

FLOW-3D Cast FAQ

Version 2009-02-05

Table of Contents

Calculation only uses one CPU.....	1
Gridlines in preprocessing and postprocessing are different.	1
Does the FLOW-3D Cast calculate graphite expansion?.....	1
What type of neutral file format does FLOW-3D Cast support?.....	1
How does density evaluation option [ifrho] work?.....	2
How do I define heat transfer between mould and a separate part of mould.	3
Can I see the metal temperature and the mould temperature at the same time	3
How do I see only the mould temperature.....	3
What is the difference between rapid and dynamic shrinkage model?.....	3
If moving obstacle include cooling or heating channels. Are pointer locations moved with the moving obstacle?	4
Heat transfer between mould and gas was not applied.	4
How do I model filters?.....	4
How can I model cooling channels?.....	5
How can I model more than 3 cooling channels?.....	7
What are the steps for doing restart simulation (solidification after filling).....	7
What is the save format for history data.....	8

Calculation only uses one CPU.

The calculation engine uses only one CPU even when my FLOW-3D license allows me to use several CPU's. If I run the simulation with the FLOW-3D user interface it uses all CPU's.

Set the system wide environment variable **OMP_NUM_THREADS** to the number how may concurrent threads your FLOW-3D license allows.

The problem happens because the FLOW-3D calculation engine requires user interface to specify the maximum number of allowed threads by setting the **OMP_NUM_THREADS** variable. The FLOW-3D Cast user interface tries to determine how many threads the FLOW-3D your calculation engine license allows but in some cases this can fail and then it sets the limit to 1. One can override the number of calculation threads by specifying this environment variable.

Gridlines in preprocessing and postprocessing are different.

The cell borders do not seem to be aligned in the post processing but in preprocessing they are. Why does this happen?

- 1) The gridlines in FLOW-3D Cast preprocessing and postprocessing are different. In preprocessing they show the edge lines of the calculation elements. In the postprocessing they show the centerlines of the calculation elements
- 2) The gridlines inside the preprocessing layers can move around a bit. The FLOW-3D tweaks the gridlines a bit in order to avoid quick changes in the mesh accuracy that can cause problems in calculation convergence. The border gridlines of the preprocessing layers are – however – not moved. If you have a gridline that you want to stay exactly at given location you should define it as an edge of a layer.

Does the FLOW-3D Cast calculate graphite expansion?

The expansion of the metal material in the mushy zone is not currently supported by FLOW-3D Cast.

What type of neutral file format does FLOW-3D Cast supports?

The format of the point cloud file is following:

```
-->clip
Mat. No.: 1 2 3 9
-1
0
-0.141890E+02 -0.131465E+02 -0.799620E+00
0.804380E+00 0.441660E+00 -0.799620E+00
0.119996E+01 0.441660E+00 -0.799620E+00

-->end clip
```

First line is a header line. There can be several of these. In reading points for neutral file FLOW-3D Cast ignores these.

Second line is the end mark of header information (must contain value -1). Header end mark is mandatory.

Third line contains number of data values for each point after the point x,y,z coordinates. In this example it's 0 (=no data, this point cloud file is essentially a neutral data file itself (except that some neutral files contain the data point end mark which is a line containing just '-1'. Such end mark is not allowed here [which I consider as a bug and have recorded it as such]).

4th line is the minimum coordinate (ignored by FLOW-3D Cast)

5th line and lines from there forward contain the point coordinates in which data is requested.

Once the data points are imported into FLOW-3D Cast the 'Write neutral file' menu is available.

It write the following kind of file:

```
-->clip

time= 2.85631329E-01
neutral file output
-1
2 tn tw
-1.41890001E+01 -1.31459999E+01 -8.00000012E-01
8.04380000E-01 4.41659987E-01 -7.99619973E-01 -1.00000004E+35 6.73149963E+02
1.19995999E+00 4.41659987E-01 -7.99619973E-01 -1.00000004E+35 6.73150024E+02
1.59554005E+00 4.41659987E-01 -7.99619973E-01 -1.00000004E+35 6.73149841E+02
1.99111998E+00 4.41659987E-01 -7.99619973E-01 -1.00000004E+35 6.73149963E+02
2.38668990E+00 4.41659987E-01 -7.99619973E-01 -1.00000004E+35 6.73149963E+02
2.78226995E+00 4.41659987E-01 -7.99619973E-01 -1.00000004E+35 6.73150024E+02
3.17785001E+00 4.41659987E-01 -7.99619973E-01 -1.00000004E+35 6.73150024E+02
3.57343006E+00 4.41659987E-01 -7.99619973E-01 -1.00000004E+35 6.73150085E+02

-->end clip
```

Again, first we see header information ending at single line containing -1. Then there is the number of variables followed by variable names. Next line contains the minimum coordinate information (it is the point having smallest x,y and z coordinates found in any of the data points of the exported data).

