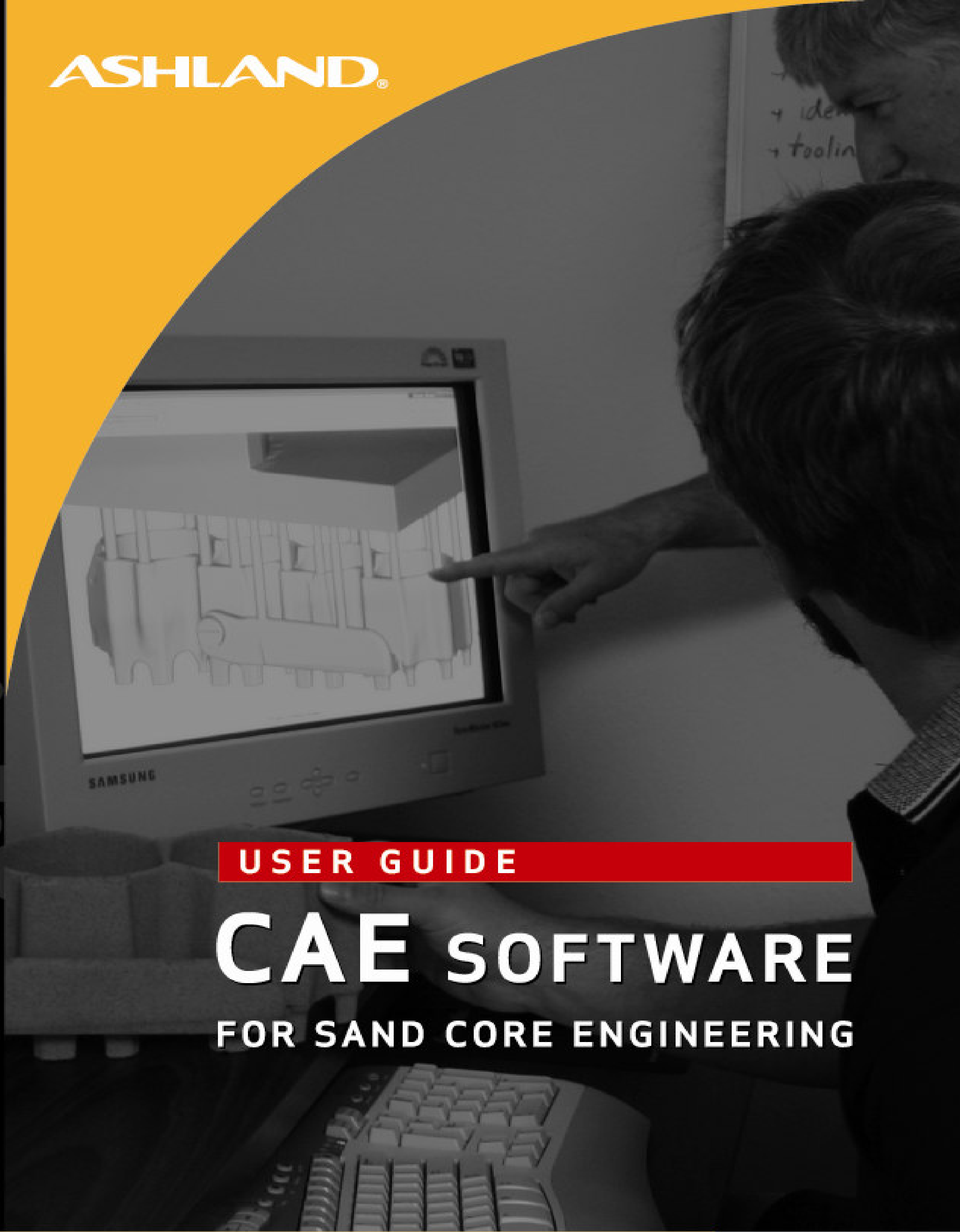
ASHLAND®

Arena-flow®



USER GUIDE

CAE SOFTWARE

FOR SAND CORE ENGINEERING

Arena-flow, LLC

CPFD™

Arena-flow, LLC
10899 Montgomery Blvd. NE, Suite B
Albuquerque, NM 87111
Phone 505-275-3849
Fax 505-275-3346
info@arena-flow.com

Contents

1 Sand Core Modeling

1.1 Sand Core Engineering

In this Chapter you will learn about:

- *Process modeling in the foundry industry*
- *The importance of sand cores and molds*
- *The cost of poor sand core design*
- *The benefits of proper sand core engineering*

It has been suggested that the foundry industry is nearly 6000 years old[1]. Mankind has been making metal castings for a long time, without the aid of computers. With a few simple ingredients such as ore, fire and a clay riverbank, castings have been taking shape for millennia.

Today the overall process has changed little; the melt is more sophisticated containing precise alloys and is degassed to reduce some defects, the fire has been usurped by modern induction furnaces and the riverbank has been replaced a variety of permanent or non-permanent molding processes. Regardless, to make a casting, one must liquefy metal, move the liquid metal into a mold, and remove the casting following solidification.

While the process itself has changed little, the tools of the trade have changed a great deal. Process modeling has been introduced through the 1980s and 1990s and is now common place in many of today's foundries. Consider an automotive component, for example – before the foundry order is placed, the component has already undergone a great deal of analysis in virtual space. It has likely been subjected to a structural analysis using Finite-Element Analysis (FEA) software. Further, if the component is an engine block, a thermal analysis has likely also been used as part of the design process. This thermal analysis was also likely coupled with a fluid flow analysis of the cooling passages using a Computational Fluid Dynamics (CFD) software package. Once the order is received by the foundry, and often before, the component is subjected to both metal flow and solidification analyses.

Note: Process modeling is common practice in today's foundries. FEA, CFD, thermal, metal flow, solidification – everything is modeled except the cores themselves.

While it is reasonable to expect varying levels of accuracy from the various analysis methods mentioned above, process modeling overall has been found to be useful. Keep in mind that computational models can only be as good as the input data and they do not replace good, old-fashioned thinking, but are simply tools to be used by sensible foundry engineers to enable good design decisions. When used correctly, process modeling has resulted in improved product quality, reduced scrap, faster time to market, reduced frustration from trial and error, innovative designs, lighter, stronger and more efficient products, enhanced process understanding, environmental benefits, lower total costs and even improved customer relations.

> **Note:** Core defects can result directly in casting defects!

Until the early 21st century, however, process modeling was not used for the design and manufacture of commercial sand cores and molds. This is not because the cores and molds are unimportant; core defects can result directly in casting defects. Severe problems result in misshapen cores. This can be caused by an incomplete fill of the tooling, an incomplete hardening of the sand during the curing process, breakage of the core or simply core deformation during the metal fill process. Other problems include metal penetration due to core density, other gas-related defects and difficulty in removing the sand during shake-out of the finished casting.

The cost of these core-related defects can be staggering. The best-case scenario involves scrap cores. If scrap cores can be detected before the metal is poured the sand may be reclaimed with a simple loss of time, efficiency and consumables.

Inevitably some defective cores will go undetected and will be formed into a core package, resulting in defective castings. Once defective castings are created, the best-case scenario involves scrapping the castings. In addition to the cost of scrapping the defective cores, all cores in the package with their related time and consumables are now lost. Further, although the metal may be melted once again, there is a large cost of energy required to do so. Scrap castings represent a significantly larger cost than scrap cores.

However, the cost of not scrapping the castings is much higher still. Once defective castings are permitted to be released, costs can be measured in the millions of dollars. For the automotive industry this is typically seen in recalls. Even higher still can be the cost of a "poor quality image" in the eyes of consumers.

The preceding paragraphs were not intended to paint a grimmer than realistic picture of the modern casting industry. In fact, many cores, molds, castings and products are being manufactured defect-free today. However, even these can benefit from process modeling. By computationally verifying the efficiency of the design one can minimize cycle time costs, equipment costs, consumable costs, maintenance costs, staffing costs and likely other costs as well.

In summary, simulation of the coremaking process results in higher quality cores which in turn results in the production of higher quality castings. The overall process can be made to run faster and more efficiently at a lower total cost.

> **Note:** Simulation of the coremaking process closes the virtual design loop from end to end!

Additionally, simulation of the coremaking process closes the virtual design loop from end to end, for some processes. Once again, let us consider our automotive engine block example. Before contacting the foundry, the casting designer has probably already created a Computer-Aided Design (CAD) model of the block and used Computer-Aided Engineering (CAE) analysis on the structural, thermal and fluid flow performance of the final component. *It is then assumed that the core package can be constructed efficiently* and simulation is once again resumed by analyzing the metal flow and solidification of the casting. Next, machining analysis is performed and the virtual design process continues.

What if one was not required to assume that "*the core package could be constructed efficiently*"? What if during the sand core engineering process modeling, it was determined that a minor change in design would make a component that was much easier to manufacture? What effect does this have on the performance of the casting, and ultimately the engine? By simulating the coremaking process, the design loop can be closed electronically, permitting the up-front analysis of the entire manufacture process from performance, to core making to casting and beyond.

1.2 CPFD™ Technology

In this chapter you will learn about:

- *Why there has been little sand core process modeling to date*
- *An evaluation of different approaches to sand core modeling*
- *The CPFD™ methodology – enabling technology for sand core engineering*
- *The history and applicability of the CPFD™ technology*

If the quality of sand cores and molds has such a great influence on the final casting quality, why has only a little modeling been performed before the early 21st century? Several answers could be given. Some would say that the final product is the casting, not the cores. True enough, however it is the core package that is actually constructed by the foundry, not the casting directly. Once the cores are produced and assembled, the metal is simply poured in through risers and gates to make the casting. Some would also cite previous poor experiences with modeling as a reason to avoid sand core simulation. However, by considering the validation in the following chapter and the aforementioned importance of proper core design, this is little more than a poor excuse at best.

One very real reason for not modeling the process is that no validated tool has been commercially available, previously. True. How would one construct such a tool? Let us consider various methods and discuss the merits of each.

What needs to be modeled? When blowing or shooting sand cores, a high pressure air supply is connected to a sand magazine and the air-sand mixture flows into the core tooling. The sand is constrained to remain within the tooling while the air exhausts through numerous discrete vents.

There are several approaches which may be used to model the filling of cores and molds with sand. All start with the basic conservation laws such as the conservation of mass and momentum. The formulation of these equations can be classified broadly as continuous or discrete[2].

The challenge in accurately modeling the filling of sand cores and molds is that the motion of both air and sand is important to the filling behavior. Aerodynamic drag often initiates the sand motion; however as the sand densely packs a region the air must be displaced and flows through vents as well as ejector pin and parting line clearances. Since neither the air nor the sand can be neglected, neither a purely continuous nor a purely discrete modeling approach would be able to accurately capture the multiphase physics. A single-phase, continuous approach would either neglect the sand entirely, or model the air-sand mixture as a heavy liquid. A single-phase, discrete approach would neglect the air filed entirely.[3]

Clearly a multiphase approach is required and several have been attempted with little success[4,5]. The most commonly attempted multiphase method for modeling sand core filling is referred to as the “Eulerian – Eulerian” or “continuous – continuous” Computational Fluid Dynamics (CFD) approach. This means that there two distinct phases in the model, both utilizing a continuous, Eulerian formulation of the equation sets resulting in separate fields for the air and sand. Both phases are effectively treated as liquids with different densities and viscosities.

The largest problem with the continuous, Eulerian – Eulerian approach is the underlying assumption that sand is a fluid. Put simply, sand is not a fluid. A granular material such as sand will not deform under any applied stress as would a fluid. Granular materials support their weight through inter-particle contact forces, not a hydrostatic pressure gradient as a fluid does. A granular material is quantized in nature and cannot be infinitely subdivided as a fluid. Additionally, sand has a size distribution. In spite of these shortcomings, continuous, Eulerian – Eulerian methods are still being attempted as extensions to existing metal flow and solidification software packages[3].

Note: Since CFD could not accurately model sand core filling, a new approach was required – CPFD™.

Note: CPFD™ technology models the air as continuous and the particles as discrete – it's two solvers in one!

Since CFD could not accurately model sand core filling a new approach was required – CPFD™[6] or Computational Particle Fluid Dynamics. CPFD™ uses an "Eulerian – Lagrangian" or "continuous – discrete" formulation for computing multiphase sand core blowing. The air is continuous in an Eulerian frame of reference while the sand is discrete in a Lagrangian frame of reference. In a way, CPFD™ can be thought of as two solvers in one – a CFD solver for the air and a particle solver for the sand. More information on the numerics behind CPFD™ can be found in the literature[2,7,8,9,10].

To date, three commercial software packages have been built upon the revolutionary CPFD™ technology: ***Arena-flow***®[11] for sand core engineering, ***Seefoam***™[12] for lost foam pattern filling and ***Barracuda***™[13] for applications in the chemical process and power generation industries.

1.3 CPFD™ Validation

In this chapter you will learn about:

- ***Arena-flow***® *validation against fundamental experiments*
- ***Arena-flow***® *validation against data from test cores*
- ***Arena-flow***® *validation for production cores and molds*
- *Other CPFD™ validation*

On a fundamental level CPFD™ sounds like it has the right approach, but does it work? Validation for ***Arena-flow***® can broadly be classified into three categories: fundamental experiments, test cores and production cores. The purpose here is not to provide an exhaustive list of all validation performed. Such a list would not only be ever-changing, but also subject to various disclosure and secrecy agreements. Rather, the purpose here is to provide a brief overview of some of the published validation to date, to provide a point of reverence for further validation work and a point of confidence for commercial application of the software.

Often software vendors publish examples of their technology for the most complex cases imaginable. A useful modeling method must be able to solve problems with real-world geometry and complexity; however such cases rarely qualify as validation examples. It is easier to claim success for an unknown, complex result than to validate against fundamental physical phenomena. In the latter case, the true answer is known or the result can be verified, thus there is little room for error in the judgment of the reader. For the complex case however, any reasonable result can seem plausible.

Arena-flow® has been validated for very complex cores and blowing machines, however our discussion of validation will start with fundamentals. It is our purpose here to show the fundamental, granular nature of sand motion and validate the CPFD™ methodology against the fundamental data.

Our first example involves sand falling in a familiar hour glass or "sand clock". An hour glass has been used as a timekeeping device for centuries. What is so special about sand, rather than water, as the timekeeping media? The answer is simple – sand flow through an orifice is constant; not dependent on the height of the sand above. Liquid flow through the same geometry is dominated by the ever-changing hydrostatic pressure head above the orifice.

Figure 1 shows an ***Arena-flow***® result for sand flowing in an hour glass[3]. Sand is initially densely packed in the upper region. The sand falls under gravity and the rate of fall is constant, regardless of the depth of the sand pile in the upper region. The flow of sand under gravity may be applicable to sand draining from storage hopper or filling magazines. It is important to note that the flow of a liquid through the same geometry would be fundamentally different.

> **Note:** Sand flows through an orifice at a constant rate – fluid does not. The CPFD^{TM} method correctly predicts this phenomenon.

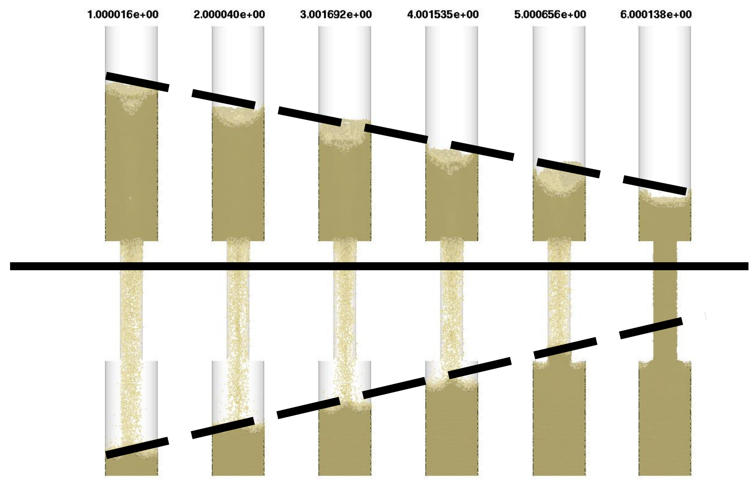


Figure 1: Linear sand flow through an hour glass

> **Note:** Granular materials can stack at non-horizontal surface angles – fluids cannot. The CPFD^{TM} method correctly predicts this phenomenon.

Figure 2: Granular nature of sand surface angles

This geometrically simple test case demonstrates the correct granular nature in the CPFD^{TM} method as opposed to the fluid nature of CFD. Figure 2 shows the sand surface angles produced if the hour glass were poured at a 45 degree angle[3]. Once again, the granular nature of the sand is clearly evident as it is permitted to support its weight at non-horizontal angles. A fluid could not do this.

Another example of a fundamental granular flow experiment is a "U-Tube". This is also a geometrically simple experiment. Sand falls under gravity around a U-shaped bend constructed from a square-section, clear polycarbonate tube with internal dimensions of 22mm x 22mm. A measured amount of sand was permitted to drop from one side of the tube and high-speed video data was collected as the sand stacked up around the bottom[3].

Note: ***Arena-flow***® correctly calculates the falling of sand in a U-Tube using the CPFD™ method.

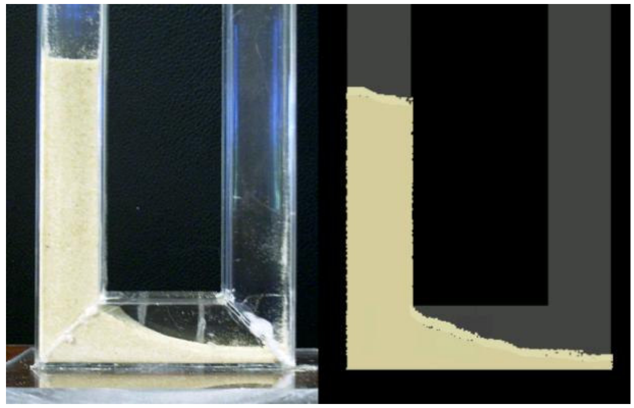


Figure 3: Sand shape after falling under gravity in a U-Tube

Figure 3 shows the experimental result at left and the CPFD™ computational result at right. Agreement is excellent. Most of the sand stacks up in the vertical arm, however some sand is permitted to flow to the right both in the experiment and in the calculation. The steady-state solution for a fluid under these circumstances is fundamentally different, with an equal fluid height in each of the arms of the tube.

Arena-flow® correctly predicts fundamental granular flow problems using the CPFD™ technology, but what about high-pressure sand core blowing? A test core known at the "Ashland T-Tool" was constructed. A single blow slot permits sand and air flow from a magazine and air exits through numerous vents along the bottom and in the side arms. It was constructed in a manner which permits the various vents to be easily opened or closed.

Note: ***Arena-flow***® correctly computes the filling of the Ashland T-Tool compared to high-speed video data.

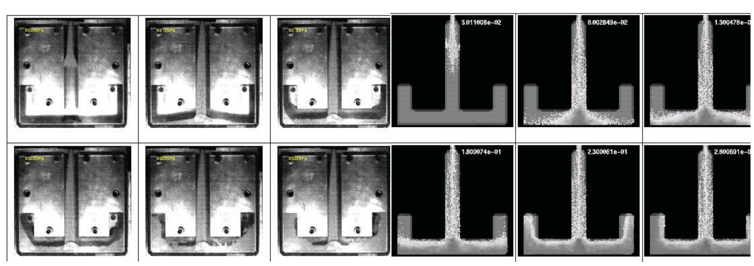


Figure 4: Ashland T-Tool filling (video left, calculation right)

Figure 4 compares the ***Arena-flow***® results on the right with the high-speed video data to the left. ***Arena-flow***® correctly predicts both the speed and shape of the sand flow in the tooling. Of note, the sand stacking at the bottom of the tooling, and also the outside-to-inside filling of the outer arms are captured very well.

Note: ***Arena-flow®*** correctly computes the curing of the Ashland T-Tool compared to high-speed video data.

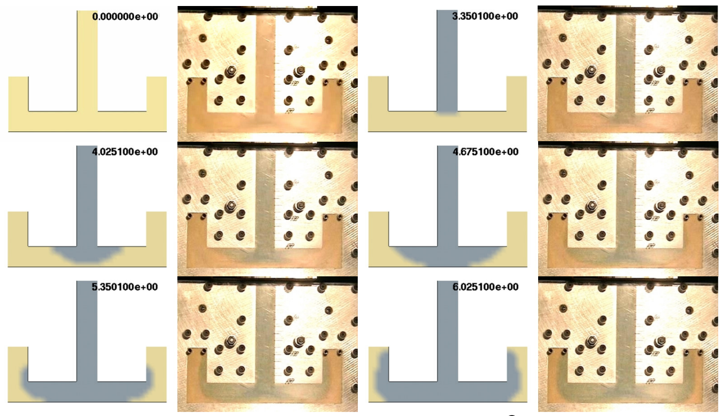


Figure 5: Validation of the Arena-flow® curing model

Figure 5 compares the computed and observed curing of the cold-box core as well. Experiments have been performed with indicators in the sand-binder mix to enable video data capture of the curing process[3]. The shape and speed of the cure are captured extremely well by the computation.

The use of the CPFD™ method in ***Arena-flow®*** has also been validated for many production cores and molds. A few published examples are included here. An early example of the CPFD™ technology being applied to sand core blowing involves the filling of a automotive cylinder block water jacket[14,15,16,2]. The first reference won the prestigious Howard F. Taylor award from the American Foundry Society (AFS) in 2003 and went on to win Best Paper from the World Foundry Organization (WFO) in Istanbul, Turkey in 2004.

Arena-flow® was used to model the filling of the water jacket core under various conditions with various blow tubes opened or closed. A strong correlation was observed between the calculated and observed results. In particular the air pressures were monitored in the magazine, blow plate and core as shown in Figure 6. Figure 7 shows the correlation between the measured and computed results.

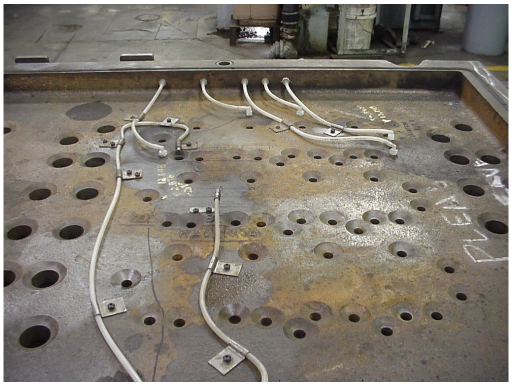


Figure 6: Location of pressure taps in blowing machine

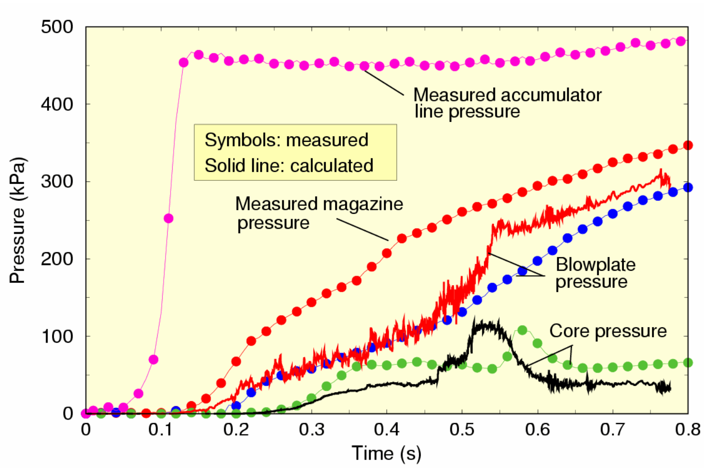


Figure 7: Comparison between measured and calculated pressures in a production blowing machine

Note: *Arena-flow*® correctly computes the air behavior in a blowing machine as well as the sand motion. This is characteristic of the CPFD™ technology.

Figure 7 is significant because it shows that the CPFD™ method computes the correct behavior of the air flow, as well as the sand. The pressures in the cores are much lower than the pressures observed in the blow plate, and a pressure spike and drop is also computed well as the core fills.

Figure 8 shows a comparison of the final core shape for a highly-complex, heat exchanger flue way core with a repeatable defect[2]. ***Arena-flow*®** was able to predict both the location and cause of the defect with remarkable accuracy.

Note: ***Arena-flow***® correctly computed both the location and cause of a repeatable defect in a highly complex, heat exchanger flueway core.

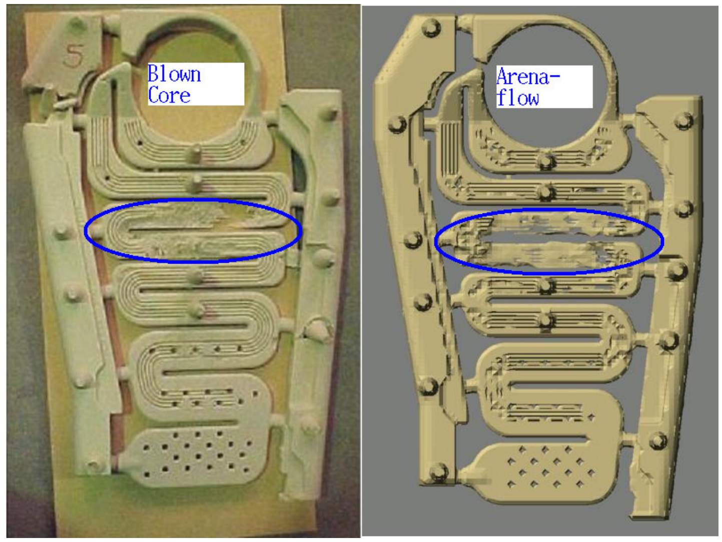


Figure 8: Comparisons of an actual core (left) and the *Arena-flow*® prediction (right) for a complex, heat exchanger flue way core with a repeatable defect

While ***Arena-flow***® has also been used with great success for a wide variety of problems ranging from single cores to family boxes, from binder-coated sand to shell sand, from sections of cores to magazine filling simulations and from blowing to curing; an exhaustive list of validation examples is not warranted here. The examples provided herein are intended to give the reader confidence in the soundness of the numeric method behind ***Arena-flow***®, and the applicability of the method to sand core blowing and curing.

Additional comfort can be taken in the validation of the CFPD™ technology for other applications. Other software packages have successfully applied the CFPD™ methodology for the filling of lost foam tooling[10], modeling cyclone separators[8], gas-solid risers[9], turbulent bed reactors, jet penetration in fluidized beds, fluidized bed dryers and other fundamental fluidization applications.

1.4 Arena-flow® Capabilities

In this chapter you will learn about:

- *The capabilities of* ***Arena-flow***®
- *The modules in* ***Arena-flow***®

Thus far we have discussed the importance of high-quality sand cores to overall product quality, outlined the CPFD™ methodology as a means of modeling core blowing and shown the accuracy of the method for fundamental experiments, test cores and production cores. With the fundamentals in place, our attention can be directed toward the usefulness of ***Arena-flow***® as a foundry tool.

