB in size. This can be reduced by increasing the saving increment. In most simulations, what happens initially as the metal enters the casting is of interest, and you may want to capture this in small increments of say 1%. However, as the casting fills the incremental changes are not as great. It is possible to stop the FLOWCast simulation part way through and change the percent increment for saving data (say, from 1% to 5%) and then proceed with the rest of the simulation. When you animate this simulation or create an AVI file, you will see the initial steps at 1% and then the 5% increments afterward. This can significantly reduce the disk space needed for result data files.

You can delete the FLOWCast result files from within SOLIDCast by highlighting the mesh name on the project tree, then selecting File...Remove FLOWCast Data.... Or, you can delete the mesh and the associated FLOWCast data will be deleted as well. It is recommended that after generating all of your output reports, you either delete these files or archive them on other media to avoid filling up your hard drive.

If you are running tilt-pour permanent mold simulations, keep in mind that in FLOWCast only the full mode can be run for tilt-pour simulations. The options therefore are to run the full FLOWCast fill simulation or the traditional SOLIDCast filling simulation for the warm-up cycles. For maximum speed of the initial cycles, you can choose the SOLIDCast fill simulation for the warm-up cycles and then choose FLOWCast full simulation for the final cycle.

FLOWCast assumes that the mold is well vented, i.e., that the pressure on the advancing liquid is always atmospheric. Later versions will incorporate consideration of gas pressure and vacuum conditions.