Then the data lines follow, 3 first values for each line are x,y,z coordinate of the data point - followed by data values for variables in that point.

How does density evaluation option [ifrho] work?

Constant density [ifrho=0]:

When cell is fully liquid the constant value [rhof] is used as density.

When cell is solidifying a linear interpolation is done between [rhofs] and [rhof] based on cell solid fraction

When cell is fully solidified the constant value [rhofs] is used as density.

Density based on temperature [ifrho=1]

When cell is fully solidified $\text{density} = \text{rhofs} [1 - \text{thexs}1 (t - t11)]$

When cell is solidifying a linear interpolation is done between [rhofs] and [rhof] based on cell solid fraction

When cell is fully liquid $\text{density} = \text{rhof} [1 - \text{thexf}1 (t - t11)]$

How do I define heat transfer between mould and a separate part of mould.

You need to specify the other part of the mould as a separate obstacle. Instead of selecting the interpretation 'Mould' to the geometry you need to type an alternative interpretation (such as 'Mould2'). Then you can specify heat transfer coefficient between the two moulds in the Heat Transfer Coefficients dialog.

Can I see the metal temperature and the mould temperature at the same time

I'm afraid you can't. They are generated by the calculation engine as separate variables.

How do I see only the mould temperature

Select scalar variable as 'wall temperature' , select isosurface as 'volume fraction' and select 'complement' for the surface

What is the difference between rapid and dynamic shrinkage model?

Porosity in solidifying metal occurs as a result of gas evolution as well as volumetric changes (ie. shrinkage) during phase transformation. There are two models in FLOW-3D that can be employed to predict macro porosity formation in metals due to shrinkage. The first model involves the solution of the full system of hydrodynamic equations. As a result, the evolution of velocity and pressure in the solidifying metal can be predicted. Therefore, this model is called Hydrodynamic, or First Principles (FP), shrinkage model.

The continuity and energy equations Equations (3.5) and (3.19), are modified to include volumetric source terms. These are defined by the solidification rate and solid/liquid density difference. The basis of the model is that material contracts when it solidifies, and this induces some tension in the material in the form of lowered pressure that attempts to pull surrounding liquid in to replace the volume lost to shrinkage. If the surrounding liquid is confined and cannot flow, the pressure will continue to decrease with further shrinkage until the critical pressure is reached. At that point a shrinkage cavity is allowed to open. The critical pressure can be related to the typical pressure at which dissolved gases come out of solution.

Despite being an accurate tool to study the porosity formation phenomena, the FP model may be computationally costly because at each time step the numerical algorithm involves the complete solution of momentum and energy equations. The size of the time step, controlled by various stability criteria associated with fluid flow, may also be small compared to the total solidification time of the casting. The latter may be as long as hours for large sand castings.

The other, simplified shrinkage model is based on the solution of the metal and mold energy equations only. No fluid flow equations are solved by the model. Porosity is predicted by evaluation the volume of the solidification shrinkage in each isolated liquid region in the casting at each time step. This volume is then subtracted from the top of the liquid region in accordance with the amount of liquid metal available in the cells from which liquid is removed. The "top" of liquid region is defined by the direction of gravity. The relevance of this approach is supported by the fact that in many situations fluid flow in the solidifying metal can be ignored. Porosity formation in that case is primarily governed by metal cooling and gravity. Feeding due to gravity often occurs on a time scale much smaller than the total solidification time. This model is called Rapid Solidification Shrinkage (RSS) model. The RSS model in FLOW-3D provides a simple tool to perform quick simulations of shrinkage in complicated castings.

Neither model includes the evolution of gases, therefore, porosity volume is defined by the cooling rate and solid/liquid density difference. The most important aspect of shrinkage modeling in FLOW-3D is that shrinkage cavities are explicitly described, therefore, its influence on fluid and heat flow is also taken into consideration. No volumetric expansion, e.g. due to remelting, is included in these models.

If moving obstacle include cooling or heating channels. Are pointer locations moved with the moving obstacle?

No.

You can – however - import a channel geometry, specify its interpretation as 'Input', and make it movable and specify a moving schedule for it. Then in Source Parameters specify it as heat source and specify surface temperature for the object. Then specify heat transfer coefficient to the surrounding mould from drop down menu 'Flow'->'Heat transfer coefficients..

Heat transfer between mould and gas was not applied.

You have to specify the void type for the gas around the mould. This is done by adding a pointer in the gas region using drop down menu 'Flow->Special Parameters->Region pointers'.

Alternatively you can specify the gas region using STL and giving in creating a new gas region in the geometry interpretation and assigning the STL to that.

Then open gas parameters dialog and check the 'heat transfer type [ivht] and set type to 1 (this sets the surrounding gas void type to 1). In the 'Flow->Heat transfer coefficients' you have to define the heat transfer coefficient between mould and Void region type 1.