Arena-flow®'s user-friendly interfaces are explained in more detail in section 2 of this User Guide. With the convenient project setup features, it is easy to lose sight of ***Arena-flow***®'s advanced modeling capabilities.

> **Note:** Most users are unaware of all of ***Arena-flow***®'s advanced modeling capabilities.

Arena-flow® is advanced sand core engineering software which can be used to compute:

- core fill / non-fill
- density variations in a filled core
- speed of fill
- balance of fill
- location of difficult-to-fill regions
- cause of difficult-to-fill regions
- the potential for any strange or unexpected behavior during the filling process
- the motion of "colored sand" from magazines
- the mixing of "colored sand" from various blow tubes
- magazine effects including rat-holing
- the filling of magazines
- tool wear
- resin wipeoff
- sand flux through user-defined regions (into cores, through individual blow tubes)
- air flow behavior throughout the system
- air pressure response throughout the system
- regions of poor air flow during the blowing or curing cycles
- the motion of the curing front during the curing cycle

The ***Arena-flow***® user interface is process oriented with a modular look and feel. Unique interfaces exist for binder-coated sand core blowing, shell sand core blowing, steady-state gas flow analysis, transient curing analysis and an expert interface for more sophisticated modeling. Each module is outlined below.

1.4.1 Binder-coated sand

The ***Arena-flow***® binder-coated sand core blowing module provides users with a streamlined setup specific to modeling the filling of cold-box cores.

Inputs:

- core, blow tube, magazine geometry
- sand type (material density and size distribution)
- initial sand location and packing
- transient blow pressure
- blow time
- venting (locations, sizes, open area fraction, losses)

Typical output:

- core fill / non-fill
- animation of filling pattern
- final density variations
- tool wear
- resin wipeoff
- sand motion (speed, colored sand, etc.)
- air flow and pressure distribution

1.4.2 Shell sand

The ***Arena-flow***® shell sand core blowing module provides users with a streamlined setup specific to modeling the filling of shell sand cores and molds. Inputs and typical output are very similar to the binder-coated sand module, however various model parameters are defaulted differently to better simulate the phenomena of shell sand. These custom models include sand grain interaction with the tooling walls (normal and tangential losses), sand grain interaction with packed regions of the core and sand flowability defaults.

It should also be noted that ***Arena-flow***® is an isothermal software package which means that temperature changes are not modeled. However, ***Arena-flow***® does allow users to numerically "look inside" the tooling to visualize the core filling process, enabling the engineer to make informed decisions regarding the filling behavior of shell cores and molds.

Inputs:

- core, blow tube, blow plate geometry
- sand type (material density and size distribution)
- initial sand location and packing
- transient blow pressure
- blow time
- venting (locations, sizes, open area fraction, losses)

Typical output:

- core fill / non-fill
- animation of filling pattern
- final density variations
- tool wear
- sand motion (speed, colored sand, etc.)
- air flow and pressure distribution

1.4.3 Steady-state gas flow

Note: Steady-state calculations typically run very fast – with several design iterations possible in a short timeframe.

The ***Arena-flow***® steady-state gas flow module is intended to give users a snapshot of the air flow through the filled core at the maximum gas / purge condition. Since it is "steady-state" or time independent, the solver models the sand as a porous media and runs extremely quickly, with several design iterations possible within a short timeframe, by identifying and attempting to eliminate regions of poor air flow.

The steady-state gas flow module is also useful for new tooling design. By analyzing the air flow through the core during the blowing process, several design iterations can be simulated in a short period of time. The final design from the steady-state simulations can be the starting point for the core blowing simulation with either the binder-coated sand or shell sand modules.

Inputs:

- core, gassing head (blow tube) locations
- sand location
- sand effective radius
- max gas / purge pressure
- venting (locations, sizes, open area fraction, losses, pressures)

Typical output:

- air velocities through filled core
- air pressure distribution through the system
- regions of poor air flow

1.4.4 Transient curing

The ***Arena-flow***® transient curing module tracks the motion of the curing front through a filled core. The sand is modeled as a porous media, enabling computations in a few hours or less.

It should be noted that transient curing calculations do not compute the complex chemistry involved in the cure of the core, but rather the progression of the curing front through the core. The shape and progression of the front is expected to be very accurate, however the timing and extent of cure is dependent upon many variables which are not modeled (amine type, binder type, binder concentration, temperature, humidity, etc.). By optimizing the progression of the curing front, cores are usually cured in the minimal time with the minimal use of consumables.

Note: Optimizing the progression of the curing front through the core usually also optimizes the curing cycle time and amine usage!

Inputs:

- core, gassing head (blow tube) locations
- sand location
- sand effective radius
- transient gas / purge pressure cycle
- venting (locations, sizes, open area fraction, losses) including transient pressure cycle if amine enters through the cope vents

Typical output:

- transient curing front progression through the core

1.4.5 Expert

The ***Arena-flow***® expert module is an advanced control interface. Most calculations can be setup, run and analyzed with the other modules of ***Arena-flow***®. The expert module grants the user much greater control over the specific models used by the CPFD™ solver. This enables fine-tuning of the model parameters to a user's specific process, but also creates the potential for introducing inaccuracies to the models. The ***Arena-flow***® model parameters are designed with default values which result in agreement with available data for the maximum number of test cases studies. While it may be possible to improve agreement for a single test by tuning a model, the agreement may degrade for other cases.

Users should only access the ***Arena-flow***® expert module when instructed to do so by engineering support staff. In general, it is a minimally supported interface.

1.5 Arena-flow® Usage

In this chapter you will learn about:

- *Asking yourself what you want to learn from an **Arena-flow**® calculation*
- *Asking good questions about the process and what needs to be included in the computational model*
- *Analyzing core blowing results*
- *Analyzing core curing results*
- *How **Arena-flow**® results compare with your foundry*

We are almost ready to begin using the ***Arena-flow***® Graphical User Interface (GUI), but first ask yourself a few simple questions:

- Why do you want to run a computer simulation of the coremaking process?
- What do you want to learn from the simulation?

We have outlined the importance of quality cores and even the value of process modeling earlier, but now the questions are not generic, they affect your foundry directly. You are likely familiar with core making and may have years or decades of experience. On the other hand, you might never have set foot in a foundry. Regardless, think about what you hope to learn. By thinking about the process and asking good questions before starting every project, you will get the most out of your investment into ***Arena-flow***®.

Tip: Think about the process and problem, before you run any calculation!

Here are some questions that should be considered before beginning a project:

- *Am I modeling new tooling*?

  The simulation of new tooling is different than the analysis of existing tooling. When modeling new tooling, your role is that of designer. What should be the orientation of the core in the box? Where should the blow tubes be located? What size? How many? What shape? Where should the vents go? What size? How many? The questions go on.

  With new tooling it is good to run many design iterations quickly using steady-state calculations, and then simulate the transient core blowing and curing (if applicable) on the final iteration. More changes to the design may be needed after the transient calculations are studied.

  By modeling new tooling before it is constructed, you will identify many potential problems before they occur and before metal is cut, vents are drilled or equipment is ordered.

- *What problems exist with this process*?

  With existing tooling your role is typically that of problem solver. Your software does not know everything; don't start with ***Arena-flow***®. First talk to your process staff. Talk to your machine operator. What problems are they having? What do they think would be causing these problems? How often do they occur? Ask these types of questions before beginning.

Tip: Be sure to speak with the core shop staff about existing tooling before starting your simulation. They are the experts and can save you a lot of time!

- *Are they likely blowing or curing problems*?

  By speaking with the experts on the process before starting your calculations, you may save yourself a great deal of time. For example, the operator may tell you that the cold-box core often has a poorly formed edge and there is loose sand there when the box is opened. That should tell you it is likely a curing problem, not a blowing problem.

- *How many cores are blown on the machine?*

  Sometimes many cores are blown on the same machine. Each core could be identical or several different cores could be formed simultaneously in a "family box." How much of this should be modeled? If several different cores are constructed, think about modeling each one individually first since the individual calculations may run much quicker than the simulation of the entire machine. You can learn about the potential problems of each core (assume a uniform sand feed from the magazine above), and then later run the entire machine if warranted.

  Alternately, if the problems occur near the outer blow tubes on the blow plate, it is likely necessary to run the entire magazine. Think it through. If you want another opinion before beginning, contact support – we're here to help.

- *Do all cores on the machine have problems, or only some*?

  If several identical cores are produced on the same machine with only some having problems, what should that tell you? Think about what could be causing the problem and what needs to be included in your simulation before beginning.

- *Should I model the magazine*?

That's a big question. When simulating an entire magazine, the CPFD™ technology in ***Arena-flow***® not only needs to model more computational cells, but also must model a lot more sand. Remember, CPFD™ is like 2 solvers in one! Modeling a magazine requires more run time and memory, but it may be necessary. Think it through before beginning. If you do not model the magazine, you are assuming an ideal sand distribution in the blow plate at the top of your model. Is this a valid assumption for your process?

- *What about magazine filling*?

  Even if you model the blowing of the core(s) with the entire magazine, you may still not be modeling exactly what happens in the core shop. Is the magazine truly full at the start of the blow cycle? This can be modeled if necessary.

- *Who else should I speak with about this*?

  Whenever considering design changes be sure to involve the right staff. Is your proposed change feasible? Is it practical? There's no sense in modeling a magazine that's twice as high to solve a problem if the existing magazine is flush against the ceiling! Save yourself time and keep everyone "in the loop."

> **Tip:** Be sure to involve all the affected parties in the consideration of any modifications to ensure the proposed changes are feasible and practical.

- *What if …*?

  Design iterations are the most powerful use of any simulation tool. Once you've done the work to set up the first calculation, start thinking about what can be changed. What if I moved this vent? What if I closed this blow tube? What if…?

> **Note:** Design iterations are a very powerful use of any simulation tool. Once you've set up your first model, you've done 95% of the work on your second!

These are only some of the questions to be asked. This is not an exhaustive list, but rather the type of thinking that should proceed and accompany any process modeling.

Once you run the calculation, be sure to continue asking questions while post-processing the results. More detailed information about how to use the results to answer these questions is given in the "Post-processing" chapter of this User Guide and in the training material, but be sure to ask these questions when looking at your results.

These are some questions to ask when studying the results of a core blowing simulation:

- Did the simulation show the core filling?
- What does the final density plot tell me?
- What does the transient filling animation tell me?
- How balanced is the fill?
- Does the transient filling animation explain the density variations in the filled core?
- At any point in time is there an unfilled region in the core with no direct path for sand to take between a blow tube and that region?
- What regions have the potential for problems?
- What causes those regions to have the potential for problems?
- Does the colored sand show the filling more clearly?
- How is the air flowing during the core filling?
- Where do I expect tool wear?
- If this is a shell process, where could the filling be even worse than predicted due to premature hardening of the core?
- Would I have expected these results?

Tip: Continue asking and answering good questions when studying your results!

Tip: When modeling the shell process, consider which regions may be worse than predicted due to premature hardening of the core. With ***Arena-flow***® you can see the sand motion to make these types of informed decisions.

Similarly, when studying core curing, ask questions like these:

- What does the air flow do (steady-state module)?
- What does the curing front progression look like?
- Is the curing front progression uniform and optimal?
- What am I modeling?
- What am I not modeling?

One last note on how ***Arena-flow***® is used in real foundries. What happens in the foundry is what is real. The computational results are merely a model. The real machine will likely have some variation from blow to blow. Very seldom does a machine produce 100% scrap. Often a "big problem" could be 10% scrap or perhaps 5% scrap. In the 5% scrap example, what is different between the 95% good cores and the 5% bad?

Tip: Always consider the assumptions inherent in your computational model.

Tip: Always look deeper than the final fill of the core and consider how the core fills and why!

Next ask yourself which core ***Arena-flow***® simulated? Was it one of the 95% good or the 5% of lower quality? Very often you will get excellent agreement between the simulation and the actual process – great. However, sometimes there will be some variation. Always consider your assumptions inherent in your computational model. Always look beyond the final fill of the core and consider *how* the core fills and *why*. This is the real value of ***Arena-flow***®.

Arena-flow® is a useful tool when complemented with sound engineering judgment which allows you to look inside your tooling to help you understand what's happening in your core making process. You are the expert about your foundry and its processes. By using the information provided by ***Arena-flow***®, you can make quality decisions toward creating optimal cores and subsequent castings.

2 Arena-flow® Graphical User Interface

2.1 Arena-flow® Installation

In this chapter you will learn about:

- *Installing **Arena-flow**® on your Linux computer*
- *Activating your **Arena-flow**® software license*
- *Optimizing performance for **Arena-flow**® systems*

Before being able to run ***Arena-flow***® on your Linux hardware, the software must first be installed and your license must be activated.

Most installations of ***Arena-flow***® can be completed via the following steps; full installation notes can be found on your installation CD. Before beginning, please close all active programs, have your “root” password handy and ensure your CD drive is mounted as “executable” to run the ***Arena-flow***® installer. Please contact support with any questions.

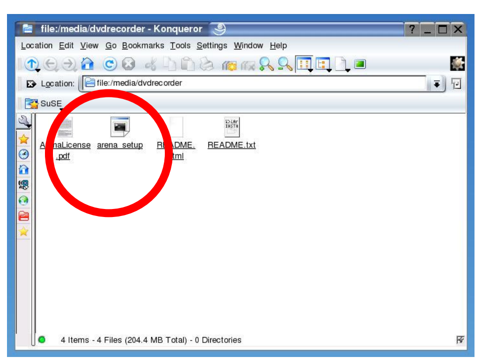
To install ***Arena-flow***®:

1. Insert the ***Arena-flow***® install CD into the computer. The computer must be running Linux, and newer SUSE versions are preferred.

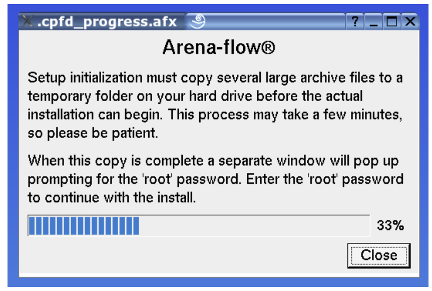
2. Browse to the CD drive. A window may pop up or you may need to click on your CD/DVD desktop icon.

3. Launch the “arena_setup” program. For SUSE 10.1 and newer, the arena_setup program must be run in a terminal window.

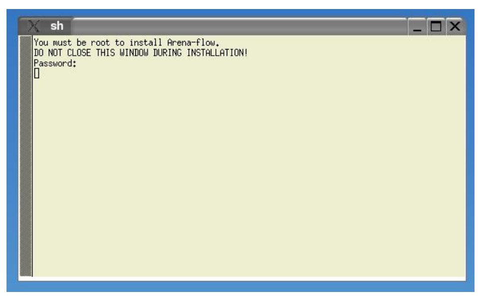
Important: With some computers and some CD drives it takes a few moments for the progress bar to appear. Please be patient and only click on the “arena_setup” program once.



The “arena_setup” program will copy several large files to the hard drive. This may take several minutes and you may not see any screen output. Please be patient. A progress bar will appear to keep you informed.

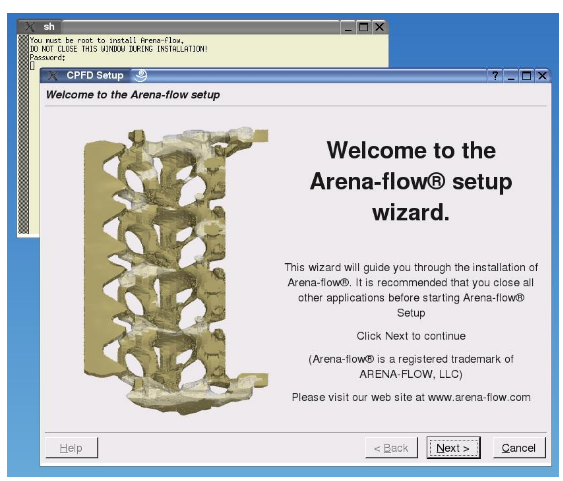


4. Once the file copy is complete, a window will appear for you to enter the computer’s “root” password. Please enter the “root” password and keep this window open during the installation process.



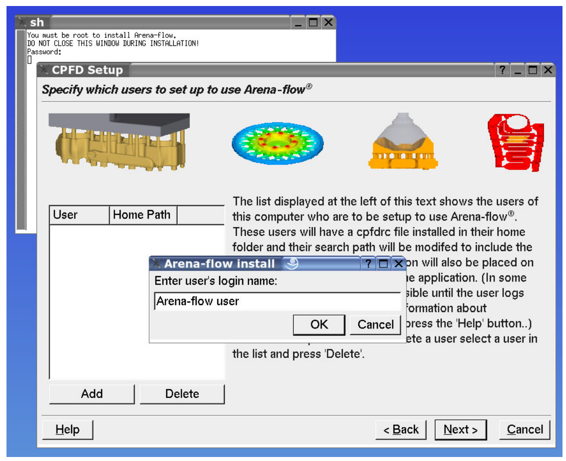
After entering the “root” password, the ***Arena-flow***® setup utility is launched.

Important: Be sure to keep the password window open during the installation process. Closing the password window will terminate the setup.



5. Navigate through the windows using the "Next" and "Back" buttons. Select default values wherever possible. Please note that for Ubuntu, Red Hat, CentOS and some other Linux systems, the default installation locations must be used for full functionality**.**

6. When prompted, enter the users which will operate ***Arena-flow***®. This allows ***Arena-flow***® to set "PATH" variables and place a desktop icon in the appropriate user accounts.



Important: Be sure to log out and back in again after installation to ensure all changes to your system take effect.

7. When installation is complete, please log out and log back in again to ensure all changes take effect. This is required on some operating systems. It is <u>not</u> necessary to reboot your machine.

The ***Arena-flow***® setup utility automatically installs either a 32-bit or 64-bit version, pending your own hardware. The 64-bit solver has been observed to run considerably faster than the 32-bit solver for practical, commercial applications. Please contact your sales representative for more information.

If you have any questions during the installation process, please contact support@arena-flow.com.

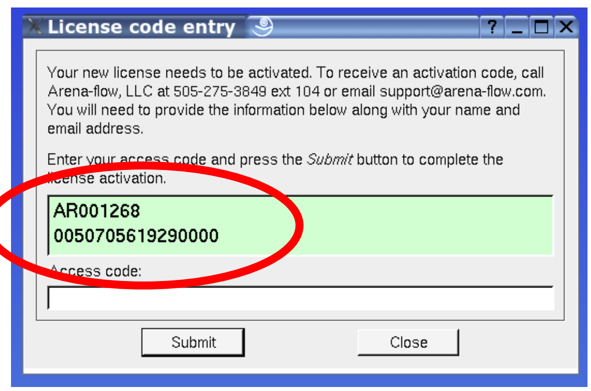
In order to run ***Arena-flow***® you will need a valid license file or hardware key. You only need to install the license file when you first install the software. For future software updates a new license file is not required, provided that the license is not expired.

To activate your license, simply launch ***Arena-flow***® with your license file installed. Instructions for the installation of the license are provided with the file.

Click on the desk-top icon to launch ***Arena-flow***®.

Arena-flow® will detect the presence of the unactivated license file and prompt you to contact support for activation.

> **Note:** Once you activate your license, it remains activated on the computer until it is expired or removed. You do not need to reactivate a license every time you install a newer version of ***Arena-flow***®.



> **Note:** All license information and "access codes" are case sensitive.

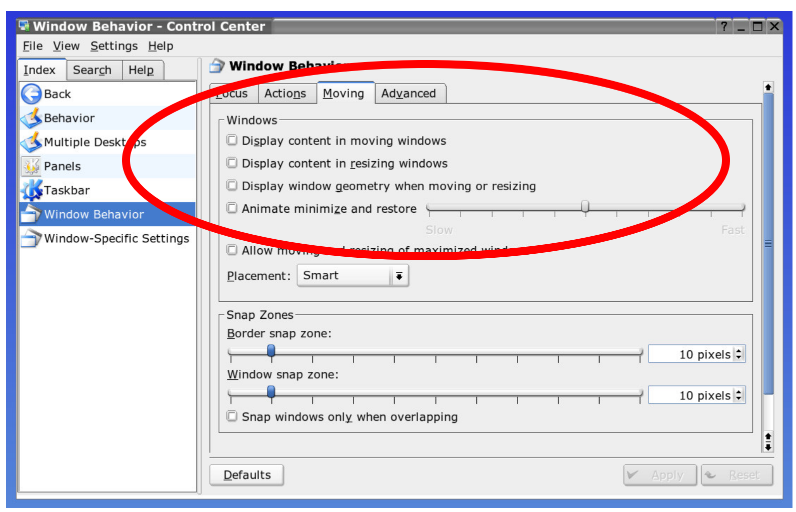
Please send the two activation code numbers to support@arena-flow.com, or call for more immediate assistance. Also note that the alphanumeric characters are case sensitive. The support staff will provide you with an "Access code". Please enter the "Access code" where prompted.

You will be prompted to accept the terms of the "***Arena-flow***® Computer Aided Engineering (CAE) Software License and Agreement" and then the ***Arena-flow***® GUI will appear. Once your license is activated, it remains activated on the computer until it expires.

Before running calculations with ***Arena-flow***®, it is important to ensure your system is set up optimally, to obtain the most value from your investment. The CPFD™ technology uses a great deal of your CPU and memory, so be sure to maximize the computer "horsepower" available for the ***Arena-flow***® solver.

Tip: Please turn off all unneeded programs which may divert computational resources away from ***Arena-flow***®. In particular, deactivate all screen savers and extraneous window drawing settings.

To do so, turn off all unwanted programs including screen savers, and turn off all animation settings. Additionally, for graphical output using GMV, be sure that your computer does not attempt to redraw the contents of windows which are being moved or resized. For newer SUSE systems, this setting can be found in the "Control Center", under "Desktop", "Window Behavior" as shown below.



2.2 Project Management

In this chapter you will learn about:

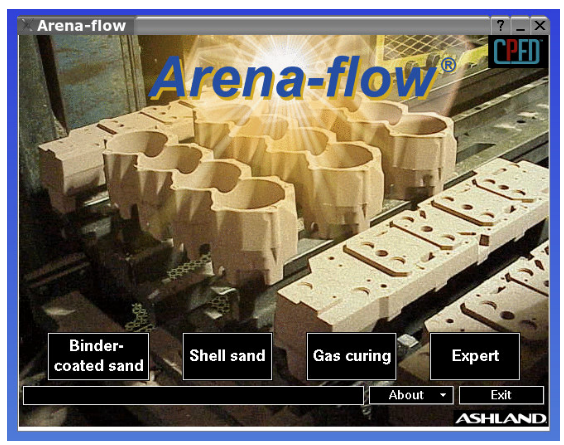
- *The **Arena-flow**® software package*
- *The files necessary to run an **Arena-flow**® calculation*
- *Project directories*

- *Files requires for software support*
- *Disk space and project archiving*

The ***Arena-flow***® program installed on your computer is actually a full engineering software suite including a Graphical User Interface (GUI – arena), CPFD™ grid generator (cpfd.g), CPFD™ solver (arena.x), shared memory interactive utility (act), third-party graphical post-processor (GMV) and various other support utilities. This may sound confusing, but the ***Arena-flow***® GUI is all you need to coordinate this powerful engineering software package.

To get started, simply click on the

desktop icon, or type “arena” at the command prompt. This will launch the ***Arena-flow***® GUI.



There are several ways to interact with the ***Arena-flow***® GUI, as indicated by the various modules: “Binder-coated sand”, “Shell sand”, “Gas curing” and “Expert”. These modules will be further explained in the following chapters. However, regardless of the module used, all communication between the various programs included in the ***Arena-flow***® software package is handled through a project file.

> **Note:** The ***Arena-flow***® GUI is all that is required to manage all programs and files.

All the information required to run an ***Arena-flow***® calculation is stored in, or referenced by, a project file. The project file links to all the input files. Examples of input files include CAD models with a .stl or .STL extension, transient boundary conditions with a .i extension, sand size distribution files with a .sze extension and vent files with a .tvt extension. The ***Arena-flow***® GUI passes the information from the project file and any linked files to the ***Arena-flow***® software suite. The ***Arena-flow***® software suite in turn creates more files. The post-processing of these files is coordinated by the GUI as well. Schematically this may be viewed as shown in Figure 9.

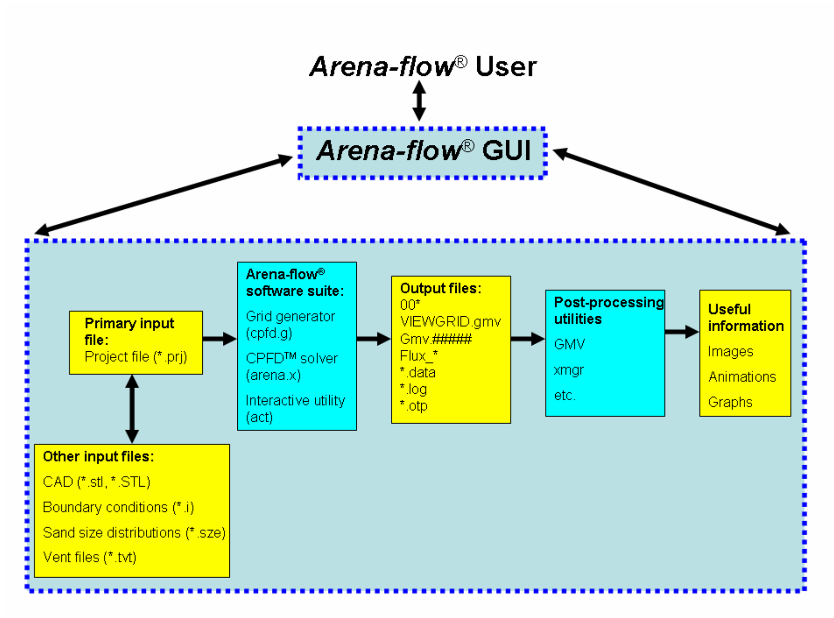


Figure 9: The Arena-flow GUI controls all programs and input / output files

> **Important:** Always run every ***Arena-flow***® calculation in its own directory.

Due to the many files required and produced, it is very important to run each ***Arena-flow***® simulation in its own directory. It may be useful to keep a list of all runs and the corresponding directories in a text file.

> **Tip:** When creating a new directory for a new simulation, be sure to copy these files. These will also be required when contacting software support.

When copying files to a new directory for a new simulation, be sure to copy:

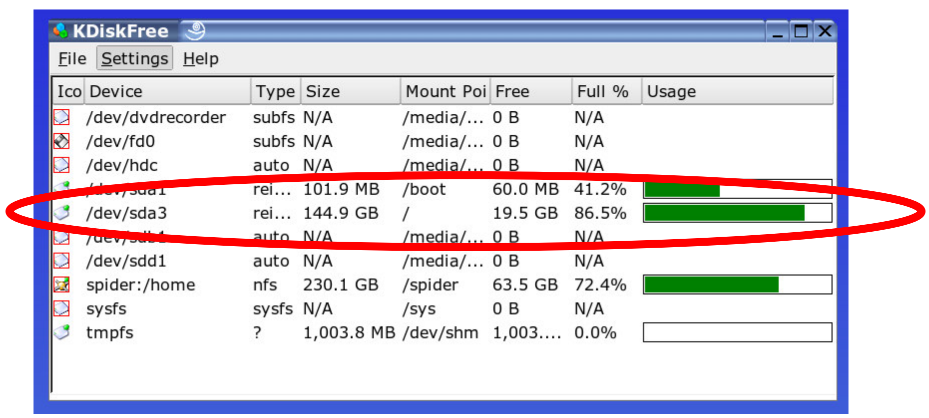
- the project file (*.prj)
- the CAD (*.stl or *.STL, unless linked)
- the boundary conditions (*.i)
- the sand size distributions (*.sze)
- the vent file (*.tvt)

When contacting ***Arena-flow***® software support the same files will be required. Additionally, the support engineer may require some additional files such as:

- history.log
- info.log
- various output files

Because of the many files produced by an ***Arena-flow***® computation, you should be mindful of your disk space. It is good practice to have at least 15 – 20 GB of hard drive space available before starting a calculation.

The hard drive space can be monitored by typing "df" at the command line or by launching the "KDiskFree" program on SUSE systems. To launch the program click on "Start", "System", "File System", "KDiskFree". This may vary from version to version. A utility is launched as shown below:



Always view the drive that is mounted as "/". In this example the drive is 144.9 GB and has 19.5 GB available.

To free more disk space you must delete files from your hard drive. It is good practice to archive any useful data before deleting it. Archiving a directory involves copying the information to another computer, an external hard drive or burning a CD or DVD.

When burning a CD or DVD, you may not have sufficient space to store every file. In such cases it is recommended to store the following:

- All files needed to recreated the calculation (*.prj, *.stl, *.STL, *.i, *.sze, *.tvt)
- All files needed to view the results (00*, VIEWGRID.gmv)
- Some Gmv files - use your discretion depending on how much disk space you have. For example, Gmv.*0 and Gmv.*5 would be every 5th Gmv file.
- history.log
- Flux*
- *.data
- info.log

2.3 Grid Generation

In this chapter you will learn about:

- *Input CAD files*
- *The purpose of the grid for a CPFD™ solver*
- *Defining a grid for use with the various **Arena-flow®** modules*
- *Generating a grid*
- *Evaluating the quality of the grid*
- *Grid resolution for the CPFD™ solutions*

Every ***Arena-flow®*** calculation begins with a definition of the geometry to be studied. Commercial Computer-Aided Design (CAD) packages are excellent for defining three-dimensional geometric regions. The output from these packages serves as the input for ***Arena-flow®*** computations.

Important: A suitable CAD file for use with ***Arena-flow®*** must come from a three-dimension solid model and must be output in STL format.

A suitable input file for an ***Arena-flow®*** calculation must be created from a three-dimensional solid model and must be output in STL (STereoLithography) format. Please note that STL files built up from two-dimensional CAD packages are unsuitable for use with ***Arena-flow®***. If you have any questions about the suitability of a particular STL file, please contact support.

Tip: Whenever possible, work with binary STL files to save space and time.

All mainstream three-dimensional solid modeling CAD programs can output STL files either by “saving as” STL or “exporting to” STL or rapid prototyping formats. Often the CAD package will provide the option to export in ASCII or binary format. Whenever possible, please work with binary files to save space and time.

Tip: Whenever possible, create your STL files with the z-axis pointing up.

The STL output file consists of triangular surface patches and outward normals. The STL format was designed for the rapid prototyping industry and has become that industry's standard transmission format. It is easiest to work with STL files in ***Arena-flow***® if the z-axis is up (i.e. the gravity vector points in the –z direction)

Important: The STL file must consist of the open area through which the air and sand flow. The complement of this volume (i.e. the tooling) is unacceptable.

The STL file must consist of the open area through which the air and sand flow. This solid volume would include the core(s), blow tube(s), blow plate and magazine regions. Please note that the complement of this volume (i.e. a solid model of the tooling) is unacceptable as input into ***Arena-flow***®.

Multiple STL files are permitted, and recommended. Whenever possible, create separate STL files for your core, blow tubes and blow plate / magazine as shown in Figure 10. This will facilitate design permutations as your project progresses.

Tip: Whenever possible, create separate STL files for your core(s), blow tube(s) and blow plate / magazine.

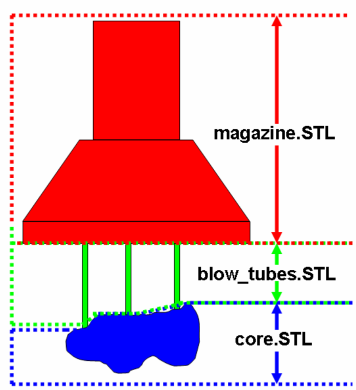


Figure 10: Use of multiple STL files to separate the core(s), blow tube(s) and blow plate / magazine regions

Please note that a CAD model of the vents is not required. The vents will be applied as discrete boundary conditions to the CPFD™ model.

Once the CAD is created it can be imported into ***Arena-flow***® to create a suitable CPFD™ model of the problem. This model is related to a "grid" or "mesh" which translates the real, three-dimensional region of space into the model space. Let us consider the purpose of the grid.

With a CFD solver, the grid is used to subdivide the domain of interest into a finite number of control volumes. The fluid-flow equations are then discretely mapped onto this finite space, and integrated through space and time. Although all discretizations of continuous models contain approximations, for a useful method as more and more grid cells are used, the answer more and more resembles reality for some quantity of interest.

Note: The CFD component is only one half of the CPFD™ solver. ***Arena-flow®*** also models millions of computational sand grains on a sub-grid level.

Since the CPFD™ method also contains a CFD solver, CPFD™ grids also are used to map the infinite, three-dimensional fluid flow domain to a finite region for solution. However, the CFD component is only one half of the CPFD™ solver. ***Arena-flow®*** also models millions of computational sand grains, which are able to move on a sub-grid level. The grid is still used for the particles to interpolate between the fluid and solid phases and to speed the calculation of spatial gradients, but the sand particles may have any three-dimensional spatial location not directly tied to the grid. Thus sand grains may move within a grid cell, or from cell to cell with no directly grid-related constraints.

Note: Typically ***Arena-flow®*** grids are much coarser than traditional CFD grids.

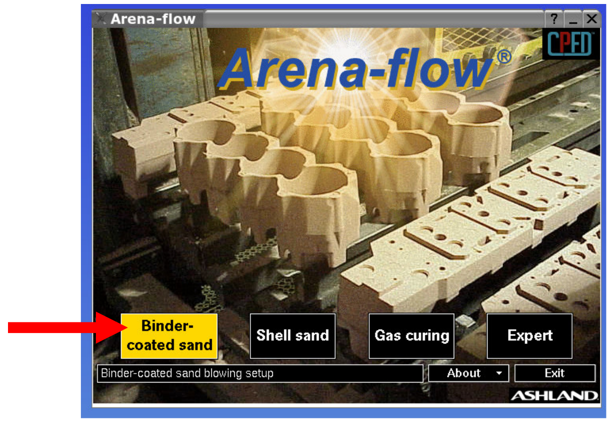
There are two sources of resolution for a CPFD™ solver compared with one for a purely CFD method. One source is from the fluid on the grid and one from the particles, free from grid constraints. Also, the methodology assumes that a statistically significant number of computational sand grains must be able to fit within a computational cell. Thus, typical ***Arena-flow®*** grids are much coarser than many traditional CFD grids.

Arena-flow® actually contains more than one type of CPFD™ solver. The solution requirements for core blowing differ from those of core curing, and the appropriate solver is used for each. Each solver has its own gridding constraints. These are discussed in the following subsections.

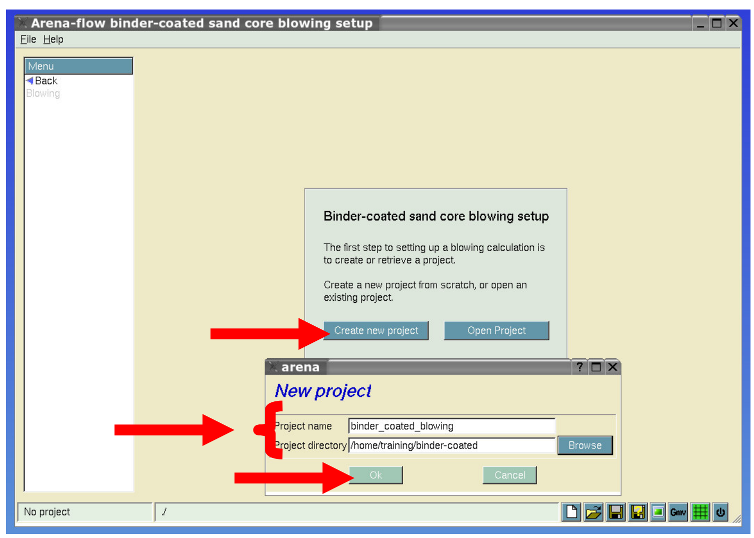
2.3.1 Binder-coated sand grid generation

To model the blowing of a cold-box core with ***Arena-flow***®, click on "Binder-coated sand" as shown.

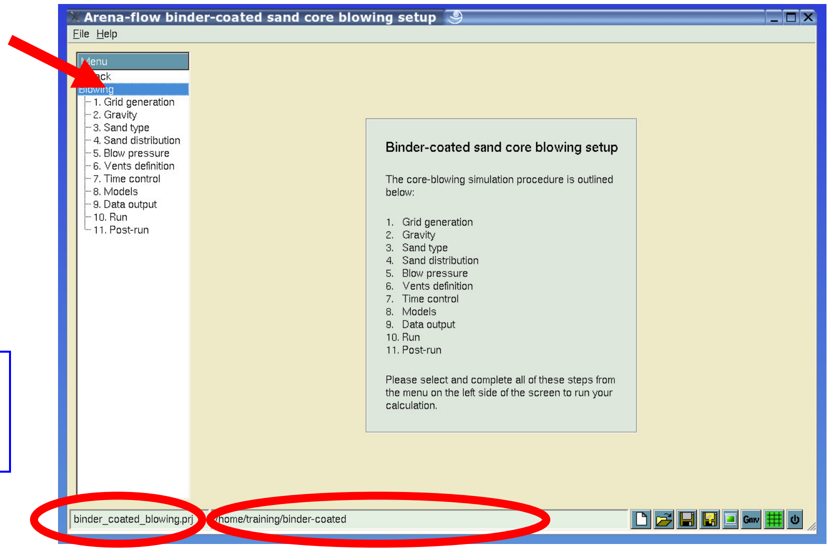
Note: Begin every ***Arena-flow***® calculation by specifying the process you wish to model.



This raises the "***Arena-flow***® binder-coated sand core blowing setup" window. Click on "Create project" and enter a "Project name" and "Project directory". Recall that each ***Arena-flow***® calculation should be run in its own directory. Click "OK" when complete.

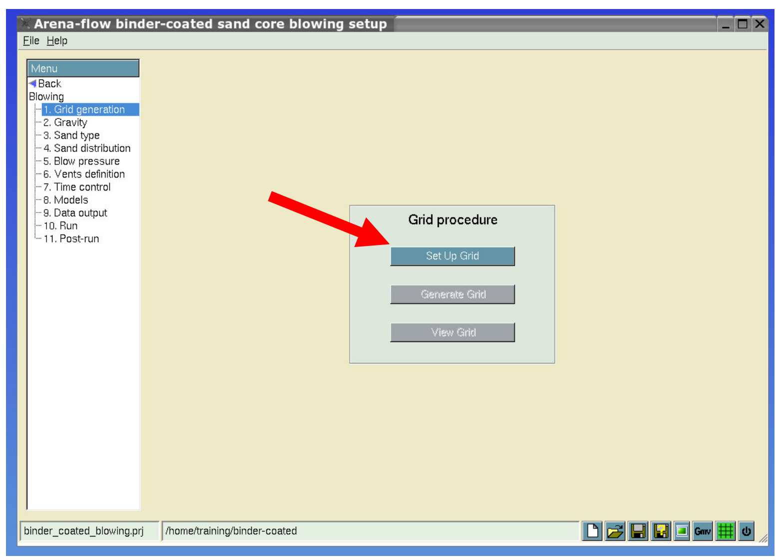


Next you will see a page outlining the core blowing procedure. Also notice the project file name in the lower left corner, and the working directory in the lower center. Always ensure you are working with the correct project file in the correct directory.



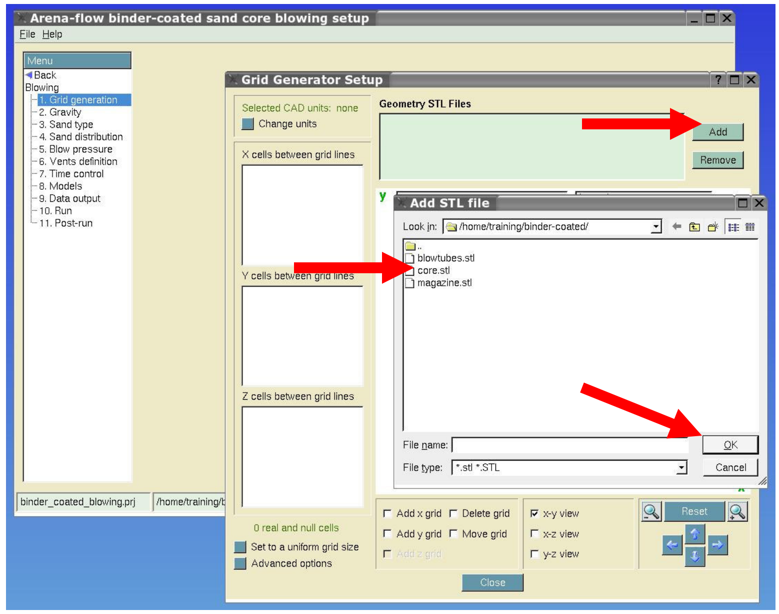
Tip: Always verify your project file name and working directory.

You can navigate through the simulation procedure steps by clicking on the menu items on the left. Click on “Grid generation” to go to the “Grid procedure” window.

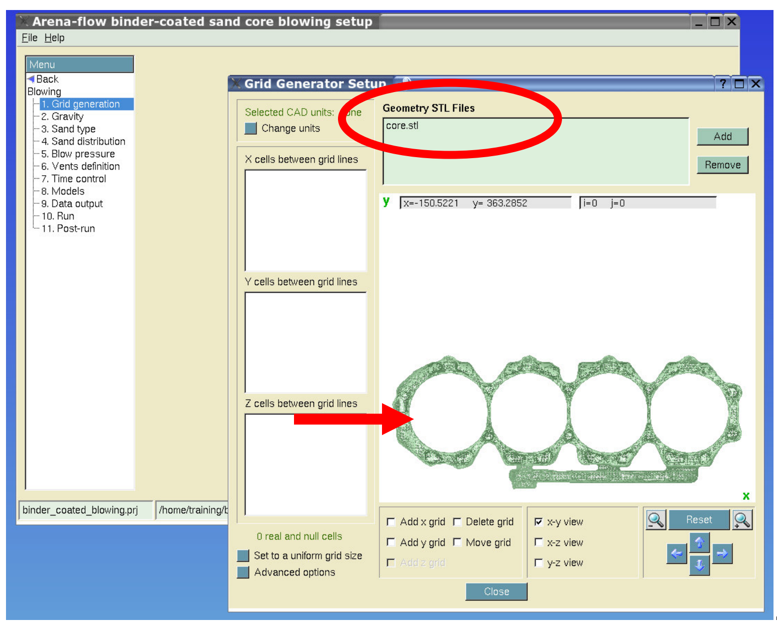


Note: Work through the simulation procedure by clicking through the menu items on the left of the window.

This window contains three items: “Set Up Grid”, “Generate Grid” and “View Grid” (***Arena-flow***® 6.0 shown). Notice that the latter two are not available at this time. To get started, click on “Set Up Grid” to raise the “Grid Generator Setup” window.



The first step in generating a grid is to tell ***Arena-flow***® which STL files will be used. Click on “Add” to raise the “Add STL file” window. You can browse through your computer and select the desired STL files. Click “OK”. It is not required that your STL file be stored in your working directory.



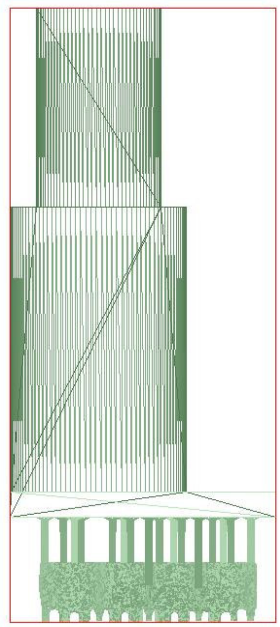
The STL filename appears in the “Geometry STL Files” section and the STL triangles appear in green in the viewing window. Multiple STL files can be added by repeating this procedure.

Once the STL files are added to the "Grid Generator Setup" window, a mesh must be defined. The specification of the grid is discussed later in this chapter – the appropriate extent and parameters of the grid are covered here.

There are two options available when modeling the blowing of binder-coated sand cores:

1. Modeling the entire blowing machine.

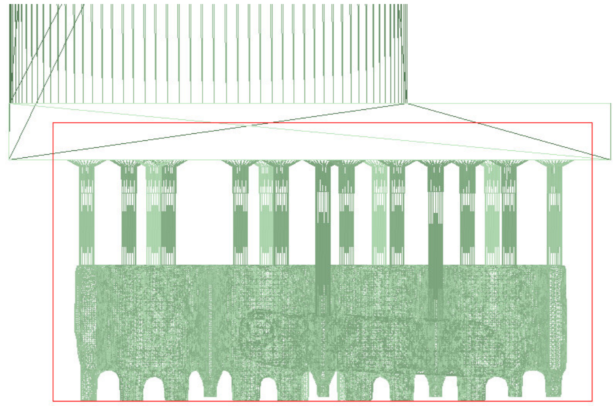
Note: To capture magazine effects when modeling binder-coated sand core blowing, model the entire blowing machine including core(s), blow tubes, blow plate and magazine.



   In the example shown, the grid would span the core(s), blow tubes and blow plate / magazine. This calculation would capture magazine effects such as rat-holing but requires many cells and computational sand grains to be modeled. This translates into increased runtime and memory requirements.

2. Modeling the core, blow tubes and part of the blow plate.

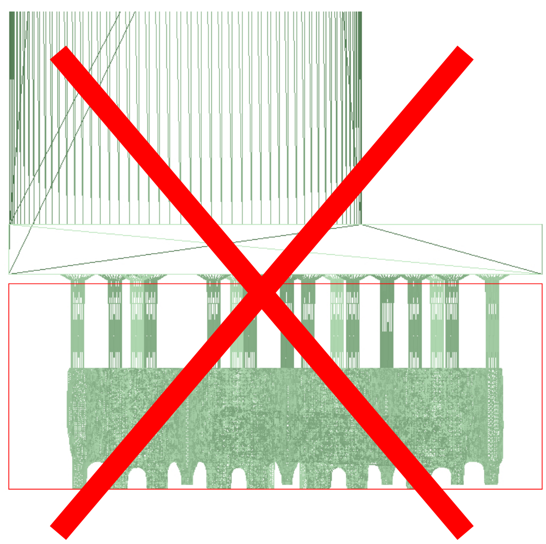
Note: If magazine effects are not required, model only the core, blow tubes and part of the blow plate. This will assume an ideal pressure distribution and sand feed from the magazine, above.



In the example shown, the grid would span the core(s), blow tubes and only part of the blow plate. This calculation would run faster than the first option (entire blowing machine), but would not be able to capture magazine effects such as rat-holing. This calculation would assume a uniform sand and air pressure feed into the top cells from above.

Important: Never start your core blowing model at the top of the blow tubes. Always include at least a portion of the blow plate.

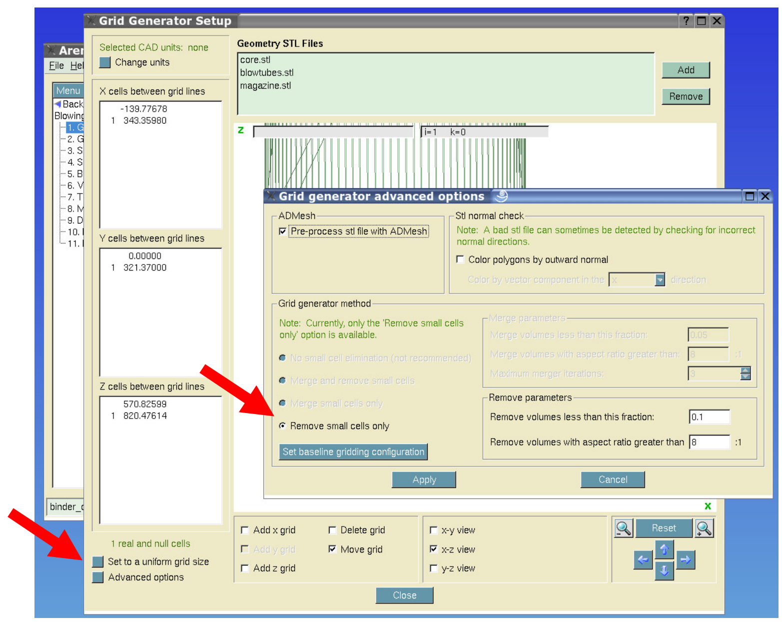
Regardless of extent of the model, never start your calculation at the top of the blow tubes.



It is bad practice with any numeric method to place your boundary conditions at locations where the parameters are unknown or at the location of strong gradients. During the blow cycle, strong gradients occur in the blow tubes where the sand volume fraction decreases rapidly from the blow plate above. Also significant pressure drops typically occur down the tubes. By modeling at least part of the blow plate, the sand packing is known to be fairly compacted, the pressure is on the order of the blow pressure, and the boundary condition is placed several cells away from the location of the severe gradients.

For binder-coated sand core blowing, ***Arena-flow***® computes the motion of both the air and the sand, thus the full CPFD™ solver is required. As such, small cells must be removed from the final mesh to permit robust calculation of particle properties such as solid volume fraction. This is controlled automatically by the ***Arena-flow***® GUI.

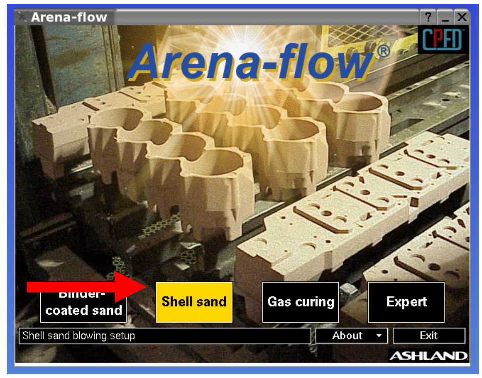
> **Note:** ***Arena-flow***® automatically selects the correct grid generation parameters for the process being modeled. Please only change these settings if advised to do so by an ***Arena-flow***® support engineer.



Users can view the grid generator settings by clicking on "Advanced options" in the "Grid Generator Setup" window. This raises the "Grid generator advanced options" window. Notice that by default small cells are removed. Please only change these settings if advised to do so by ***Arena-flow***® support engineers. Different settings may result in grids that look nicer, but may be less robust and can reduce the accuracy of the solution.

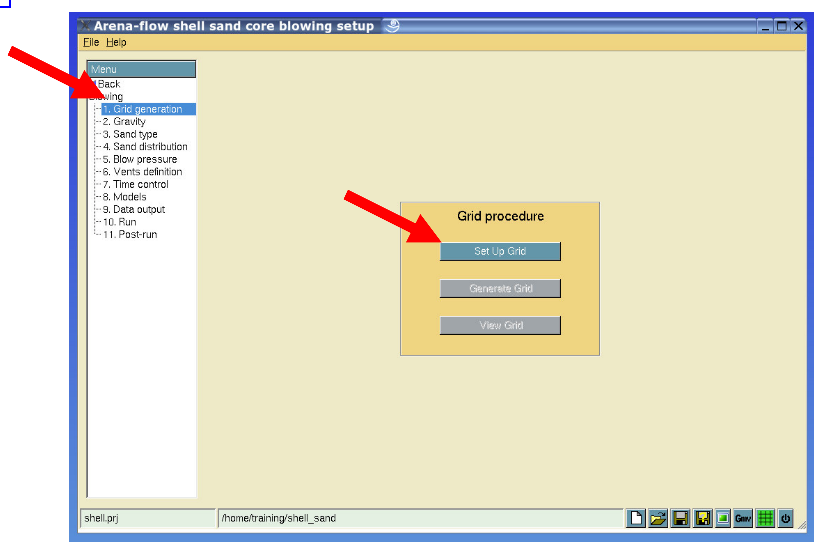
2.3.2 Shell sand grid generation

To model the blowing of a shell core with ***Arena-flow***®, click on "Shell sand" as shown.



Note: Many of ***Arena-flow***®'s modules have a similar look-and-feel. ***Arena-flow***® handles the process and sand-type differences in the solver. The GUI hides this complexity from the user.

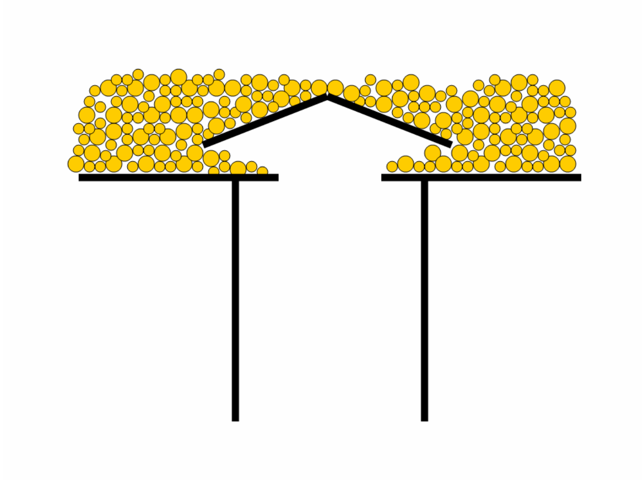
Next enter a project file name and working directory to launch the "***Arena-flow***® shell sand core blowing setup" window. Notice that much of the setup is identical to the "Binder-coated sand" module. ***Arena-flow***® simply handles the differences in sand types in the numerical implementation of the CPFD™ solver. The GUI hides this complexity from the user.