How do I model filters?

Filters are best modeled using baffles (Flow => Special Parameters => Baffles).

Parameters are

- Baffle coordinates
- Porosity
- Linear flow loss coefficients
- Quadratic flow loss coefficients

Porosity of baffle is between 0.0 (for no porosity) and 1.0, for completely open baffle. Flow losses in porous baffles can be linear or quadratic, or both, with respect to velocity. The linear losses are specified using Linear Loss Coefficient, and quadratic flow losses are defined using the Quadratic Coefficient. The latter has no dimension, while the former has the dimension of velocity.

Filter functionality is available in Extended and Advanced version of FLOW-3D Cast.

How can I model cooling channels?

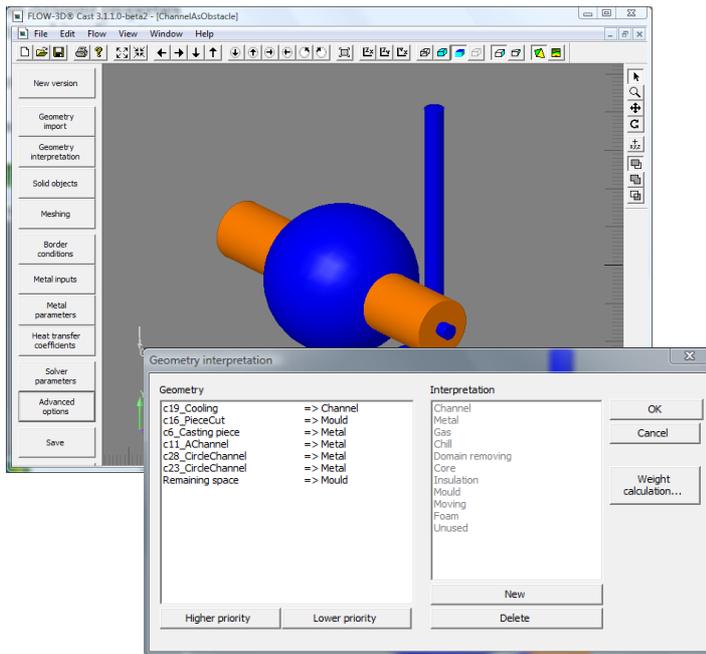
There are two methods for modeling cooling channels

- 1) By using obstacles with constant surface temperature
 - Supports time depended temperatures
 - Easy to model
 - Channel is not visualized as hollow in postprocessing

Define casting piece, feeding channels and the mould normally.

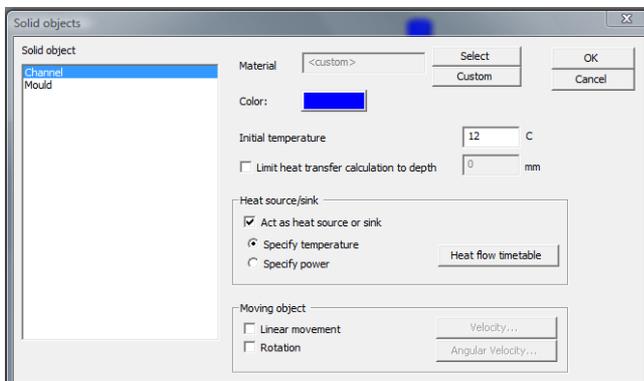
Import cooling channel geometry as STL

In geometry interpretation type in a custom text such as 'Channel'

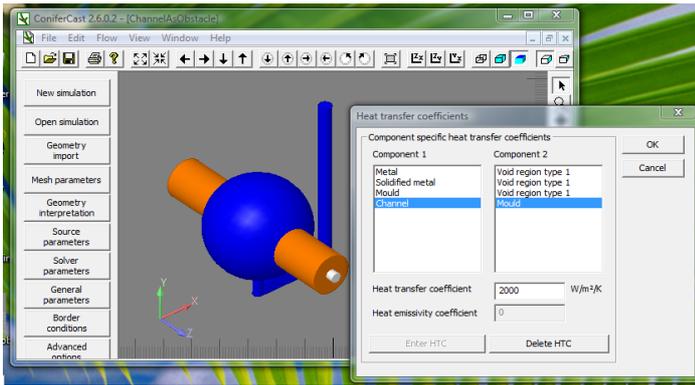


In solid objects dialog

- Give custom material to the Channel, give density x specific heat as zero, give high value for thermal conductivity
- Define the Channel as heat source/sink with surface temperature specified
- Using heat flow timetable button define channel temperature as a function of time



In Flow->Heat transfer coefficients define HTC between the mould and the cooling



- 2) By using gas (void) regions
 - Supports emissivity
 - Can only define 3 channels with different properties

Define casting piece, feeding channels and the mould normally.