As with the other modules, click on "Set Up Grid" to launch the "Grid Generator Setup" window. The rest of the grid generation functionality is the same as with other modules.

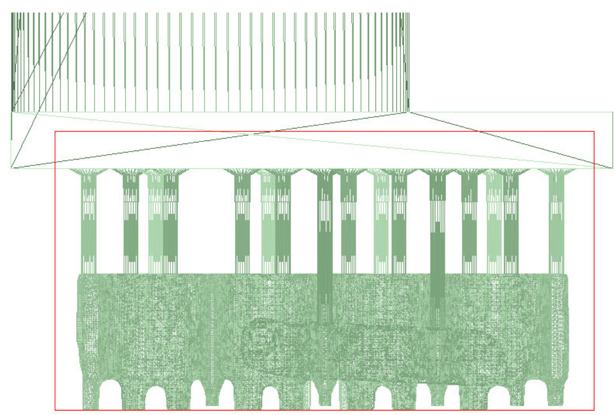
Often the blow tubes used in the shell process contain devices to stop the flow of sand when the core is removed. These devices are typically too small to be included in ***Arena-flow***® models. Out of necessity, these can be neglected. As a result, calculations of shell sand core blowing should not include the full magazine.

> **Note:** Due to internal structures in the blow tubes used for the blowing of shell cores and molds, calculations of shell sand core blowing should not include the full magazine.



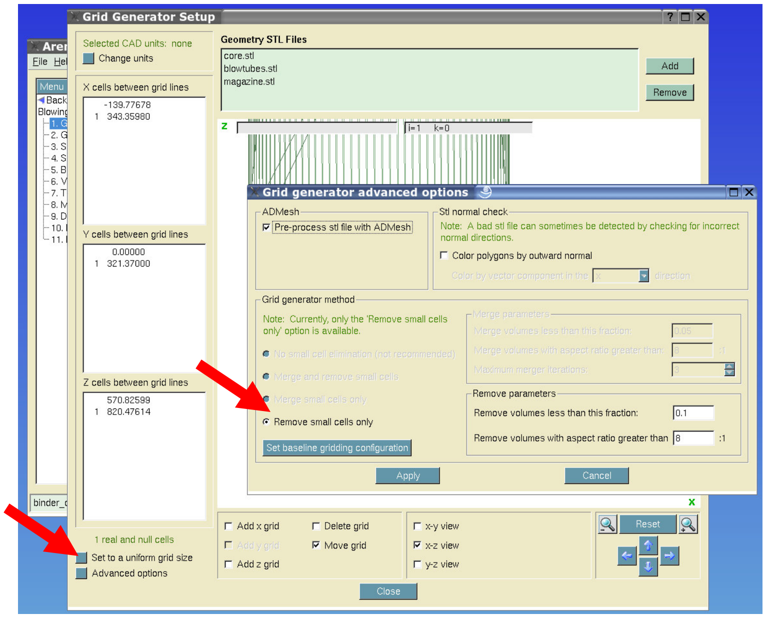
It is still very important to include a portion of the blow plate in the model when simulating the blowing of shell cores and molds. This locates the boundary condition away from locations of strong gradients.

> **Important:** Never start your core blowing model at the top of the blow tubes. Always include at least a portion of the blow plate.



The absence of the flow restriction device will alter the sand / air ratio for flow down the tubes, which may result in a discrepancy between calculated and observed filling times. However, the pattern of the filling of the tooling is what is most useful when determining the optimal fill of the core, not the time.

As with the binder-coated sand core blowing module, the shell sand module computes the motion of both the air and the sand, thus the full CPFD^{TM} solver is required. As such, small cells must be removed from the final mesh to permit the robust calculation of particle properties such as solid volume fraction. This is controlled automatically by the ***Arena-flow***® GUI.



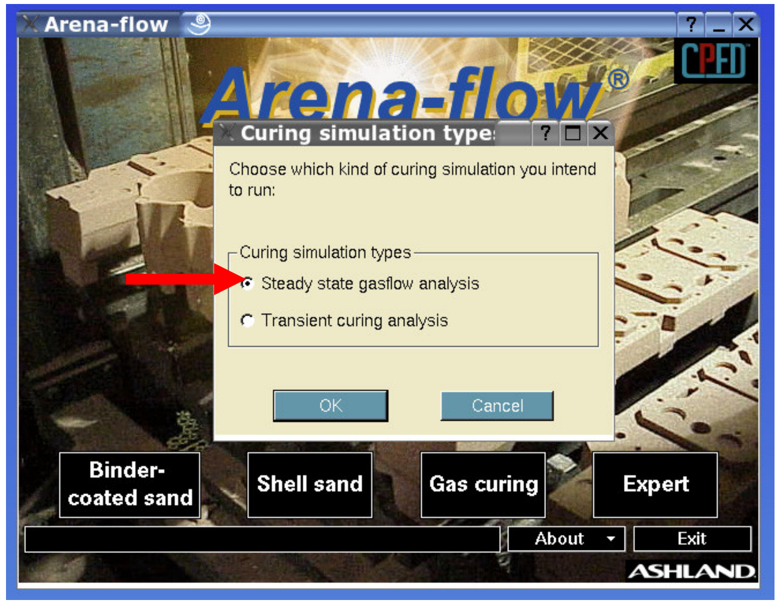
> **Note:** ***Arena-flow***® automatically selects the correct grid generation parameters for the process being modeled. Please only change these settings if advised to do so by an ***Arena-flow***® support engineer.

Users can view the grid generator settings by clicking on “Advanced options” in the “Grid Generator Setup” window. This raises the “Grid generator advanced options” window. Notice that by default small cells are removed. Please only change these settings if advised to do so by ***Arena-flow***® support engineers. Different settings may result in grids that look nicer, but may be less robust and can reduce the accuracy of the solution.

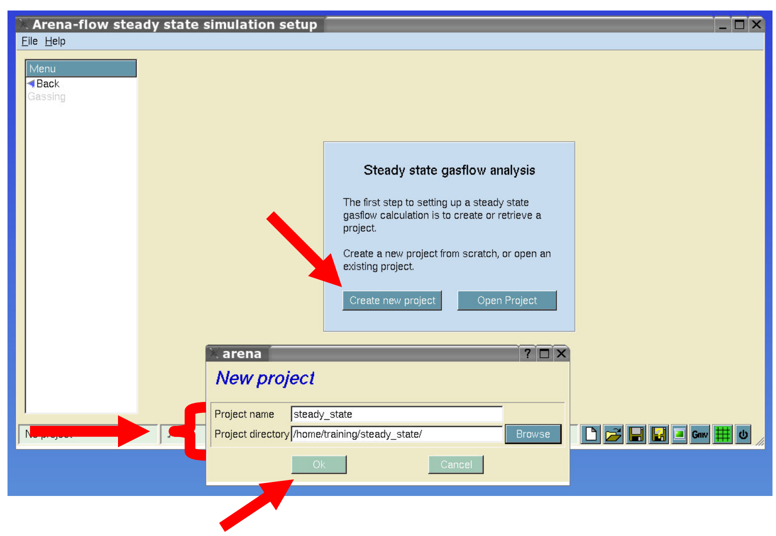
2.3.3 Steady-state gas flow grid generation

To model the steady-state gas flow through a filled core with ***Arena-flow***®, click on “Gas curing” as shown.

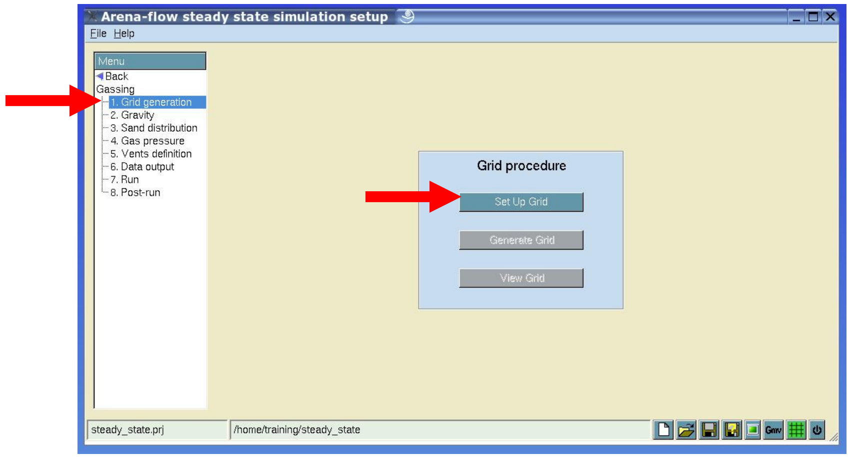
"Gas curing" actually contains two separate ***Arena-flow***® modules: "Steady state gasflow analysis" and "Transient curing analysis".



Select "Steady state gasflow analysis" to raise the "***Arena-flow***® steady state simulation setup" window.



Enter a project file and working directory to launch the interface. Once again, many of the setup steps are similar to those in other modules. ***Arena-flow***® manages the differences in simulation type in the CPFD™ solver. The GUI hides this complexity from the user.

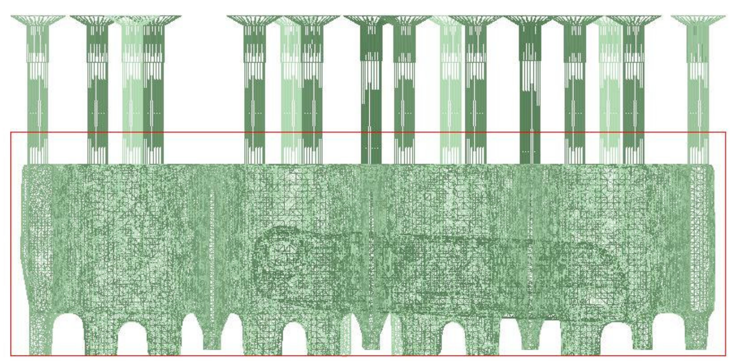


Click on “Set Up Grid” to launch the “Grid Generator Setup” window. The rest of the grid generation functionality is the same as with other modules.

When modeling the gas flow through the filled core, a pressurized source is typically applied through the gassing head, which replaces the blow tubes during the gassing cycle. For some core boxes, the gas may enter through the cope vents as well.

When creating the grids for the gas flow process, only the core and part of the gassing head (blow tube cad) are required as shown below.

Tip: For steady-state gas flow calculations, the blow tube locations are a good place to begin mesh definition.

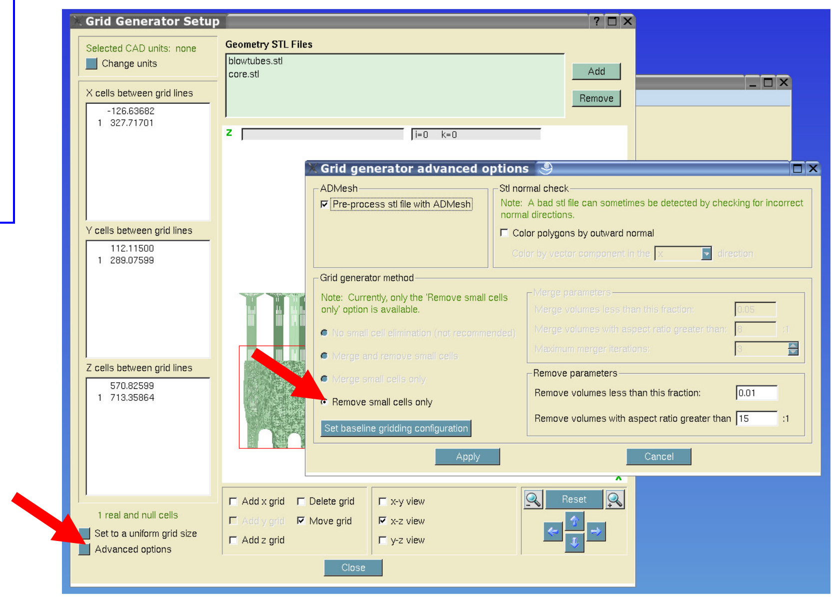


For steady-state gas flow calculations, the pressure is uniform across all tubes, thus this is a good choice of boundary location.

When modeling the steady-state air flow through the core the sand does not move. ***Arena-flow***® uses a very fast porous media solver for these calculations resulting in short simulation times, often on the order

> **Note:** ***Arena-flow***® automatically uses different solver and grid generation settings for steady-state gas flow and transient curing calculations, resulting in very fast computation times.

of minutes. Since the individual sand grain motion is not modeled, different mesh requirements exist. The ***Arena-flow***® GUI sets this automatically.

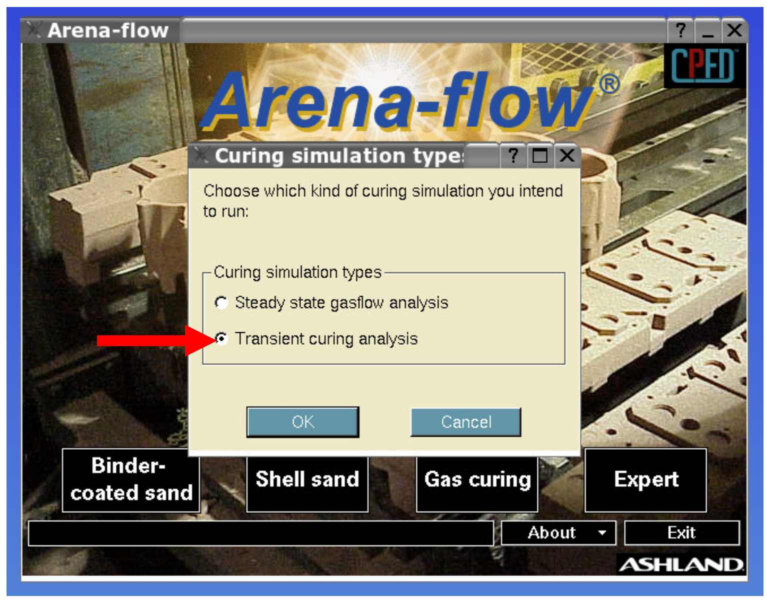


Users can view the grid generator settings by clicking on “Advanced options” in the “Grid Generator Setup” window. This raises the “Grid generator advanced options” window. Some small cells are still removed, however the different settings result in much smoother grids than those creating with any of the ***Arena-flow***® core blowing modules. Please only change these settings if advised to do so by ***Arena-flow***® support engineers.

2.3.4 Transient curing grid generation

To model the transient motion of the curing front through a filled core with ***Arena-flow***®, click on “Gas curing” as shown.

Select “Transient analysis” to raise the “***Arena-flow***® transient curing simulation setup” window

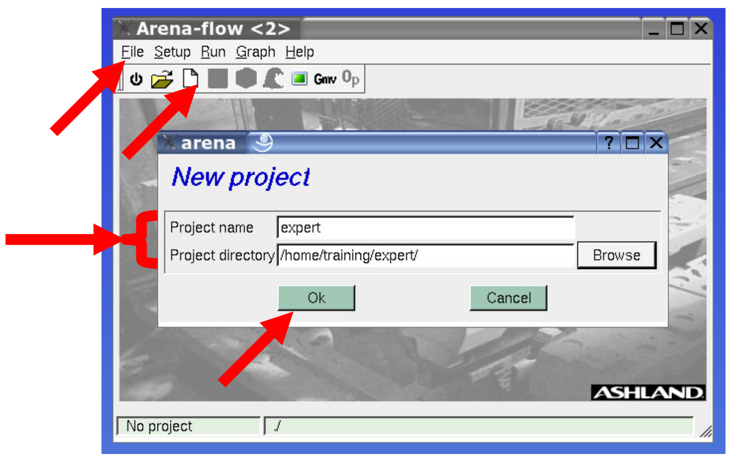


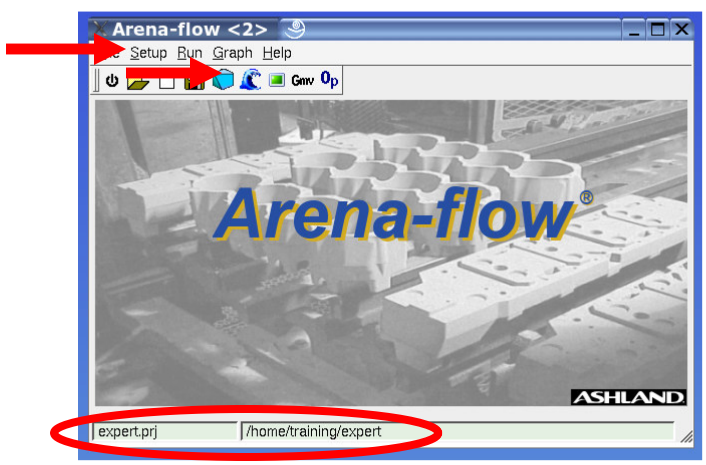
Transient curing models utilize similar grids as those used for steady-state gas flow calculations. The mesh typically encompasses the core and part of the blow tubes. A transient version of the fast, porous media solver is used, and ***Arena-flow***® sets all the solver and mesh parameters for the user.

2.3.5 Expert grid generation

To launch the ***Arena-flow***® expert module, click on “Expert” as shown.

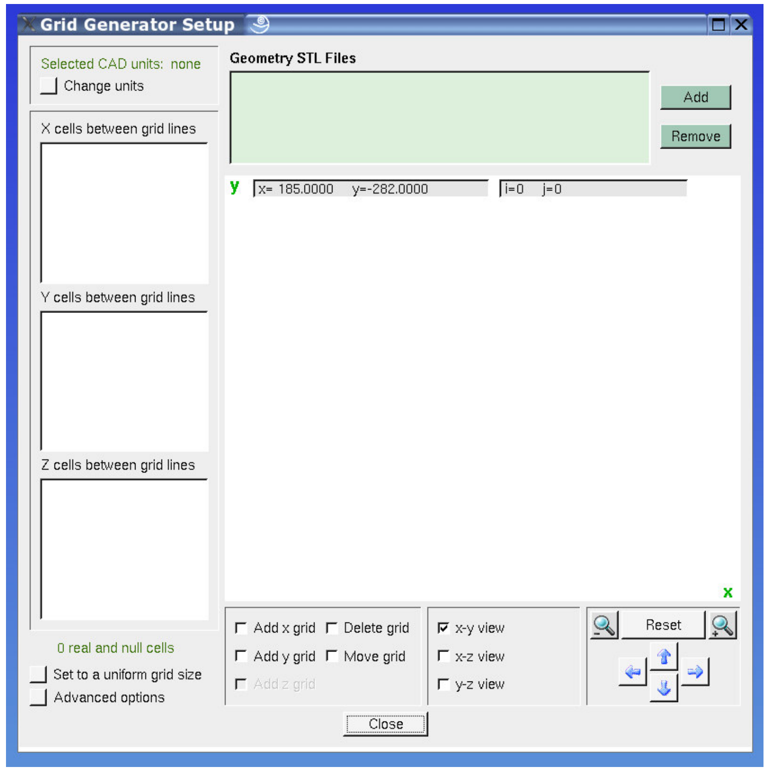
This raises the ***Arena-flow***® expert interface. To create a project file, click on "File", "New project" or click on the 🗋 button. Enter a project file name and working directory.





Notice the project file name and working directory are displayed across the bottom of your screen. Always ensure you are working with the correct project file and in the desired directory.

To launch the “Grid Generator Setup” window, click on “Setup”, “Grid”, or click on the  button.



In contrast to the process specific modules, the expert interface does not assume your intended use. Thus the grid generator defaults may not be set correctly for your process. The advanced options are defaulted to the core blowing settings. Please only use the expert interface after consulting with ***Arena-flow***® support staff.

Note: Only use the expert interface after consulting with ***Arena-flow***® support staff.

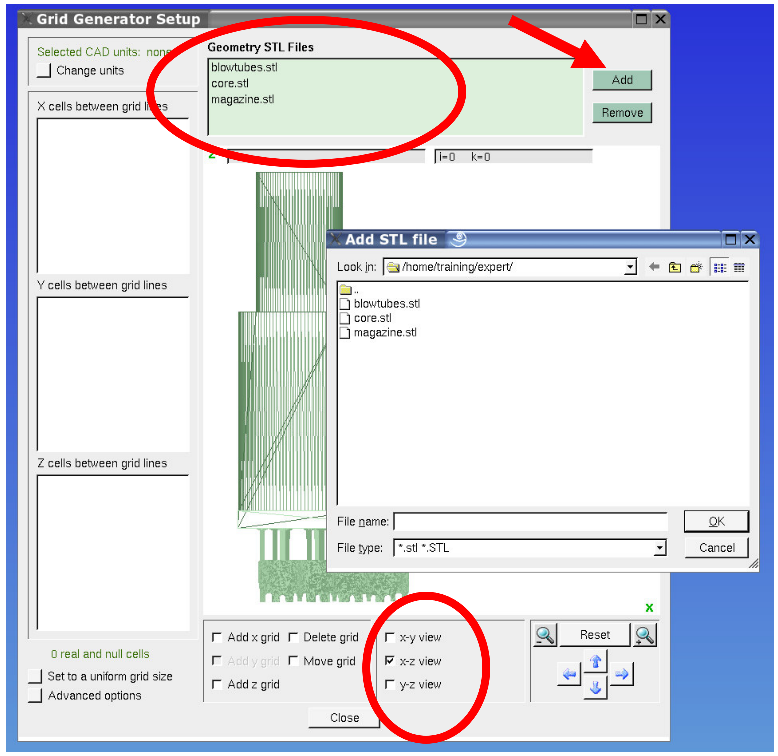
2.3.6 Generating the grid

Regardless of the module used, the grid generation activity is identical. The grid generation process consists of:

- entering the STL files
- setting the STL file units
- defining a Cartesian mesh

- generating the grid
- evaluating the grid

To enter the STL files, click on “Add” to raise the “Add STL file” window.



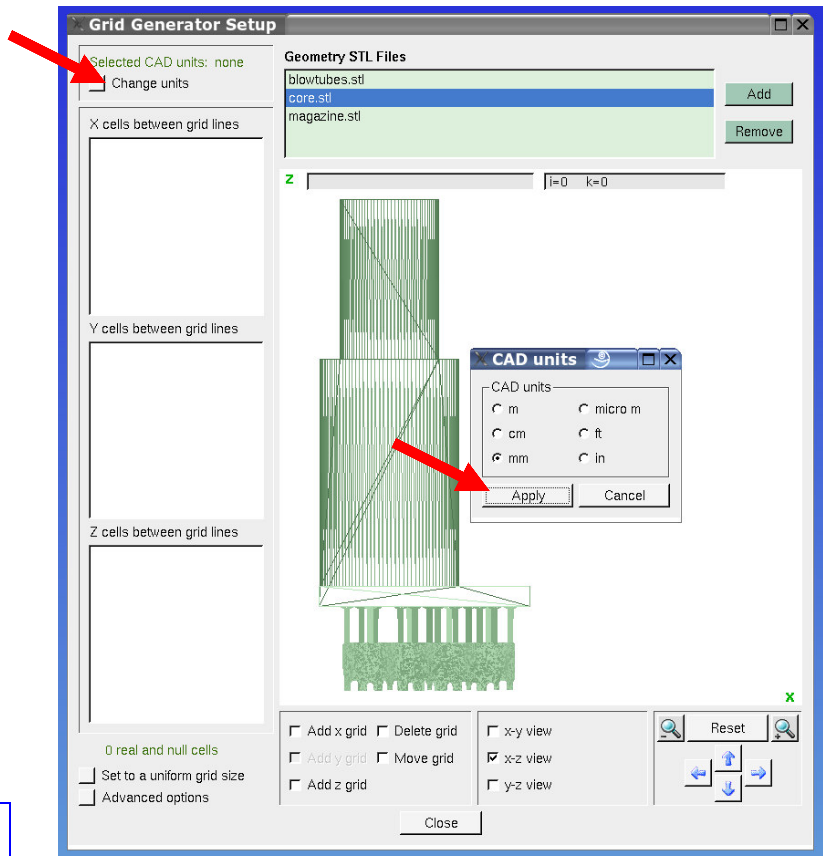
In the “Add STL file” window, select the STL files you wish to include in your simulation. Multiple files may be used, but each must be selected and added individually. Once added, the file names are visible in the “Geometry STL Files” section of the “Grid Generator Setup” window.

Tip: If multiple STL files are used, ensure they share a common origin and check this visually in the “Grid Generator Setup” window.

Once the files are added, the STL triangles will appear in green in your display window. The STL files must share a common origin. Always check to ensure your CAD fits together in the correct orientation. It is usually easier to work with CAD when the z-axis is pointed up, such that the gravity vector points in the –z axis. Check the view in the three orthogonal planes by clicking on the desired view (“x-y view”, “x-z view” or “y-z view”) in the bottom of the “Grid Generator Setup” window.

> **Note:** All ***Arena-flow***® input is in SI units, unless specified otherwise. The CAD is one exception – you must define the units used in the STL file.

Once the STL files are entered, the units must be selected. Generally, all inputs for ***Arena-flow***® calculations are in SI units (meters, kilograms, seconds), unless specified to be otherwise. However, many different companies have different standards for CAD output. To accommodate that, ***Arena-flow***® allows you to set the STL units. Click on "Change units", enter the desired "CAD units" and click "Apply".

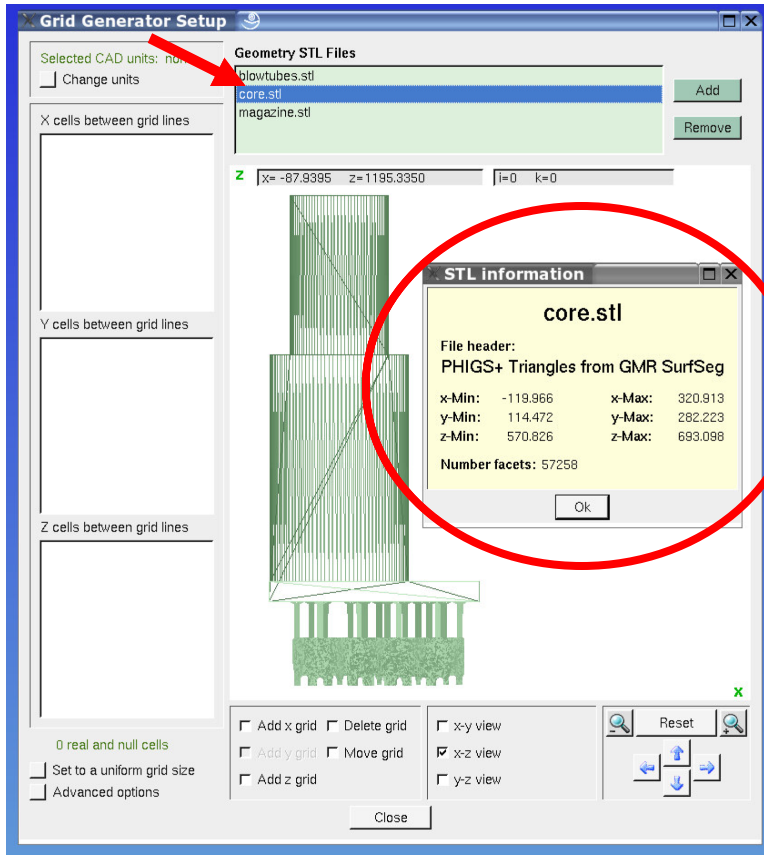


> **Note:** All ***Arena-flow***® output is in SI units, unless specified otherwise.

> **Tip:** If multiple STL files are used, ensure they are created using the same units.

This will cause a units conversion to take place during grid generation. Although the CAD may be entered in various units, the ***Arena-flow***® solver will operate in SI units. All ***Arena-flow***® output will also be in SI units, unless specified to be otherwise. If multiple STL files are used, they all must share the same origin and units scaling.

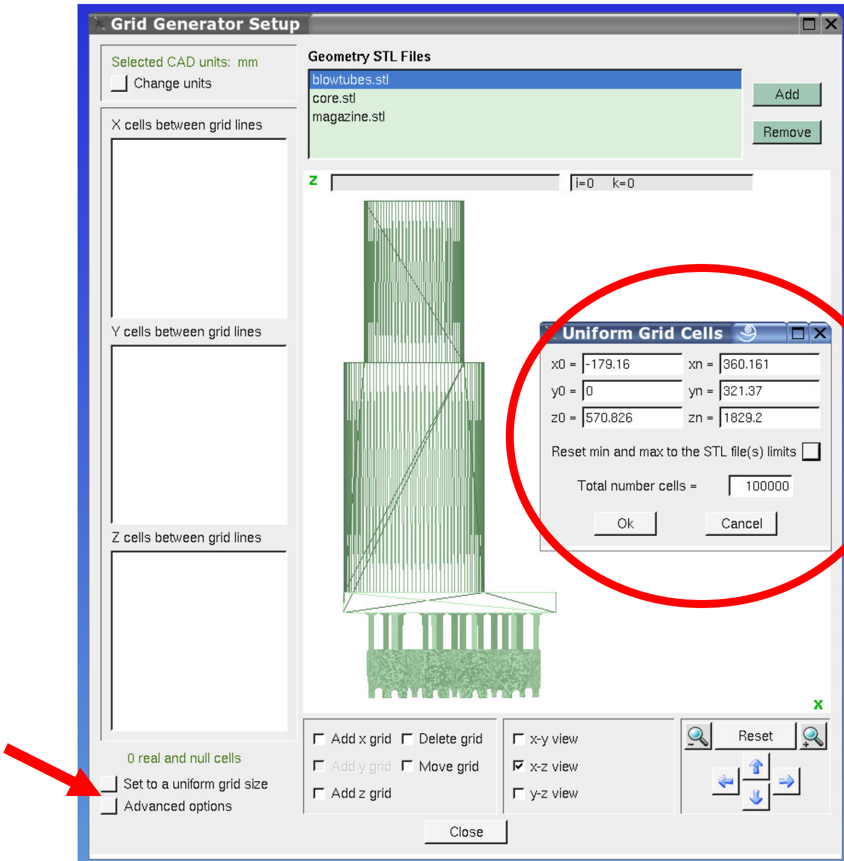
If you are unsure of the CAD units, check with your CAD operator or supplier. Also use common sense – consider what units would be reasonable for the dimensions of the problem. The limits of the STL files can be viewed by double-clicking on the filename in the “Geometry STL Files” region as shown below.

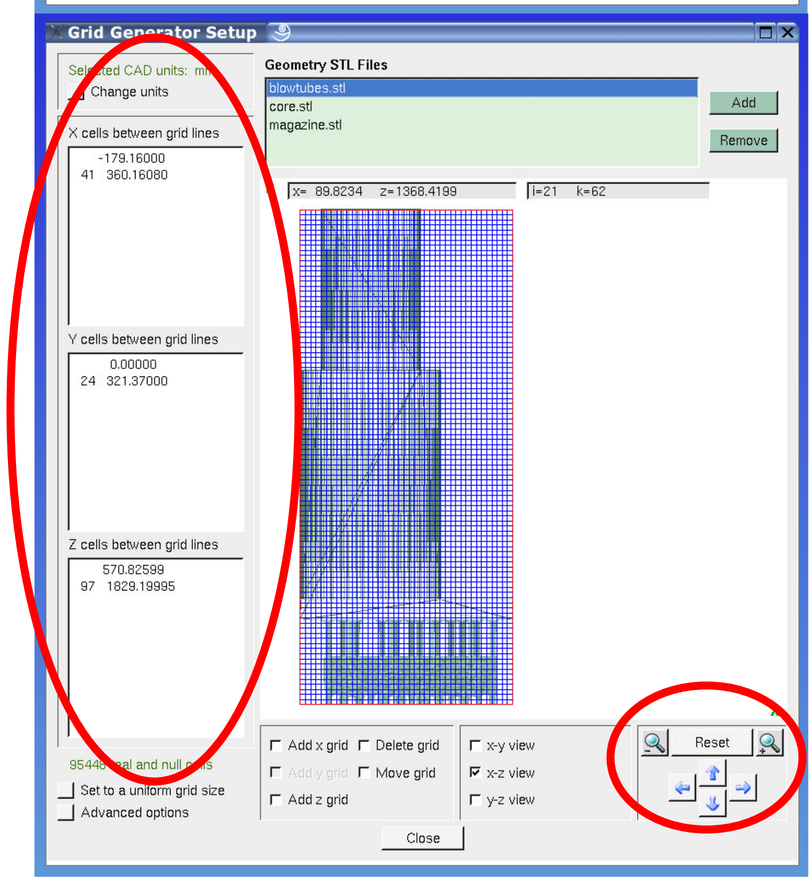


Tip: Always ask yourself whether your units are reasonable, or not!

In this example the core.stl file ranges from -119.996 to +320.913 in x. Only the units of mm would make sense physically, considering typical sizes of cores.

Once the CAD is entered and units are selected, it is time to define a Cartesian mesh. The easiest way to do this is to use the “Set to a uniform grid size” option. This raises the “Uniform Grid Cells” window. Select the range of the grid by setting the “x0”, “xn”, “y0”, “yn”, “z0” and “zn” values, where these represent the minimum and maximum values of the grid lines in the x-, y- and z-directions respectively. These values are defaulted to the extent of the STL files. Enter a total number of cells and click “OK”.


Grid Generator Setup
Selected CAD units: mm
Change units
Geometry STL Files
blowtubes.stl
core.stl
magazine.stl
Add
Remove
X cells between grid lines
Y cells between grid lines
Z cells between grid lines
i=0 k=0
Uniform Grid Cells
x0 = -179.16
xn = 360.161
y0 = 0
yn = 321.37
z0 = 570.826
zn = 1829.2
Reset min and max to the STL file(s) limits
Total number cells = 100000
Ok
Cancel
0 real and null cells
Set to a uniform grid size
Advanced options
Add x grid
Delete grid
Add y grid
Move grid
Add z grid
x-y view
x-z view
y-z view
Reset
Close


Grid Generator Setup
Change units
Geometry STL Files
blowtubes.stl
core.stl
magazine.stl
Add
Remove
X cells between grid lines
-179.16000
41 360.16080
Y cells between grid lines
0.00000
24 321.37000
Z cells between grid lines
570.82599
97 1829.19995
x= 89.8234 z=1368.4199
i=21 k=62
Set to a uniform grid size
Advanced options
Add x grid
Delete grid
Add y grid
Move grid
Add z grid
x-y view
x-z view
y-z view
Reset
Close

The grid lines appear in red and blue. The red lines are "specified" grid lines. The values of these appear in the right portion of the boxes on the left side of the "Grid Generator Setup" window labeled "X cells between grid lines", "Y cells between grid lines" and "Z cells between grid lines". The blue lines are defined by the number of cells between the "specified" grid lines, indicated by the numbers in the left portion of these boxes.

The Cartesian mesh may also be defined by

- entering numbers in the boxes directly
- using the "Add x grid", "Add y grid", "Add z grid", "Delete grid" and "Move grid" buttons at the bottom of the "Grid Generator Setup" window

Zoom and translation can be accomplished with the buttons in the lower right portion of the "Grid Generator Setup" window, or with the mouse control. The mouse control functions as follows:

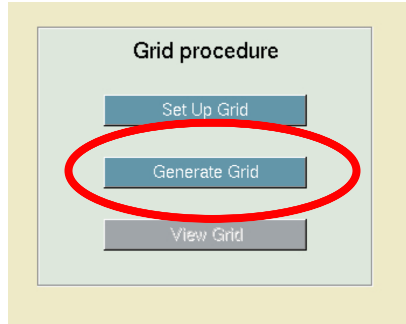
- zoom - right button
- translate - middle button

Note: ***Arena-flow***® training courses are held frequently, and contain further instruction on proper mesh definition.

More information regarding proper mesh definition practices can be learned at various ***Arena-flow***® training courses. Contact your sales representative or support engineer for more information regarding course content and schedule.

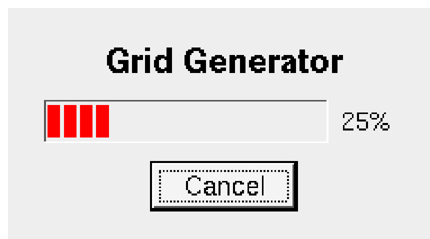
Once you have input your CAD, defined your units and specified your Cartesian mesh, you are ready to run the grid generator. The ***Arena-flow***® grid generator cuts the Cartesian mesh to the STL limits, removes small and problematic cells and writes the output necessary for post-processing.

To run the grid generator from any of the process-specific modules, close the "Grid Generator Setup" window and click on "Generate grid" in the "Grid procedure" menu. This option is made available once a grid has been defined. To start the grid generator from the expert interface, click on "Run", "Generate grid".



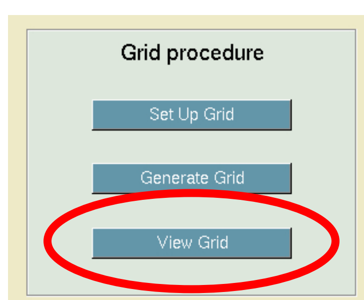
A progress bar will appear to keep you informed of the grid generator progress. Please note that the progress bar is not linear in time, but rather in steps through the grid generation algorithm. Often the grid generator will take 50% of the time to go from 90% to 100% completion. This is perfectly normal.

Note: The ***Arena-flow***® grid generator progress bar is not linear in time. Please be patient. Many grids can be generated in one minute or less.



Typical ***Arena-flow***® grid generation times range from 5 seconds to 5 minutes, depending on your cell count, STL complexity and computational resources. Many grids can be generated in one minute or less.

Once your grid is complete, it is important to evaluate its quality. Open the General Mesh Viewer (GMV) by clicking on "View Grid" from the "Grid Procedure" window of any process-specific module. This is made available once the grid has been generated. To view the grid from the expert interface, click on "Setup", "Show grid". ***Arena-flow***® 6.2 contains additional, automated grid viewing tools.



Important: Always check the quality of the grid before running the solver.
- Does the grid match the CAD?
- Are there any holes in the model?
- Are your cell counts reasonable?
- Are the cell size variations reasonable?

When viewing the grid, always check the following:
- Are there any holes in the model?
- Does the grid look reasonable?
- How does the grid compare to the CAD?

In addition, please check that the mesh adheres to the following guidelines:
- Cell counts should be
  - less than 200,000 cells for core blowing calculations (50,000 – 100,000 recommended)
  - less than 1,000,000 cells for steady-state gas flow calculations (100,000 – 500,000 recommended)
  - less than 500,000 cells for transient curing calculations (50,000 – 250,000 recommended)
- Make sure your cells are of a fairly uniform size and aspect ratio. This is more important for CPFD™ solvers than for CFD solvers. Ensure that the length of your largest Cartesian cell in any linear direction (x, y, or z) is no more than 2 or 2.5 times the length of your smallest Cartesian cell in the same direction.

Important: Never run the grid generator after starting your calculation – your results may be lost!

Once your grid is generated you may set up and run your calculations as described in the following chapters. It is important that you do not run your grid generator after starting your solver – you will lose your results. If you do this, please contact an ***Arena-flow***® support engineer immediately – there may be a way to recover your results.

Note: In ***Arena-flow***® there exist sensitivities to two types of discretization – grid and particle. The particle resolution is handled automatically by the GUI.

2.3.7 CPFD™ grid resolution

No discussion of gridding is complete without consideration of the sensitivities involved. With a CPFD™ solver such as ***Arena-flow***®, there are two levels of discretization sensitivities which must be considered – one for the fluid and another for the particles. The resolution of the sand particles, namely how many sand grains exist in the core, and what representative number are included in the model, has been the topic of much proprietary, unpublished research. The findings have been incorporated into the ***Arena-flow***® GUI, allowing users to simply set a "low", "medium" or "high" level of initial computational particles per cell. The geometric resolution is left up to the user during grid definition and generation.

When defining a mesh for an ***Arena-flow***® calculation, the blow tubes are often the limiting geometric feature. This section summarizes a study considering grid resolution effects related to blow tube meshing[17].

A series of test cases was constructed as shown in Figure 11. The geometry of each case is identical, and the same sand, venting and blow pressure was applied to each.

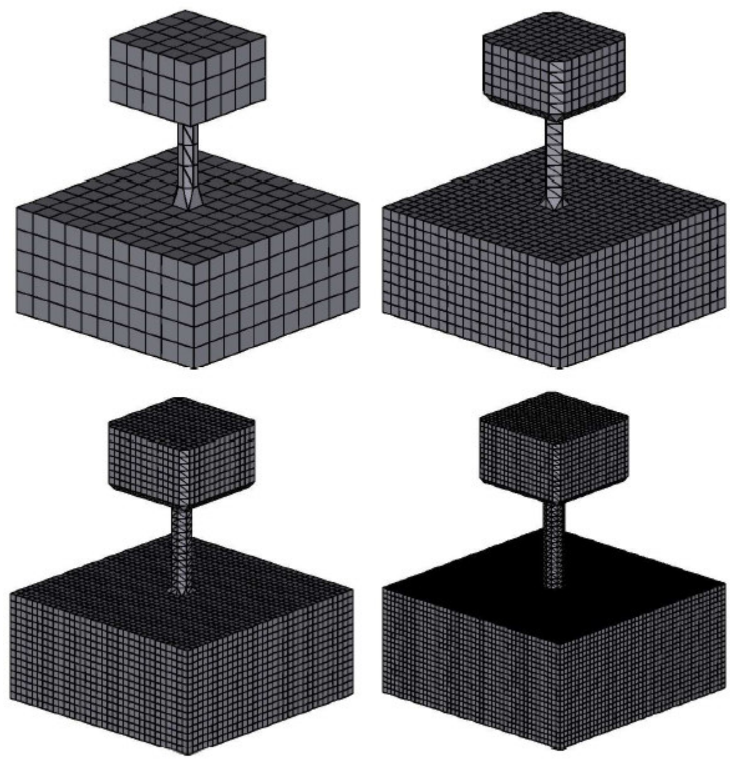


Figure 11: Grid resolution test cases with 1, 2, 3 and 4 cells across the blow tube

Figure 12 shows a close-up of the blow tube mesh. The total number of computational sand grains was held constant.

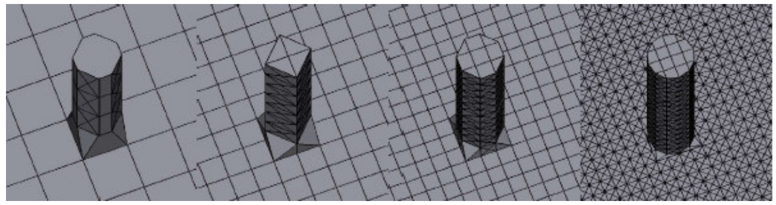


Figure 12: Blow tubes resolved with 1, 2, 3 and 4 cells across the diameter

Note: The largest difference in the computed flow rates deviates from the mean by less than 10%, even though the number of cells was varied by two orders of magnitude!

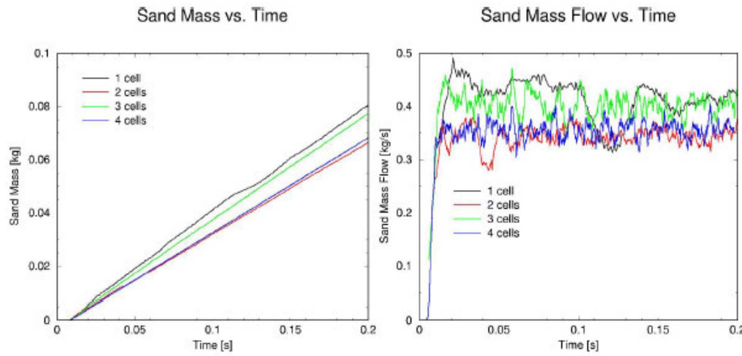


Figure 13: Sand mass flow and flow rates

Figure 13 shows the sand mass flow and sand mass flow rates through the blow tubes vs. time for the four test cases. A maximum solids flow rate is observed when the blow tube is modeled as a single cell and a minimum is seen for the two cell case. Note that the two cell case models much less of the actual blow tube cross-sectional area than the other cases. As the grid is further refined, the flow rates begin to converge. However, the computed flow rates deviate from the mean by less then 10%, even though the number of cells in the calculation was varied by two orders of magnitude[17].

These results show the relative grid insensitivity of ***Arena-flow***®. Since the CPFD™ method models both the fluid and granular phases, the accuracy of the method is not wholly determined by the solution of a single phase. Further, due to the Lagrangian nature of the particle solution, a sub-grid resolution is obtained.

The study also observed that the runtime varies by two orders of magnitude as do the cells[17] for a constant number of computational sand grains[17]. Thus, it is advantageous to run large calculations on the coarsest mesh feasible.

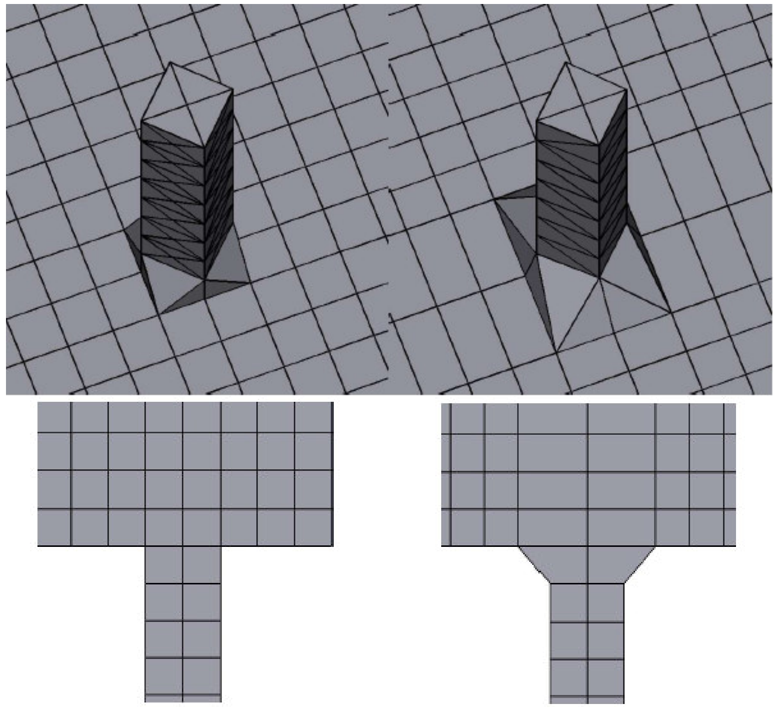


Figure 14: Effects of large are ratios on grid resolution

Important: Avoid large face area ratios between cells in the blow tubes and cells in the magazine above.

One limiting case was observed as shown in Figure 14. This shows a variation of the two-cell test case whereby the grid lines were moved away from the blow tube edges as shown in the images on the right. Here there is a large area ratio between the cell faces in the magazine and the opposite faces in the blow tubes.

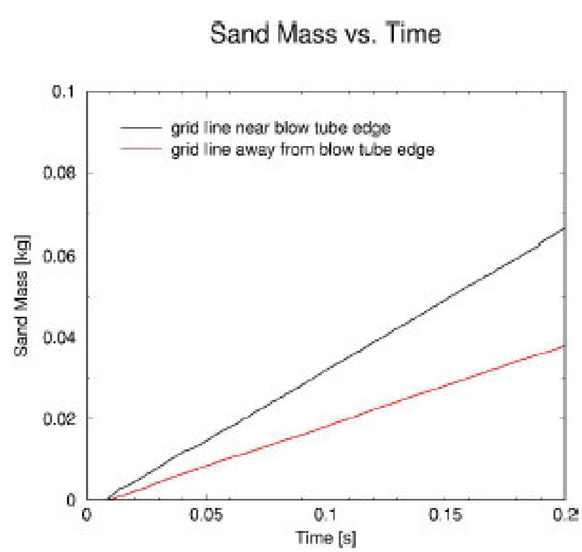


Figure 15: Flow rate reduction due to large face area ratio

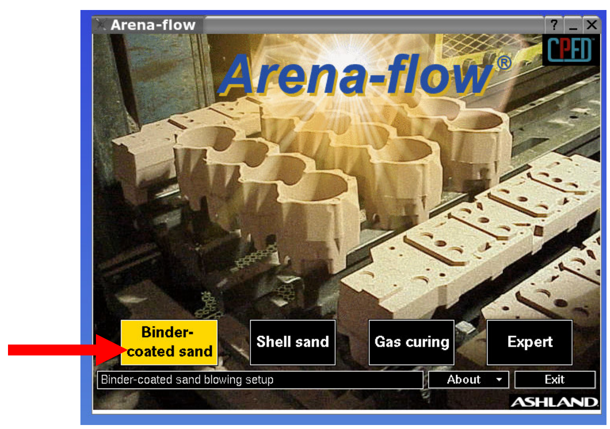
Figure 15 shows that there is a significant reduction in sand mass delivered through the blow tube when a large face area ratio is present between cells in the blow tubes, and cells in the magazine above. This effect is under further investigation. Users are to avoid this situation during grid specification.

2.4 Binder-Coated Sand Core Blowing

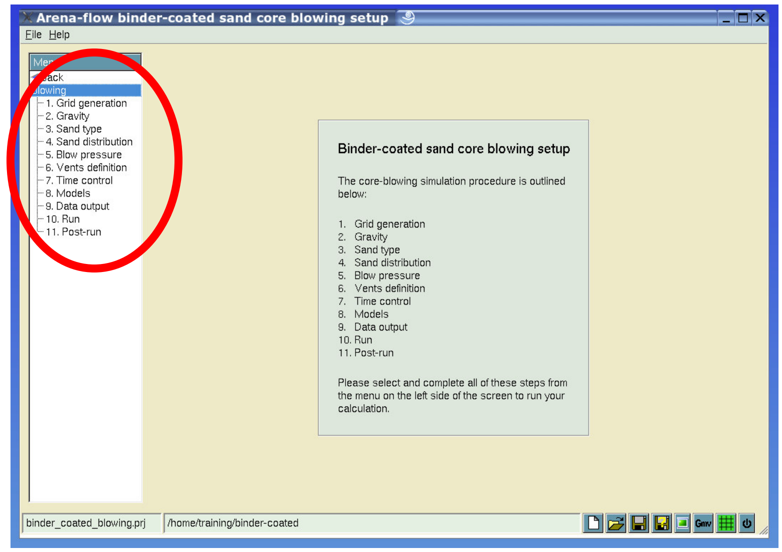
In this chapter you will learn about:

- *Setting up a binder-coated sand core blowing project*
- *Running a binder-coated sand core blowing project*
- *Evaluating the results from a binder-coated sand core blowing project*

To model the blowing of a cold-box core with ***Arena-flow***®, click on "Binder-coated sand" as shown:

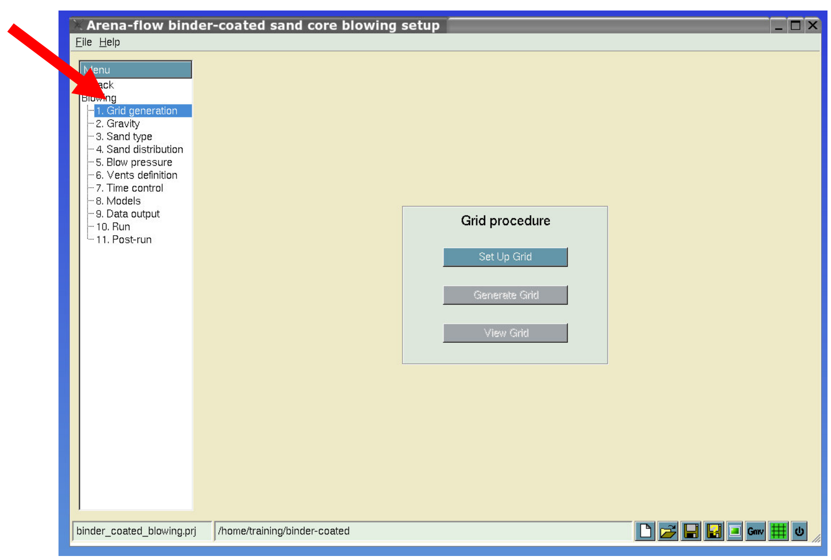


Create or open a project file in the desired working directory to launch the "***Arena-flow***® binder-coated sand core blowing setup" window. To set up and run a binder-coated sand core blowing project, work through the 11 menu items shown in the tree on the left of the window.

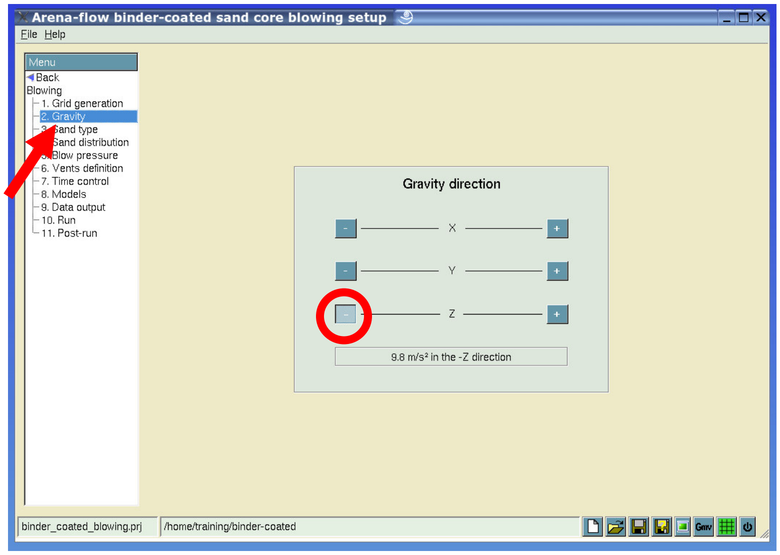


Tip: To set up and run a binder-coated sand core blowing project, work through the 11 menu items shown in the tree on the left of the window.