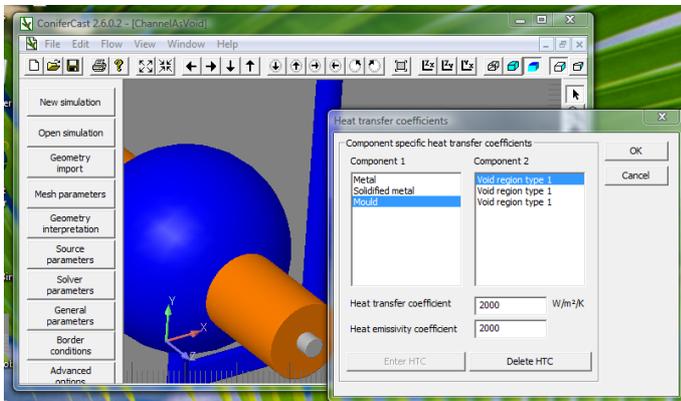
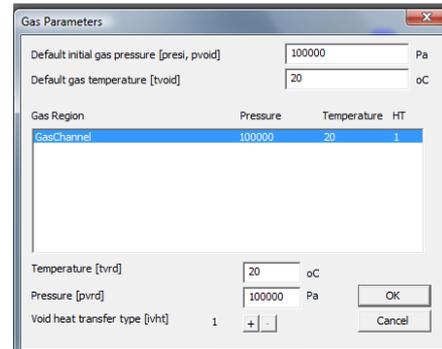
Import cooling channel geometry as STL

In geometry interpretation use 'New' button to create a new gas region and assign that to the geometry

In Flow->Gas Regions set properties for the cooling channel.

- Check Heat Transfer [ivht] flag, leave type as 1
- Set temperature [tvrd] as the channel temperature

In Flow->Heat transfer coefficients define heat transfer coefficient and emissivity coefficient between mould and void 1



How can I model more than 3 cooling channels?

As I know, I can use 3 cooling channels. If I use void1 to define ambient, then, I can use only 2 cooling channel. In case of LPDC, many cooling channels are required. How can I model this problem?

You can define multiple physical cooling channels to be of type void1 as long as their temperature and heat transfer properties are same. Different void types are required when the characteristics of the channels are different (different fluid in the channel, different temperature of fluid in channel etc).

Alternatively defining obstacles as heat sources with a low surface temperature could be used.

What are the steps for doing restart simulation (solidification after filling)

- 1) Run the filling simulation
- 2) Select 'Make restart simulation' from the Flow menu, a new document will open
- 3) If you used obstacles as metal sources, in source parameters clear any timetables for metal inputs (i.e. no more metal coming in as it's filled already and we do not calculate flow anymore)
- 4) From 'Solver parameters' dialog, disable flow calculation (ihonly=0)
- 5) Enable heat transfer model in fluid and obstacles (ihtc=2) unless - of course - it already is enabled (if you calculated this in filling)
- 6) Set shrinkage model to Rapid Shrinkage model (ishrnk=2)
- 7) Make sure the time limit (twfin) for simulation is big enough (solidification takes longer than filling)
- 8) In advanced options set Density to Function of local temperature (ifrho=1)
- 9) Click Save on the left toolbar
- 10) Click Calculate on the left toolbar

The log shows " warning - volume adjustment more than 5 percent of total volume, try using iavrck=2". What does this mean?

A cell with a large open face area and a small volume could restrict the time step to small values if there is significant flow in this cell. Should this happen, and it can be determined by monitoring the mesh locations controlling the time step as printed in the output file, it may be necessary to modify the mesh/obstacle arrangement. An algorithm to automatically adjust these ratios by making small "errors" in all volumes is available. A negative value of AVRCK no larger than -2.0 is recommended (e.g. AVRCK=-5.0 or -3.1).

Variable AVRCK:

AVRCK>0, mesh cells with area/volume ratios exceeding this value are listed in pre-processor output.
AVRCK<0, a negative value flags pre-processor to adjust cell volume fractions so ratio does not exceed magnitude of AVRCK. (This option allows larger fluid convective time steps for certain situations.)

Variable IAVRCK:

FAVOR volume adjustment for stability with larger time-step limit (see AVRCK)

=0, no adjustment

=1, maximum adjustment for maximum time-step limit

=2, smaller adjustment for improved time-step limit

So the calculation does small adjustments to volume-fraction of the cells in order to improve calculation stability. If these adjustments get too large the system warns that the accuracy of the calculation can suffer and suggest using IAVRCK=2 which would improve the accuracy but would also make the calculation times longer.

What is the save format for history data

Data is inputed in plain text format ie. they are *.txt files. Format is simple. Time values followed by value for that given time: For example:

```
time1 value1  
time2 value2  
time3 value3  
time4 value4  
..
```