The first step to creating any ***Arena-flow***® model is mesh generation. Click on "Grid generation" to go to the "Grid procedure" page.

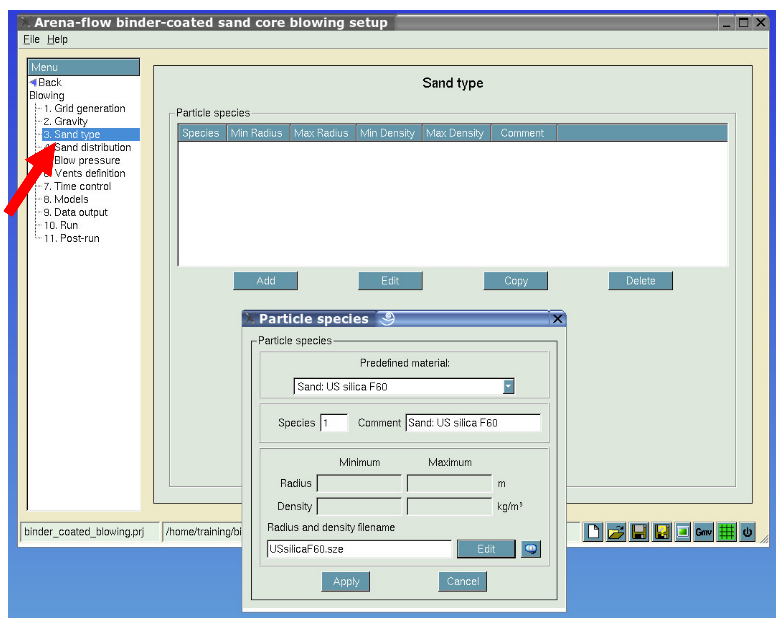


On the "Grid procedure" page, create and evaluate the grid by working through the three steps: "Set Up Grid", "Generate Grid" and "View Grid". The use of the ***Arena-flow***® grid generator for binder-coated sand core blowing calculations is outlined in section 2.3.1 of this User Guide. Proper grid generation techniques are presented in the various ***Arena-flow***® training classes. Classes are held regularly. Contact your sales representative or support engineer for more information regarding course content and schedule.



Click on the next menu item, "Gravity". This is where you tell ***Arena-flow***® the orientation of your CAD, relative to gravity. In this example, the CAD is oriented such that the z-axis is up. Thus, the gravity vector should point in the –z direction.

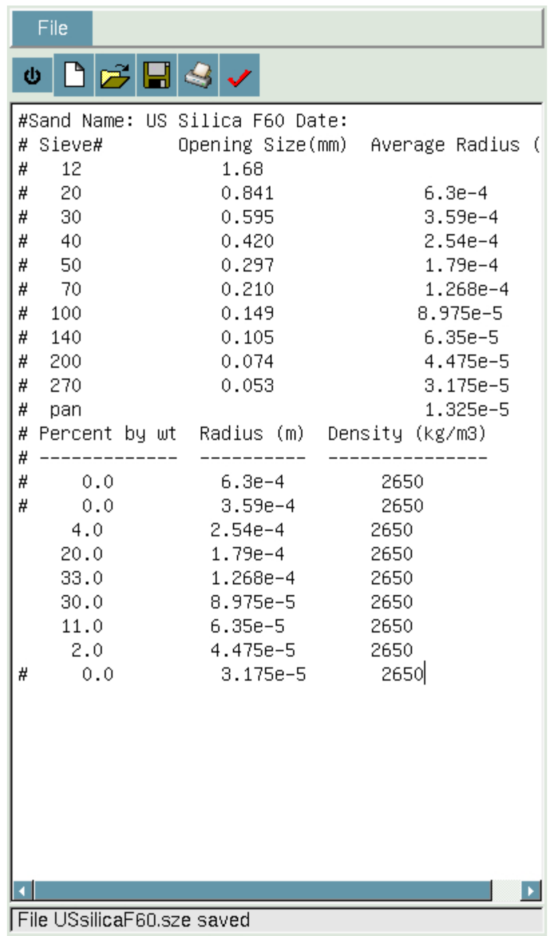
Note: The "Sand type" page defines the sands to be used in the calculation. The "Sand distribution" page defines where the sand is located (or distributed) at the start of the calculation.



Next, click on "Sand type". This is where you tell ***Arena-flow***® what sand will be modeled in the calculation. You may define multiple sand types. These may be combined to simulate a sand mixture or may be layered in the magazine to simulate colored sand experiments. The location of the sand is entered on the next page; you simply define the various sand species here.

To define a sand type for use in the calculation, click on “Add”. This raises the “Particle species” window. There are three ways to enter a sand type:

- Select a default sand type from the “Predefined material” drop-down menu
- Enter a minimum and maximum sand grain radius and density.
- Enter a custom size distribution from a sieve analysis by clicking “Edit”




```
#Sand Name: US Silica F60 Date:
# Sieve#       Opening Size(mm)   Average Radius (
#   12              1.68
#   20              0.841              6.3e-4
#   30              0.595              3.59e-4
#   40              0.420              2.54e-4
#   50              0.297              1.79e-4
#   70              0.210              1.268e-4
#  100              0.149             8.975e-5
#  140              0.105              6.35e-5
#  200              0.074              4.475e-5
#  270              0.053              3.175e-5
#  pan                                 1.325e-5
# Percent by wt   Radius (m)   Density (kg/m3)
# -------------   ----------   ---------------
#      0.0          6.3e-4        2650
#      0.0          3.59e-4       2650
     4.0          2.54e-4       2650
    20.0          1.79e-4       2650
    33.0          1.268e-4      2650
    30.0          8.975e-5      2650
    11.0          6.35e-5       2650
     2.0          4.475e-5      2650
#      0.0          3.175e-5      2650
```

Note: The size distribution file contains three columns: percent, radius and density. The percent may be incremental or cumulative. Lines beginning with a “#” character are ignored.

A sample sand size distribution file is shown. Sand size distribution files contain the .sze extension. Once a size distribution file is created for your sand, you may use it repeatedly in successive calculations.

The format of the sand size distribution file is columnar, with the columns separated by “white

> **Tip:** Remember that all ***Arena-flow***® input is in SI units, unless specified to be otherwise. The radius is thus in meters. Be careful, it's radius – not diameter.

> **Tip:** Remember that the density required is the raw (material) density of the sand.

> **Note:** ***Arena-flow***® can model granular materials other than sand, such as ceramic beads and sand additives.

space" such as spaces and tabs. The "#" character indicates that the line is a comment. Lines beginning with the "#" character are ignored by ***Arena-flow***®. The three columns are a percentage of sand, the radius and density. Please note that all input is in SI units, thus the sand grain radius is in meters and the density is in kg/m^3.

The density is the material density of the sand used, not to be confused with the bulk density of a sand mixture. In this example, the refractory material is a silica sand which typically has a material density between 2650 and 2670 kg/m^3 [18].

In actuality, the "Sand type" in ***Arena-flow***® need not be a sand at all. ***Arena-flow***® can model various granular, refractory materials such as ceramic beads and sand additives, provided the size distribution and material density are known and properly defined.

Also note that ***Arena-flow***® accepts two ways of entering a custom sand size distribution file. The first is shown, above with the percentages being an incremental percentage of sand on each sieve. The second is a cumulative distribution where the percentage is a sum of the percentages up to the current bin. For example, the percentages above (0.0, 4.0, 20.0, 33.0, 30.0, 11.0, 2.0, 0.0) could be entered as 0.0, 4.0, 24.0, 57.0, 87.0, 98.0, 100.0). A cumulative size distribution is preferred. If a cumulative size distribution is defined, the first entry must be 0.0 and the last must be 100.0. It does not matter if the sand is defined in increasing or decreasing size order.

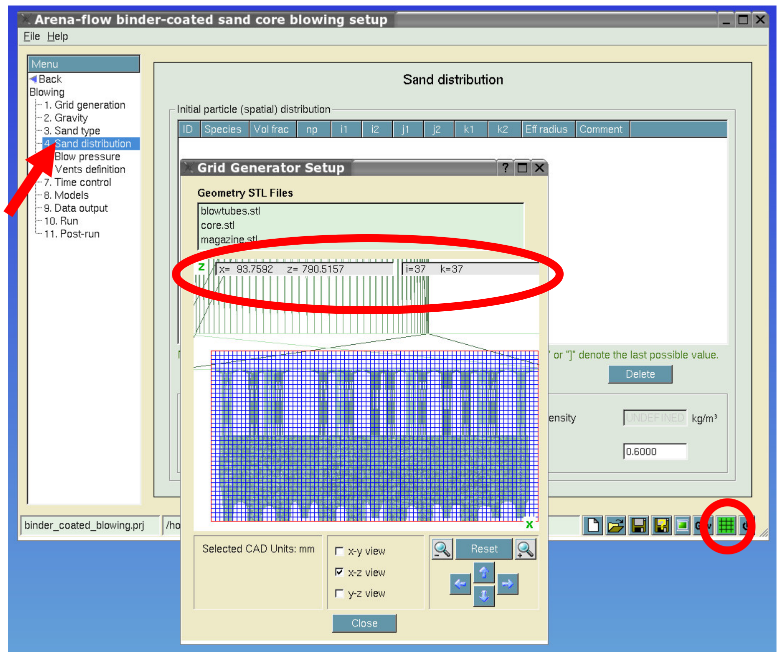
Once a sand type is defined using the "Add" button, it appears in the "Particle species" table on the "Sand type" page. This sand species can then be edited using the "Edit" button, reproduced using the "Copy" button or deleted using the "Delete" button.

Multiple sand types may be defined. If multiple sand types are used, each is identified with a unique species number. The results can then be colored by "Species" to show various sands as different colors. This is a very useful way to visualize the sand motion

Note: If multiple sand types are defined, each is identified with a unique species number. The results can be colored by species to aid in visualization.

through the system. If this is your intent, be sure to ask for “Species” to be included in the output files when using the “Mesh Viewer” window as described on “Data output” page of the GUI.

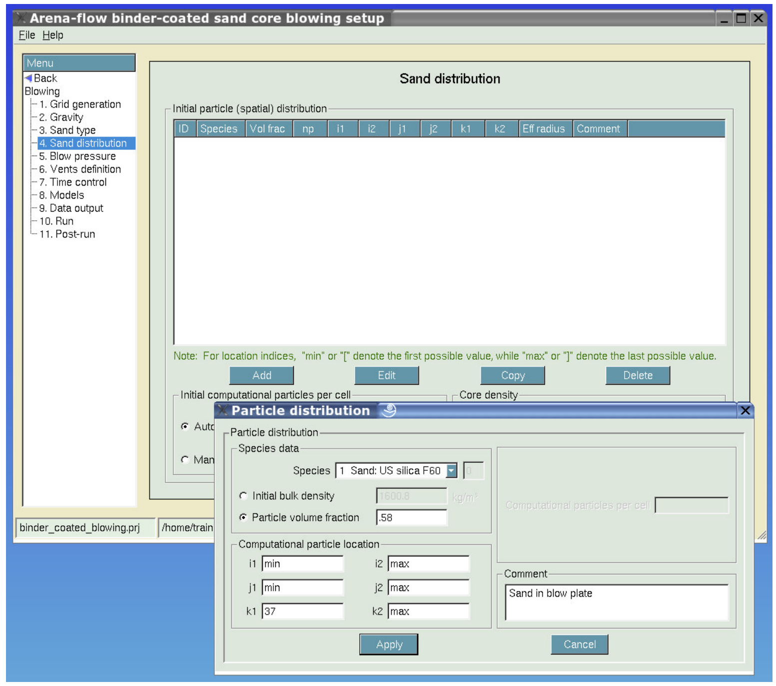
The sand is automatically assumed to be coated with a binder. Additional sand properties such as wall bounce, sand grain interaction and flowability are defaulted by the ***Arena-flow***® GUI.



After you define the sand to be used in the calculation, you must tell ***Arena-flow***® where you would like the sand to be at the start of the simulation. Click on “Sand distribution” to go to the “Sand distribution” page.

Tip: You can always refer to your grid by clicking on the reference grid button.

The location of the sand is related to the grid you defined in step 1. You may reference this grid by clicking on the button on the bottom right of the screen. This opens a non-editable version of the “Grid Generator Setup” screen. As you move your mouse over your grid, the coordinates (x,y,z) and cell indices (i, j, k) are displayed at the top of the window. Note that a two-dimensional plane is shown, thus only two of the three coordinates and grid indices are shown at one time. Determine which three-dimensional block of cells you wish to initialize with particles.



To place the sand in the cells, click on “Add” to raise the “Particle distribution” window. First select the “Species” from the drop-down list. You may select from any of the sands you defined in the “Sand type” page. If you used a comment when you defined your sand, this will also appear in the “Species” drop-down list.

Note: The “Initial bulk density” or “Particle volume fraction” must be less than or equal to the “Average core density” or “Close pack”.

Next enter either an “Initial bulk density” or “Particle volume fraction”. This is how much sand is to be placed in the cells. The “Initial bulk density” or “Particle volume fraction” must be less than or equal to the “Average core density” or “Close pack” entered on the “Sand distribution” page.

The “Average core density” or “Close pack” value controls how much sand is permitted into the cells in the core during the calculation. It is an average value, because the various physical models in ***Arena-flow***® will vary the packing from cell to cell to predict the density distribution in the filled core.

In general, it is good practice to define the “Initial bulk density” or “Particle volume fraction” to be slightly less compacted (say 2-5% less) than the “Average core density” or “Close pack”. When the core is filled, it is

Tip: It is good practice to define the "Initial bulk density" or "Particle volume fraction" to be slightly less compacted than the "Average core density" or "Close pack".

Tip: The "min" and "max" keywords can be used instead of the minimum and maximum cell or face indices whenever defining a boundary or initial condition.

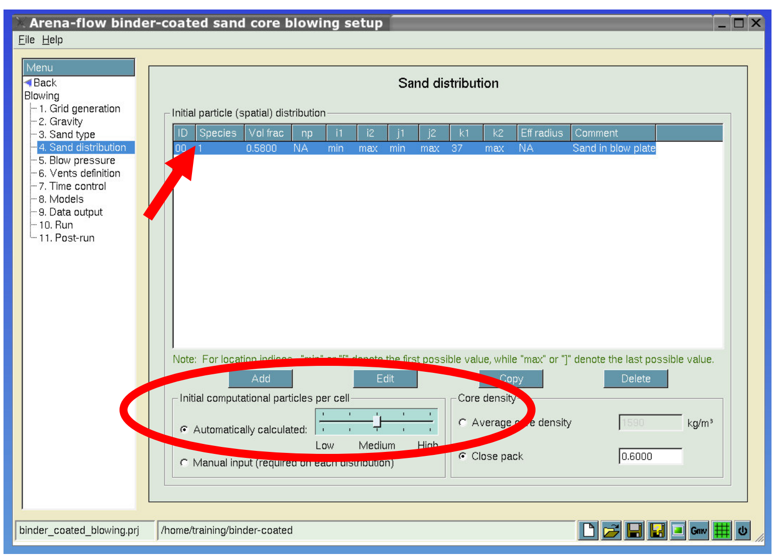
Tip: Enter comments whenever possible – it will save you time in the long run.

done so by a high pressure air source injecting the sand at high velocities. When the blow plate and magazine are filled, the sand is usually allowed to fall in under gravity. These different sand fill processes will likely result in different packing levels.

Next you must define the initial location of the sand. The "Computational particle location" asks for six values, corresponding to the minimum and maximum cell indices in the x, y and z directions defining the three-dimensional block of cells containing the sand. These cell indices can be determined by using the reference grid tool, , however, the keywords "min" and "max" can also be used to reference the minimum and maximum cell index in each respective direction.

Lastly you are permitted to enter a "Comment" for the distribution. This is not used by the solver, but can be extremely useful when setting up large projects.

Typically sand is initialized in the blow plate and magazine at the start of the calculation. Sometimes binder-coated sand remains in the blow tubes between successive blows. If you wish to model this phenomenon, you must tell ***Arena-flow***® by defining the proper sand distributions. Multiple distribution blocks may be used to define irregular regions, such as blow tubes of varying lengths. This is explained further in the training courses.



Once a sand distribution is defined using the “Add” button, it appears in the “Initial particle (spatial) distribution” table on the “Sand distribution” page. These distributions can then be edited using the “Edit” button, reproduced using the “Copy” button or deleted using the “Delete” button.

There could be hundreds of millions of individual sand grains in a core; to make the CPFD™ method efficient for practical, commercial applications, a lesser, representative number of computational particles is modeled. The number of computational particles must be large enough to be indicative of the overall core filling process, but not too large to exceed practical memory and runtime constraints.

Note: ***Arena-flow***® automatically sets the sand grain computational resolution for you.

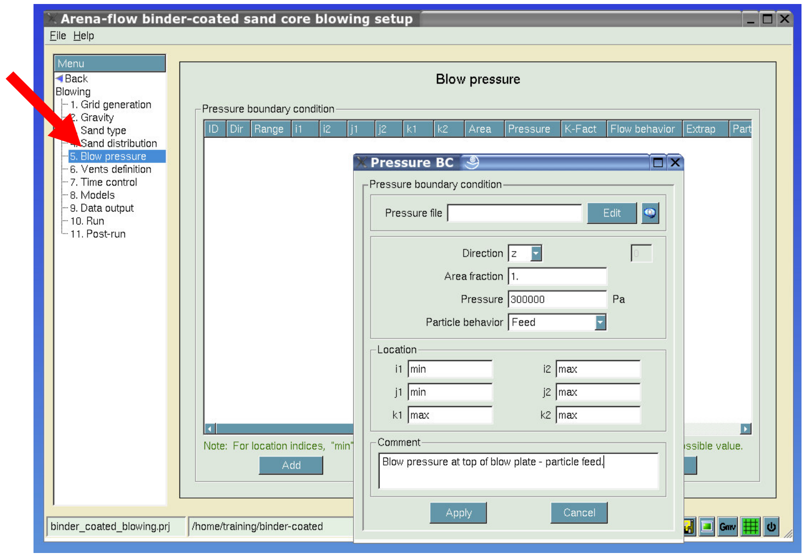
Arena-flow® automatically sets the sand grain computational resolution for you. You have control over the “Initial computational particles per cell” setting in the “Sand distribution” page via a slider bar. To use this, set the “Initial computational particles per cell” via the default, “Automatically calculated” option. Move the “Low / Medium / High” slider to the desired resolution. The default setting is “Medium”. For most commercial applications, a “Medium” or “Medium / High” setting is recommended.

Tip: Use the “Automatically calculated” “Medium” or “Medium / High” setting for “Initial computational particles per cell”.

Arena-flow® also has advanced capabilities to control the computational modeling of sand grains. This is controlled via the “Manual input” setting. Please only use the “Manual input” option if advised to do so by an ***Arena-flow***® support engineer. The “Automatically calculated” option works well for all typical, commercial applications.

After the “Sand distribution” is set, click on “Blow pressure” to go to the “Blow pressure” page. Here you tell ***Arena-flow***® what blow pressure will be used, whether the pressure is constant or time varying and where the pressure is to be applied.

Click on “Add” to raise the “Pressure BC” window.



Select the “Direction” from the drop-down menu. This tells ***Arena-flow***® upon which faces of the cells the pressure boundary condition is to be applied. If your CAD was designed with the z-axis pointing up and your blow pressure is applied at the top of the magazine (or if you are only modeling part of the blow plate), then the default directional value of “z” can be left unchanged.

Enter the “Area fraction”. This is the fraction of the cell faces which is open to the pressure. If the blow pressure is applied at the top of the magazine, leave this unchanged at 1.0. If the pressure is applied through a meshed sleeve, the “Area fraction” will be less than 1.0.

Tip: Many conversion factors to SI units can be found by clicking on “Help”, “Units Reference”.

Enter the “Pressure”. This is the blow pressure in Pa. To determine the appropriate conversion factor when converting from alternate pressure measurements to Pa, click on “Help”, “Units Reference”.

Important: The CFD component of ***Arena-flow***® is an incompressible solver; thus all pressures are gauge pressures with zero (0) representing the vented, atmospheric pressure.

Please note that the CFD component of ***Arena-flow***® is an incompressible solver. As such, all pressure measurements are relative. Calculations are typically set up with a zero (0) pressure at the vents. Thus the blow pressure should be in gauge pressure, relative to the vents.

Next specify the “Particle behavior”. If you are modeling the entire magazine, keep the default value of “No exit”. This actually means that no sand can

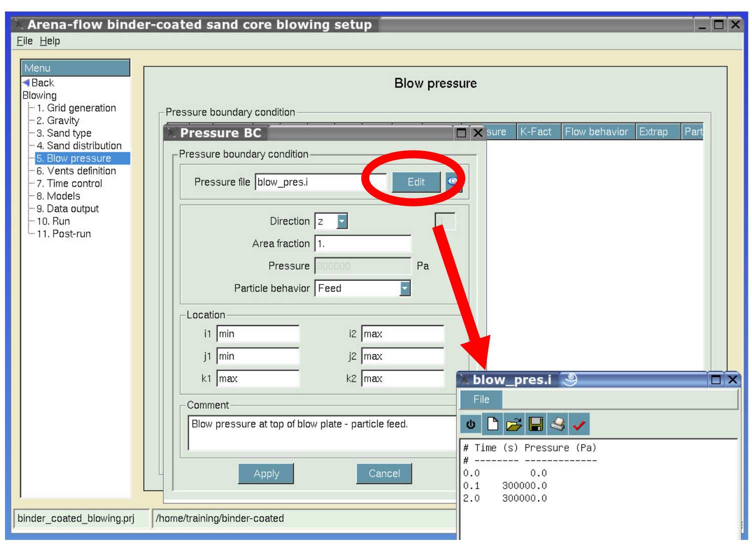
> **Tip:** If you are modeling the entire magazine set the "Particle behavior" to "No exit". If you are starting you model at the blow plate, set the "Particle behavior" to "Feed".

enter or leave the computational domain at the boundary. If you are starting your model at the blow plate, change the "Particle behavior" to "feed". This will permit more particles to enter the computational domain as others leave the cells next to the boundary, simulating a full magazine. ***Arena-flow***® calculations with "feed" activated tend to start quickly and slow down as they run, due to the increased number of computational sand grains as calculation progresses.

Next specify the location of the blow pressure boundary condition. These are boundary cell indices and they must define a plane. You may use the reference grid utility, , to help you determine proper values for the indices. Additionally, the "min" and "max" keywords can also be used to reference the minimum and maximum cell index in each respective direction.

Sometimes multiple planes are required to fully specify the blow pressure inlet location. These boundary conditions can be edited using the "Edit" button, reproduced using the "Copy" button or deleted using the "Delete" button.

Finally, you are permitted to enter a "Comment" for the distribution.



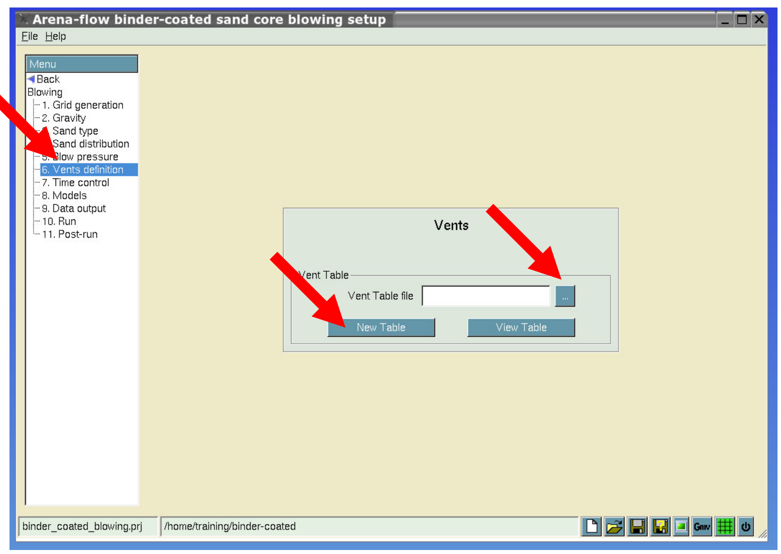
It should be noted that no distinction is made between "core blowing" and "core shooting". The terms are used interchangeably in this User Guide. Any

> **Note:** Any distinction between "core blowing" and "core shooting" is left up to the user during blow pressure boundary condition specification.

process differentiation is left up to the user in the specification of blow pressure boundary conditions.

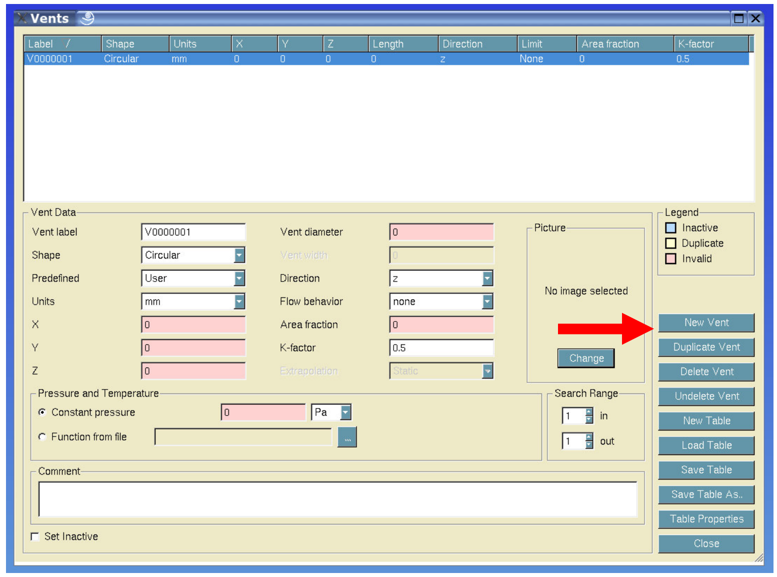
It is not required that the boundary pressure be constant during the simulation. The actual process likely involves a "ramping up" of the pressure over time. To enter a transient pressure, click on "Edit" in the "Pressure BC" window to create or edit a transient pressure file.

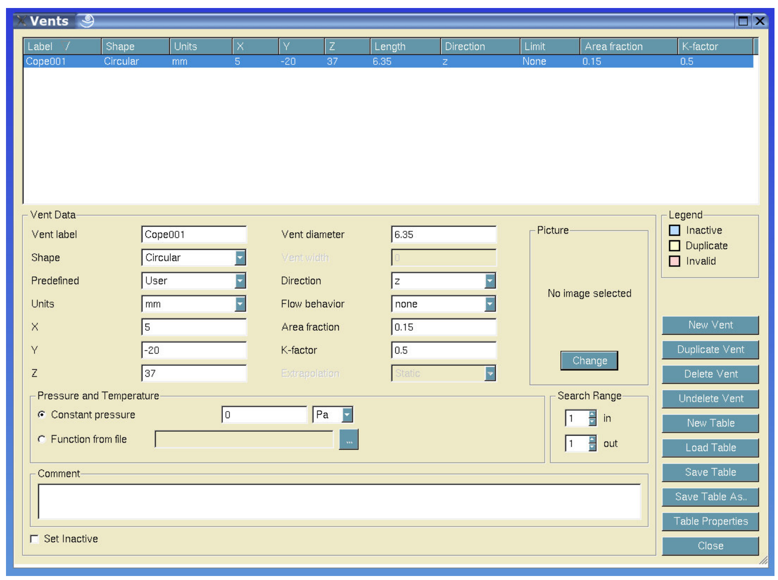
The file contains time and pressure entries in two columns, separated by "white space" (spaces, tabs, etc.). Lines beginning with the "#" character are comments and are ignored by the solver. ***Arena-flow***® will linearly interpolate between entries. The example above, ramps up from 0 to 300,000 Pa over 0.1 seconds.



Click on "Vents definition" to go to the "Vents" page. Vents are added through a vent file. To create a vent file, click on "New Table". To link to an existing vent file, click on the browse, , button. Vent files typically are identified with the .tvt extension.

New vent tables contain no information. Click on "New Vent" to begin populating the vent table. Much of the information is defaulted for you





Tip: Enter a "Vent label" to easily identify individual vents.

Enter a "Vent label" if desired. The vent label is a comment that helps to uniquely identify individual vents. This greatly simplifies later design permutations.

Note: The "Units" drop-down list affects both the location and "Vent diameter"

Enter the location of the vent center ("X", "Y", "Z"). Note that you may use non-SI units to do so, as indicated by the "Units" drop-down list.

Enter a "Vent diameter". The "Vent diameter" must also be entered with the same "Units" as the location.

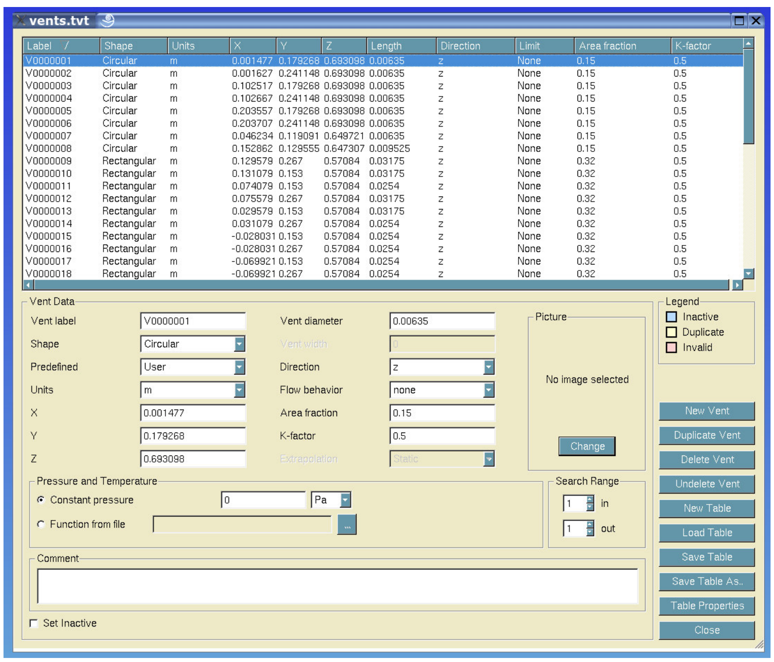
Enter a "Direction". This is the direction normal to the vent face. Typically cope and drag vents are in the z-direction if the model is oriented with the z-axis pointing up.

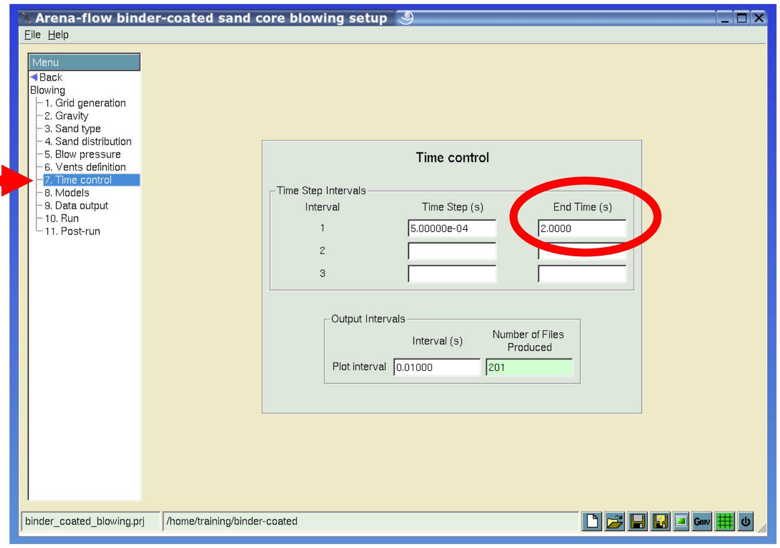
Enter an “Area fraction”. This is the fraction of the vent area that is open for the air flow. Typically vent open area fractions are between 0.1 and 0.35.

Usually all other values remain unchanged. To set a vent inactive, click on “Set Inactive”. This is a useful feature for design permutations.

An example of a populated vent table is given below. To change any parameters of an existing vent, click on the vent to load the parameters into the display region.

Remember to save your vent table before exiting.





Note: Users only need to change the "End Time" to ensure it corresponds to their blow time. All other parameters on the "Time control" page can remain unchanged.

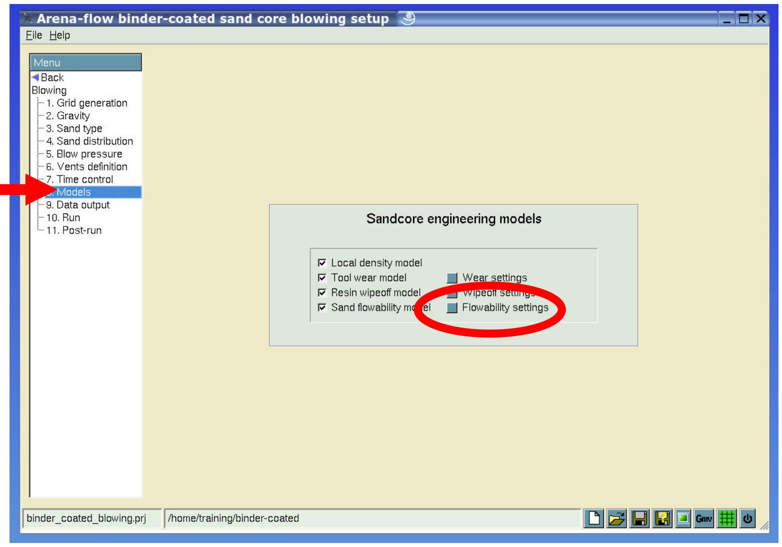
Click on "Time control" to go to the "Time control" page. Users are only required to change the "End Time". This is the blow time in seconds. Other parameters can be left unchanged.

The "Time Step" is controlled by the solver, and should be left at the default value. Only change the "Time Step" if advised to do so by an ***Arena-flow***® support engineer. The solver will adjust the time step for accuracy and stability purposes, using a robust algorithm.

The "Plot interval" is the frequency of GMV output file production. Lower plot intervals result in more GMV output files. Each GMV file takes time to write and uses a non-trivial amount of disk space. Typically about 100 – 200 GMV files are sufficient to create smooth animations of the transient ***Arena-flow***® results.

Note: When using the "***Arena-flow***® binder-coated sand core blowing setup" GUI, the "Sandcore engineering models" are defaulted for typical binder-coated sand core blowing simulations.

Next, click on "Models" to go to the "Sandcore engineering models" page. These models are defaulted for typical binder-coated sand core blowing simulations. It is usually best to leave these unchanged with the exception of the flowability model.



The flowability model is defaulted for simulations which include only a portion of the blow plate. The flowability model can be controlled by clicking on "Flowability settings". The default value is shown in Figure 16. By lowering the flowability of the sand, aged binder and non-optimal conditions can be simulated.

When simulating the performance of an entire blowing machine including a full magazine, however, please change the flowability settings as shown in Figure 17. If the sand is modeled with too low of a flowability, the rat-holes in the magazine can be over-predicted. Do not simulate magazines with a flowability setting below average – nonphysical results are possible.

Tip: The flowability settings are defaulted to simulate the filling of a single core, with a blow-plate feed boundary condition. You can lower the flowability to simulate defects which may occur under non-optimal filling conditions.

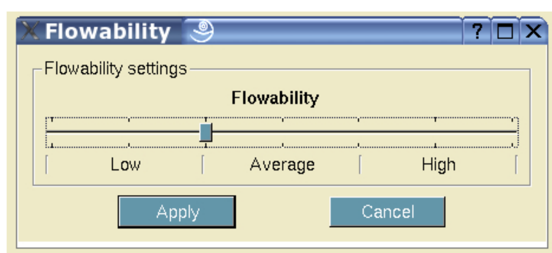


Figure 16: Binder-coated sand flowability settings for simulations involving a blow plate particle "feed" boundary condition

Important: When simulating a magazine, use a flowability setting of "Average" or higher. Using a lower flowability can result in excessive rat-holing and nonphysical results.

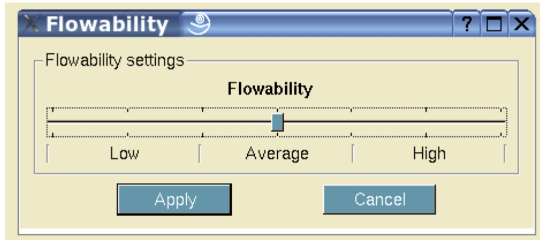
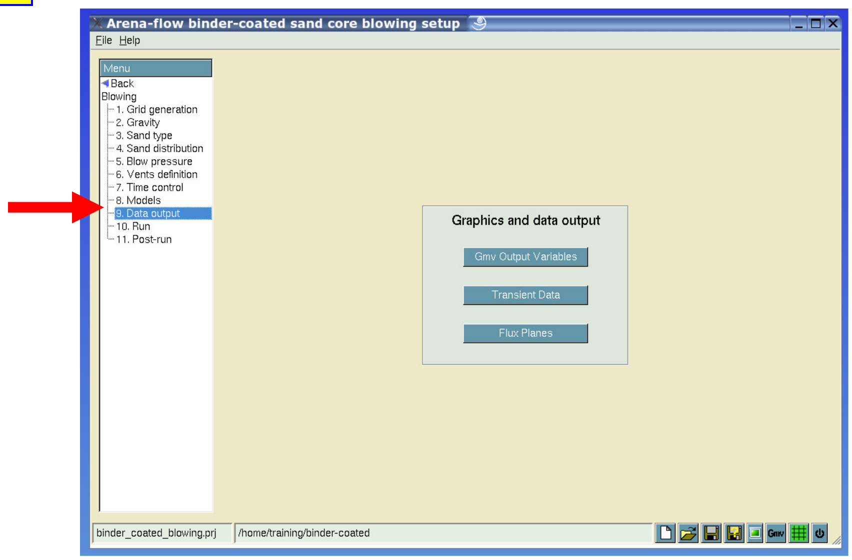


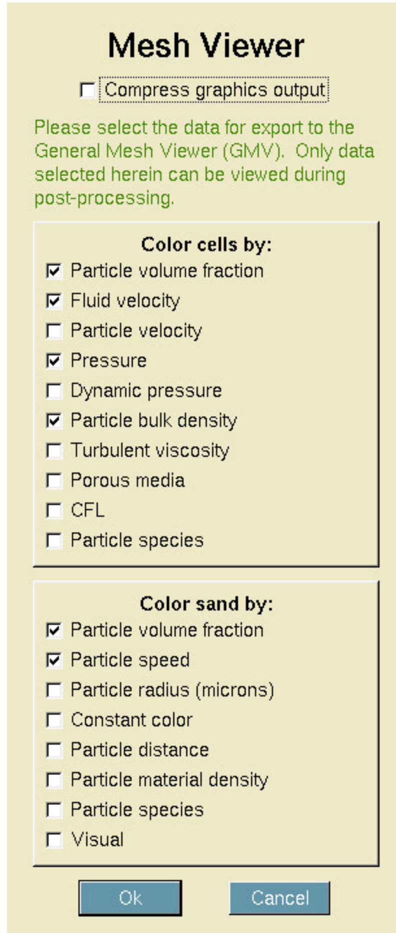
Figure 17: Binder-coated sand flowability settings for simulations involving a full magazine



Click on "Data output" to go to the "Graphics and data output" page. The ***Arena-flow***® CPFD™ solver requires a great deal of fluid and particle data to accurately solve sand core engineering problems. The ***Arena-flow***® solver may use 1GB of memory, or more, to run large problems. If all that data were output in a transient manner, a single ***Arena-flow***® calculation could easily fill even a large hard drive.

Important: If you do not tell ***Arena-flow***® to write variables to the GMV output files, the information will not be there!

For efficient post-processing, you need to tell ***Arena-flow***® what information you wish to post-process, before starting your calculation. This is typically done through the "Mesh Viewer" window. Think about your anticipated results before running your calculation - if you do not ask for the data to be written to the GMV files, it will likely not be there!

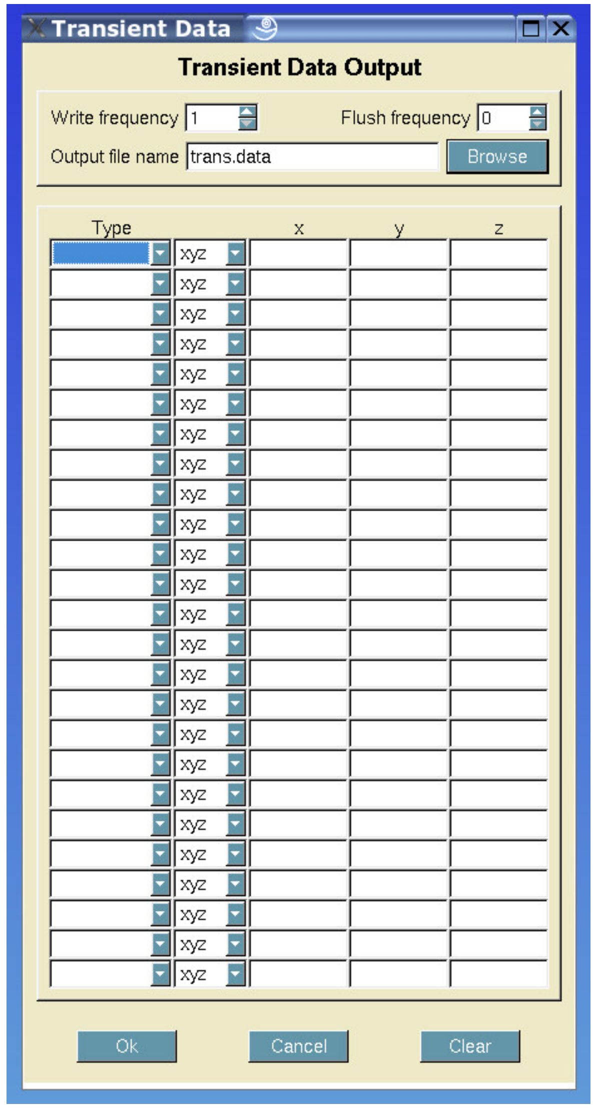


Tip: For cell data, it is good to select "Particle volume fraction", "Fluid velocity", "Pressure" and "Particle bulk density". Be sure to select "Particle species" if you are using different species of sand.

Tip: For sand data, it is good to select "Particle volume fraction" and "Particle speed". Be sure to select "Particle species" if you are using different species of sand. Be sure to select "Particle material density" if you are using sands with different material densities.

Click on "GMV Output Variables" to raise the "Mesh Viewer" window. Here you select what quantities you will wish to view during post-processing. Select the quantities by which you may wish to color your cells in the top of the window. Select the quantities by which you may wish to color your particles in the bottom of the window.

Typically, it is good practice to select "Particle volume fraction", "Fluid velocity", "Pressure", and "Particle bulk density" for cell data and "Particle volume fraction" and "Particle speed" for sand data. If you defined multiple sand species, be sure to select "Particle species" for both the cells and the sand. If you are using sands with different material densities, be sure to select "Particle material density" for the sand data output.



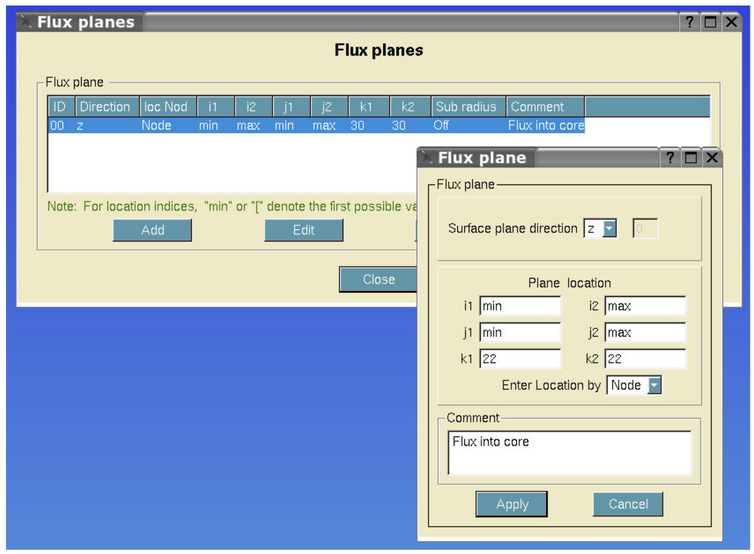
Note: Transient data specification is not required.

The specification of transient data is not required. The "Transient Data Output" window allows you to request that information from pre-identified cells be written to a file during the calculation.

To launch the "Transient Data Output" window, click on "Transient Data" on the "Graphics and data output" page. Specify an "Output file name"; the default is trans.data. Transient data output locations are specified by selecting the data type on the left, and identifying the location at the right. The location can be either spatial coordinates (x, y, z) or cell indices (i, j, k). Tell ***Arena-flow***® which you are using via the xyz / node drop down menu.

Arena-flow® will write the desired data to the output file in columnar form during the run. The output file is in a plain text format, and may be graphed via any plotting utility supporting columnar input. The "Transient Data Output" feature is an excellent way to monitor pressures at discrete locations in your system.

Note: Flux plane specification is not required.

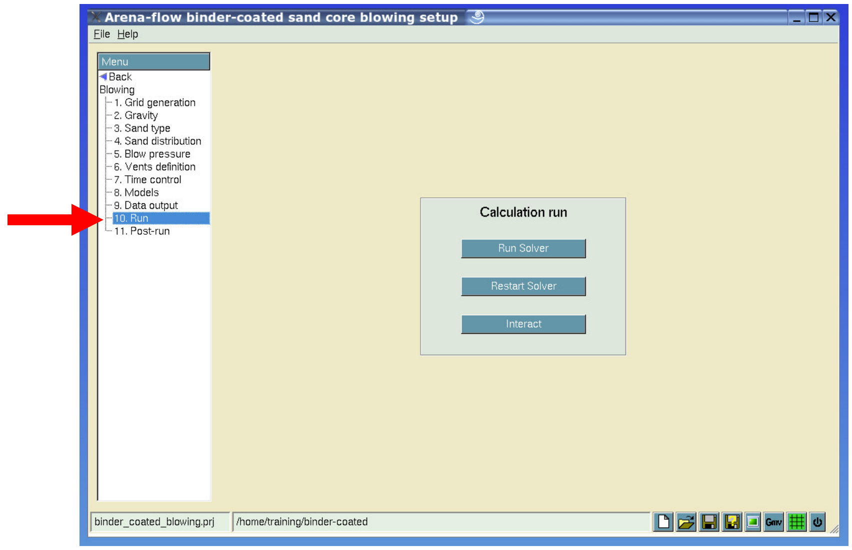


As with transient data, the specification of flux planes is also not required. Flux planes track all fluid and particles which cross them during the calculation.

To launch the "Flux planes" window, click on "Flux Planes" on the "Graphics and data output" page. Individual planes can be defined by clicking "Add".

To define a flux plane, select the "Surface plane direction" and the "Plane location". The "Plane location" can be entered in spatial coordinates (x, y, z) or cell indices (i, j, k – i.e. "Node"). Tell ***Arena-flow***® which you are using via the "Enter Location by" drop down menu.

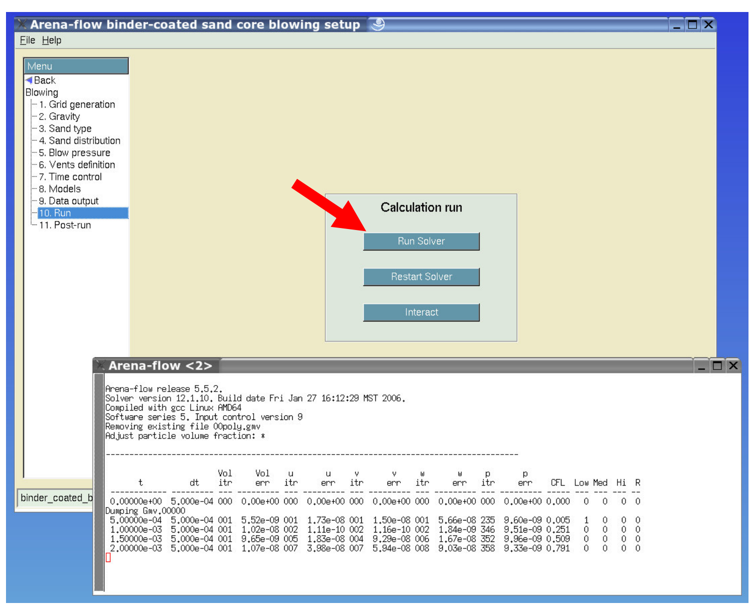
Each flux plane creates a text file containing columnar data. The files will be named Flux_## where ## is the flux plane "ID". The output file is in a plain text format, and may be graphed via any plotting utility supporting columnar input. The "Flux plane" feature is an excellent way to monitor the flow through blow tubes.



Tip: You can run the ***Arena-flow***® solver by clicking "Run Solver" in the GUI, or by typing arena.x at the command line.

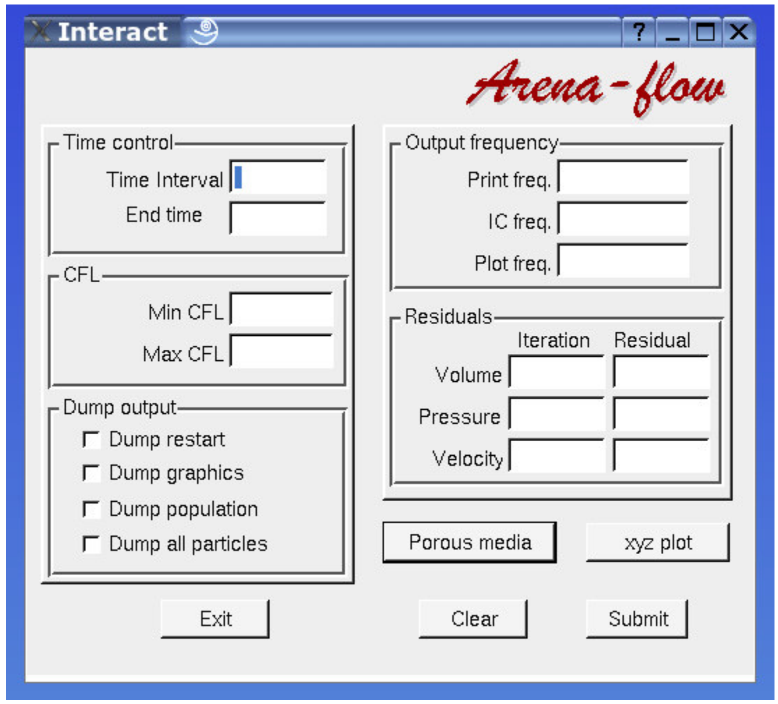
Click on "Run" to go to the "Calculation run" page. Once your calculation is fully set up, click on "Run Solver" to start the simulation. The solver may also be run from the command line with the "arena.x" command to allow for scripting. The argument is the project file name.

When you click on "Run Solver" in the GUI, a new window opens displaying solver information during execution. It is very important to leave this window open during the run. Closing this window will halt solver execution.



Important: Be sure to leave the ***Arena-flow***® solver window open during the calculation. Closing it will stop the run.

The first column of data output in the solver window shows the simulation time. More information regarding solver output is found in section 2.10, Additional Features.



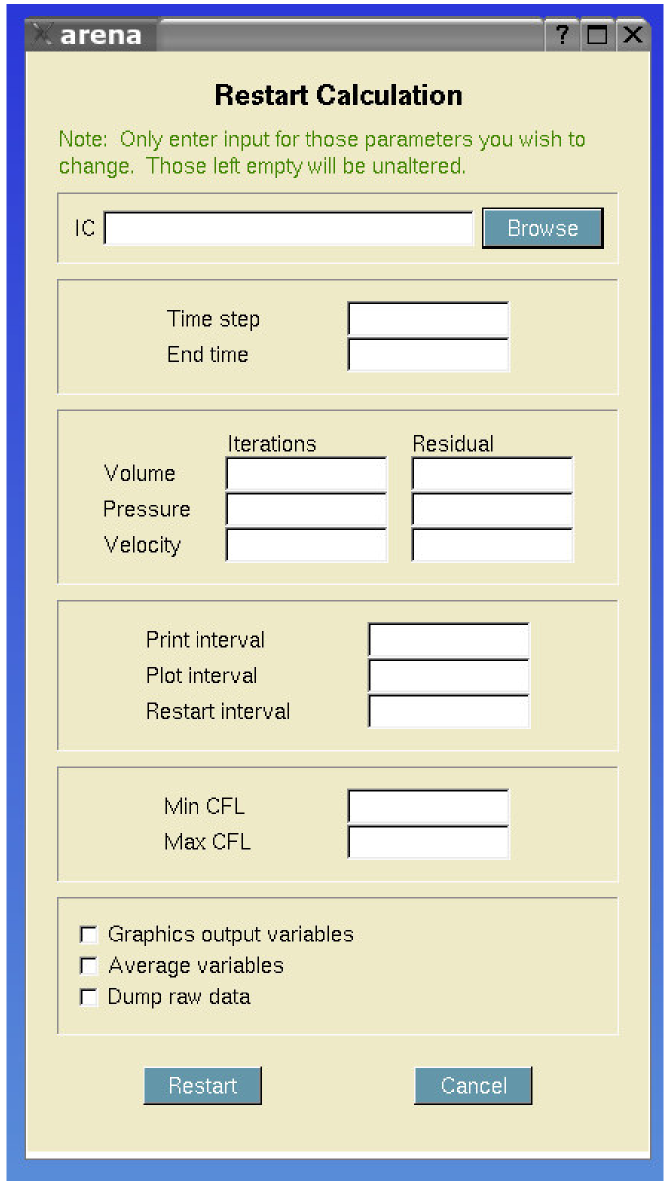
Note: It is possible to change some solver parameters while the calculation is running.

It is possible to change some solver parameters while the calculation is running. Click on "Interact" on the "Calculation run" page to raise the "Interact" window.

A good use of the "Interact" utility is to change the calculation end time during the run. For example, if your core is full, change the "End time" to the current time and click "Submit". This will end the calculation. Alternately, if you are nearing your originally defined end time and your core is still not full, extend the "End time" by entering a new value and click "Submit".

Tip: Only change parameters (other than the "End time") if you are advised to do so by an ***Arena-flow***® support engineer.

Only enter the information you wish to change. Use this utility cautiously, and only change other parameters if advised to do so by an ***Arena-flow***® support engineer. This utility is under redevelopment.

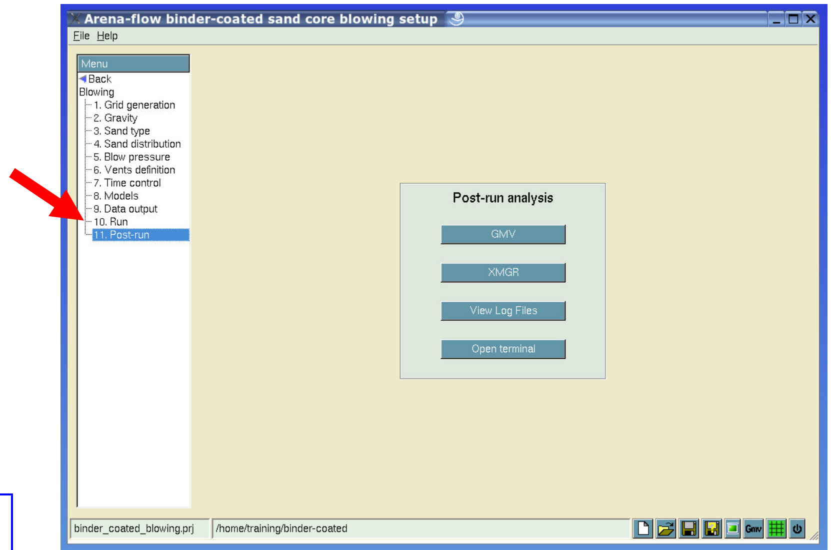


Note: It is possible to restart the solver after completion.

It is also possible to restart your calculation once it has completed. To do so, click on "Restart Solver" on the "Calculation run" page to raise the "Restart Calculation" window.

Tip: Only change parameters (other than the "End time") if you are advised to do so by an ***Arena-flow***® support engineer.

You must enter a restart file (IC). Restart file names start with "IC_" and may contain additional characters. Click on "Browse" to select the file. Only enter the information you wish to change from the original run. If you would like to run the calculation for longer than originally intended, please enter a new "End time". As with the "Interact" utility, please only change other parameters if advised to do so by an ***Arena-flow***® support engineer.



Tip: All of the utilities on the "Post-run analysis" page can be used while the calculation is running.

Tip: GMV can be used to view your ***Arena-flow***® results, while the calculation is running!

Note: You can use any plotting utility to graph columnar data. Use of XMGR is not required.

Click on "Post-run" to open the "Post-run analysis" page. All these features can be used while the calculation is running. The utilities are as follows:

- "GMV" is the general mesh viewer. This is used to post-process the graphical output from ***Arena-flow***®. The use of GMV is further described in section 2.9, Post-Processing. Using GMV, you can view your results while your calculation is running.

- "XMGR" is a plotting utility. It can be used to plot columnar data such as flux plane and transient data output files. XMGR is an open-source program. More information regarding the use of XMGR can be found in a web search. It is not required to use XMGR, please use your favorite plotting utility to graph columnar data.

- "View Log Files" launches a viewer to open text files. This can be used to view various output files.

Tip: Use the terminal window to enter additional Linux commands.

- "Open terminal" launches a Linux command prompt in the working directory to allow for the input of various commands.

Important: Check these things every time you start an ***Arena-flow***® binder-coated sand core blowing calculation.

As soon as you start a binder-coated sand core blowing calculation running, check the following:

- Check that your boundary conditions are applied in the correct location
- Check that your vents are applied correctly
- Check that your flux planes are defined as desired (if used)
- Check that your sand is in the correct location
- If you are not simulating a full magazine, ensure that the particle “Feed” condition is applied to the “Blow pressure” boundary condition
- If you are simulating a full magazine, ensure the flowability setting is “Average” or higher

Important: Check these things every time you analyze the results from an ***Arena-flow***® binder-coated sand core blowing calculation.

Once your calculation is complete, be sure to view:

- Transient filling of the core
- Variations in the final core density
- Sand speed in regions of poor fill
- Air pressure and flow during filling
- Regions of likely tool wear

More information regarding analyzing your results is presented in the various ***Arena-flow***® training courses. Please consult your training manual. Contact your sales representative or support engineer for more information regarding course content and schedule.

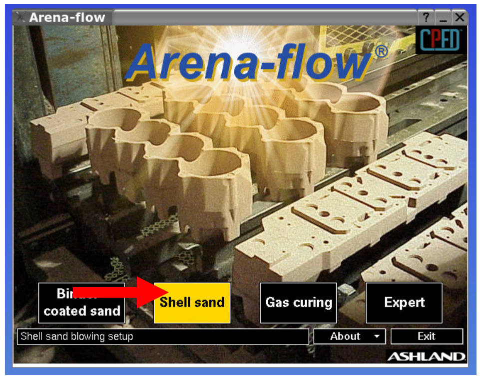
2.5 Shell Sand Core Blowing

In this chapter you will learn about:

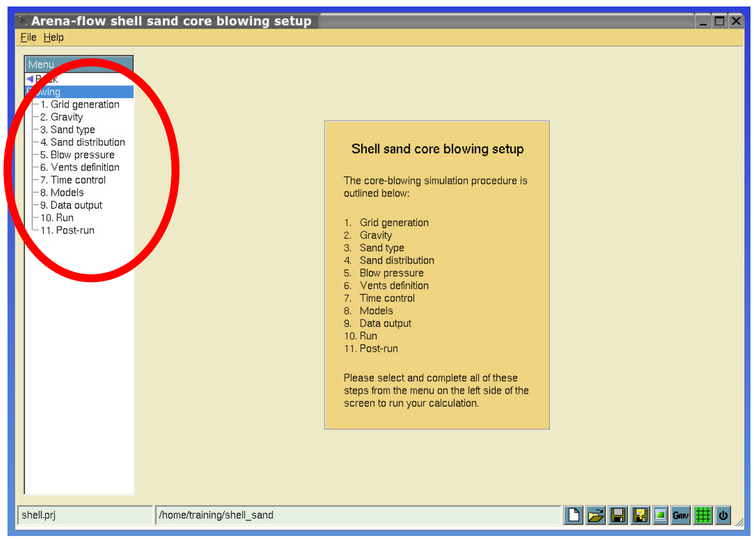
- *Setting up a shell sand core blowing project*
- *Running a shell sand core blowing project*
- *Evaluating the results from a shell sand core blowing project*

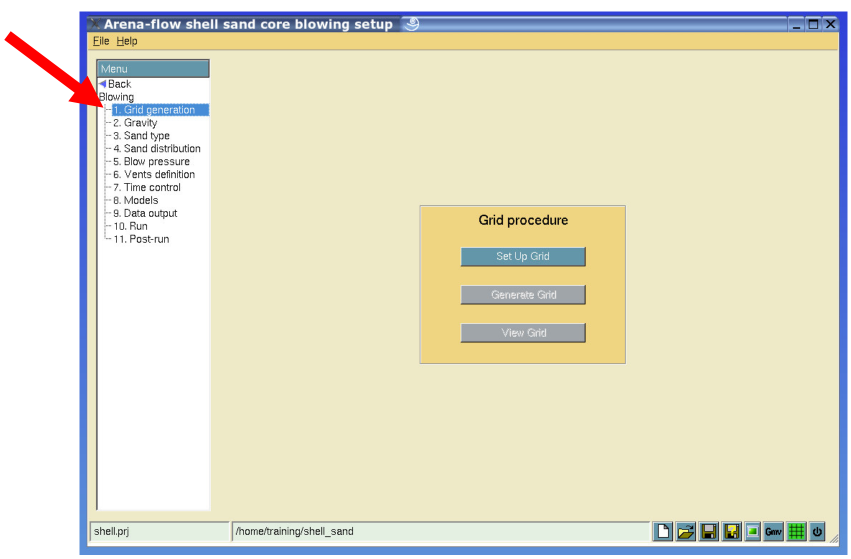
To model the blowing of a shell core with ***Arena-flow***®, click on “Shell sand” as shown.

Create or open a project file in the desired working directory to launch the “***Arena-flow***® shell sand core blowing setup” window. To set up and run a shell sand core blowing project, work through the 11 menu items shown in the tree on the left of the window



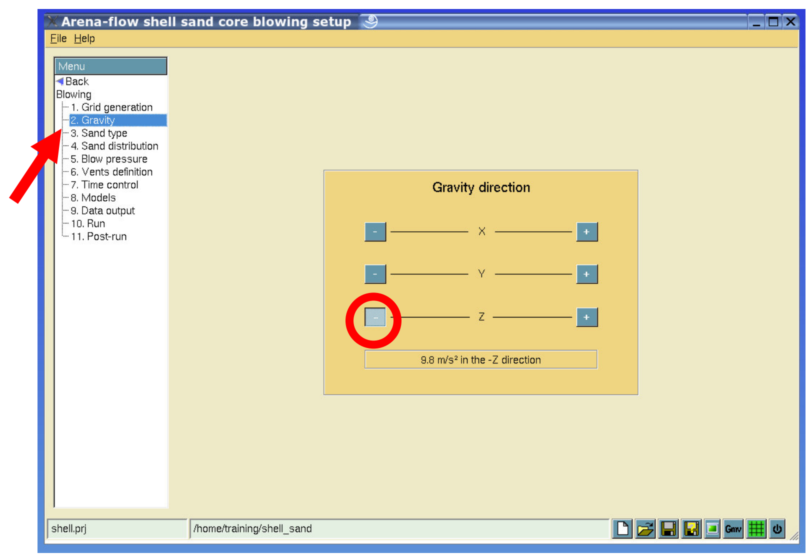
Tip: To set up and run a shell sand core blowing project, work through the 11 menu items shown in the tree on the left of the window.



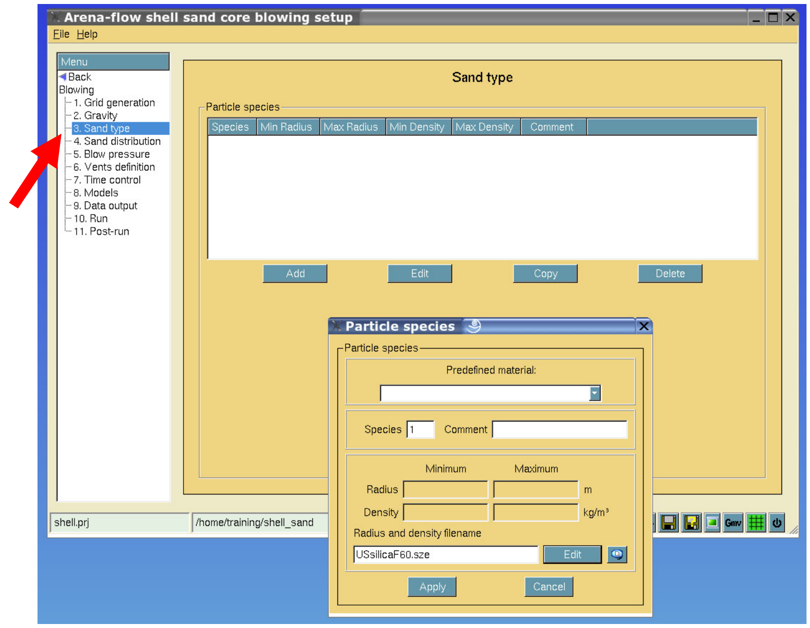


The first step to creating any ***Arena-flow***® model is mesh generation. Click on "Grid generation" to go to the "Grid procedure" page.

On the "Grid procedure" page, create and evaluate the grid by working through the three steps: "Set Up Grid", "Generate Grid" and "View Grid" (***Arena-flow***® 6.0 shown). The use of the ***Arena-flow***® grid generator for shell sand core blowing is outlined in section 2.3.2 of this User Guide. Proper grid generation techniques are presented in the various ***Arena-flow***® training classes. Classes are held regularly. Contact your sales representative or support engineer for more information regarding course content and schedule.



Click on the next menu item, "Gravity". This is where you tell ***Arena-flow***® the orientation of your CAD, relative to gravity. In this example, the CAD is oriented such that the z-axis is up. Thus, the gravity vector should point in the –z direction.

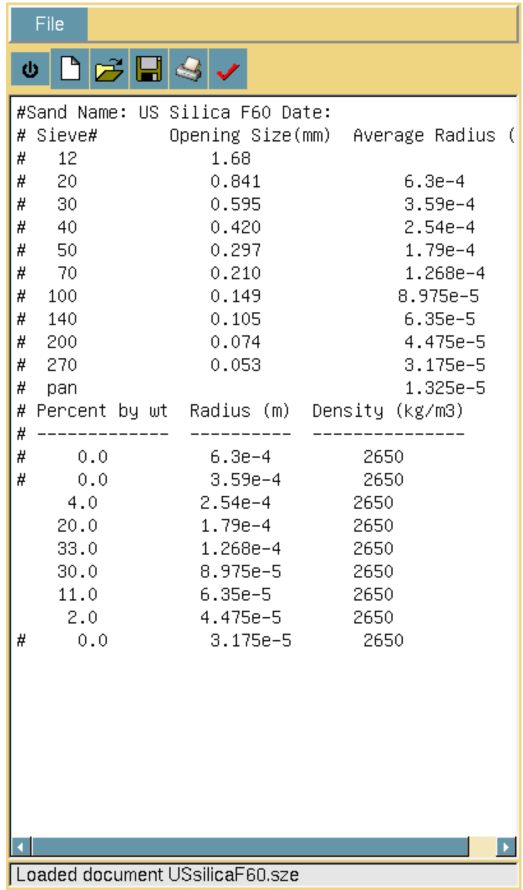


Note: The "Sand type" page defines the sands to be used in the calculation. The "Sand distribution" page defines where the sand is located (or distributed) at the start of the calculation.

Next, click on "Sand type". This is where you tell ***Arena-flow***® what sand will be modeled in the calculation. You may define multiple sand types. These may be combined to simulate a sand mixture or may be layered in the magazine to simulate colored sand experiments. The location of the sand is entered on the next page; you simply define the various sand species here.

To define a sand type for use in the calculation, click on "Add". This raises the "Particle species" window. There are three ways to enter a sand type:

- Select a default sand type from the "Predefined material" drop-down menu
- Enter a minimum and maximum sand grain radius and density.
- Enter a custom size distribution from a sieve analysis by clicking "Edit"




```
#Sand Name: US Silica F60 Date:
# Sieve#       Opening Size(mm)  Average Radius (
#   12             1.68
#   20             0.841              6.3e-4
#   30             0.595              3.59e-4
#   40             0.420              2.54e-4
#   50             0.297              1.79e-4
#   70             0.210              1.268e-4
#  100             0.149             8.975e-5
#  140             0.105              6.35e-5
#  200             0.074              4.475e-5
#  270             0.053              3.175e-5
#  pan                                1.325e-5
# Percent by wt  Radius (m)  Density (kg/m3)
# -------------  ----------  ---------------
#      0.0          6.3e-4        2650
#      0.0          3.59e-4       2650
    4.0          2.54e-4       2650
   20.0          1.79e-4       2650
   33.0          1.268e-4      2650
   30.0          8.975e-5      2650
   11.0          6.35e-5       2650
    2.0          4.475e-5      2650
#      0.0          3.175e-5      2650
```

Note: The size distribution file contains three columns: percent, radius and density. The percent may be incremental or cumulative. Lines beginning with a "#" character are ignored.

A sample sand size distribution file is shown. Sand size distribution files contain the .sze extension. Once a size distribution file is created for your sand, you may use it repeatedly in successive calculations.

Tip: Remember that all ***Arena-flow***® input is in SI units, unless specified to be otherwise. The radius is thus in meters. Be careful, it's radius – not diameter.

The format of the sand size distribution file is columnar, with the columns separated by "white space" such as spaces and tabs. The "#" character indicates that the line is a comment. Lines beginning with the "#" character are ignored by ***Arena-flow***®. The three columns are a percentage of sand, radius and density. Please note that all input is in SI units, thus the sand grain radius is in meters and the density is in kg/m^3.

Tip: Remember that the density required is the raw (material) density of the sand.

The density is the material density of the sand used, not to be confused with the bulk density of a sand mixture. In this example, the refractory material is a silica sand which typically has a material density between 2650 and 2670 kg/m^3 [19].

> **Note:** ***Arena-flow***® can model granular materials other than sand, such as ceramic beads and sand additives.

In actuality, the "Sand type" in ***Arena-flow***® need not be a sand at all. ***Arena-flow***® can model various granular, refractory materials such as ceramic beads and sand additives, provided the size distribution and material density are known and properly defined.

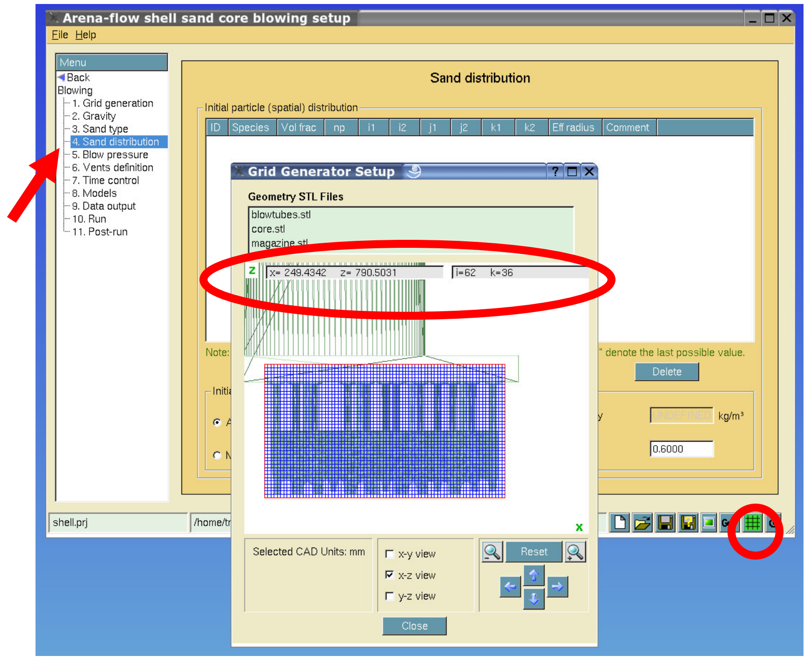
Also note that ***Arena-flow***® accepts two ways of entering a custom sand size distribution file. The first is shown, above with the percentages being an incremental percentage of sand on each sieve. The second is a cumulative distribution where the percentage is a sum of the percentages up to the current bin. For example, the percentages above (0.0, 4.0, 20.0, 33.0, 30.0, 11.0, 2.0, 0.0) could be entered as 0.0, 4.0, 24.0, 57.0, 87.0, 98.0, 100.0). A cumulative size distribution is preferred. If a cumulative size distribution is defined, the first entry must be 0.0 and the last must be 100.0. It does not matter if the sand is defined in increasing or decreasing size order.

Once a sand type is defined using the "Add" button, it appears in the "Particle species" table on the "Sand type" page. This sand species can then be edited using the "Edit" button, reproduced using the "Copy" button or deleted using the "Delete" button.

> **Note:** If multiple sand types are defined, each is identified with a unique species number. The results can be colored by species to aid in visualization.

Multiple sand types may be defined. If multiple sand types are used, each is identified with a unique species number. The results can then be colored by "Species" to show various sands as different colors. This is a very useful way to visualize the sand motion through the system. If this is your intent, be sure to ask for "Species" to be included in the output files when using the "Mesh Viewer" window as described on "Data output" page of the GUI.

The sand is automatically assumed to be a shell sand with various properties defaulted by the ***Arena-flow***® GUI. Wall bounce and sand grain interaction are indicative of a partly-heated shell sand.

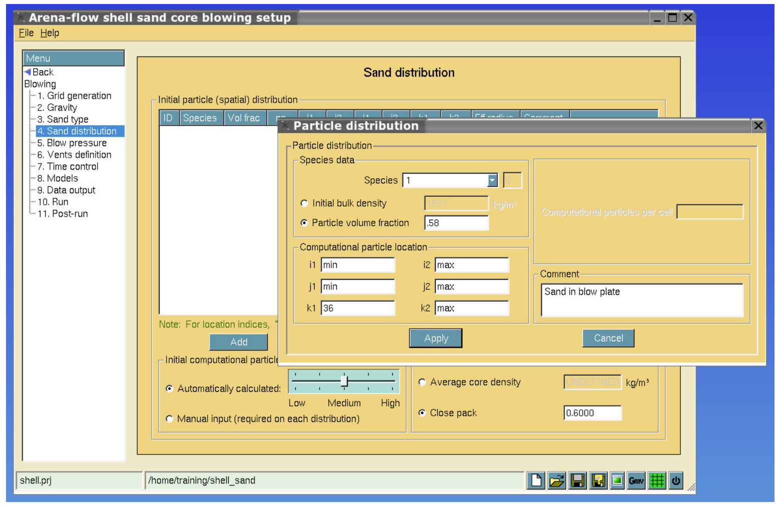


After you define the sand to be used in the calculation, you must tell ***Arena-flow***® where you would like the sand to be at the start of the simulation. Click on "Sand distribution" to go to the "Sand distribution" page.

> **Tip:** You can always refer to your grid by clicking on the reference grid button.

The location of the sand is related to the grid you defined in step 1. You may reference this grid by clicking on the button on the bottom right of the screen. This opens a non-editable version of the "Grid Generator Setup" screen. As you move your mouse over your grid, the coordinates (x,y,z) and cell indices (i, j, k) are displayed at the top of the window. Note that a two-dimensional plane is shown, thus only two of the three coordinates and grid indices are shown at one time. Determine which three-dimensional block of cells you wish to initialize with particles.

To place the sand in the cells, click on "Add" to raise the "Particle distribution" window. First select the "Species" from the drop-down list. You may select from any of the sands you defined in the "Sand type" page. If you used a comment when you defined your sand, this will also appear in the "Species" drop-down list.



Note: The "Initial bulk density" or "Particle volume fraction" must be less than or equal to the "Average core density" or "Close pack".

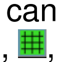
Next enter either an "Initial bulk density" or "Particle volume fraction". This is how much sand is to be placed in the cells. The "Initial bulk density" or "Particle volume fraction" must be less than or equal to the "Average core density" or "Close pack" entered on the "Sand distribution" page.

The "Average core density" or "Close pack" value controls how much sand is permitted into the cells in the core during the calculation. It is an average value, because the various physical models in ***Arena-flow***® will vary the packing from cell to cell to predict the density distribution in the filled core.

Tip: It is good practice to define the "Initial bulk density" or "Particle volume fraction" to be slightly less compacted than the "Average core density" or "Close pack".

In general, it is good practice to define the "Initial bulk density" or "Particle volume fraction" to be slightly less compacted (say 2-5% less) than the "Average core density" or "Close pack". When the core is filled, it is done so by a high pressure air source injecting the sand at high velocities. When the blow plate and magazine are filled, the sand is usually allowed to fall in under gravity. These different sand fill processes will likely result in different packing levels.

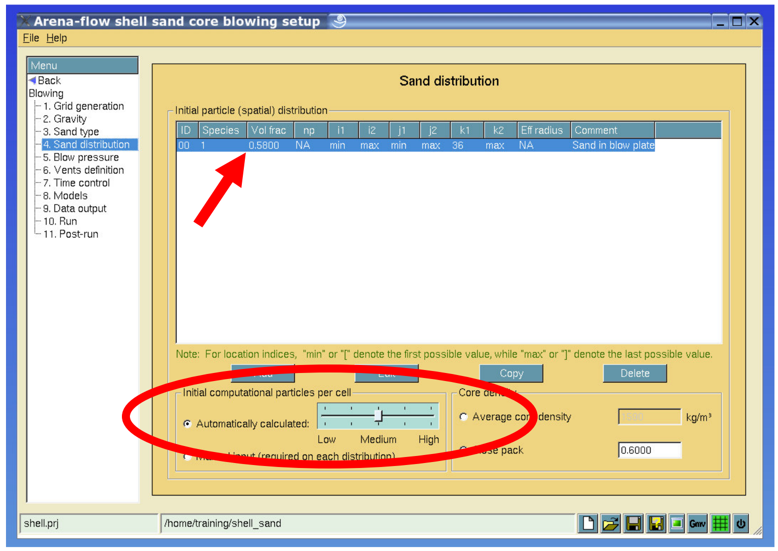
Tip: The "min" and "max" keywords can be used instead of the minimum and maximum cell or face indices whenever defining a boundary or initial condition.

Next you must define the initial location of the sand. The "Computational particle location" asks for six values, corresponding to the minimum and maximum cell indices in the x-, y- and z-directions defining the three-dimensional block of cells containing the sand. These cell indices can be determined by using the reference grid tool, , however, the keywords "min" and "max" can also be used to reference the minimum and maximum cell index in each respective direction.

Tip: Enter comments whenever possible – it will save you time in the long run.

Lastly you are permitted to enter a "Comment" for the distribution. This is not used by the solver, but can be extremely useful when setting up large projects.

Sand is initialized in the blow plate at the start of the calculation. Shell sand typically flows out of the blow tubes between successive blows.



Once a sand distribution is defined using the "Add" button, it appears in the "Initial particle (spatial) distribution" table on the "Sand distribution" page. These distributions can then be edited using the "Edit" button, reproduced using the "Copy" button or deleted using the "Delete" button.

There could be hundreds of millions of individual sand grains in a core; to make the CPFD™ method efficient for practical, commercial applications, a lesser, representative number of computational particles is modeled. The number of computational particles must be large enough to be indicative of the overall core filling process, but not too large to exceed practical memory and runtime constraints.

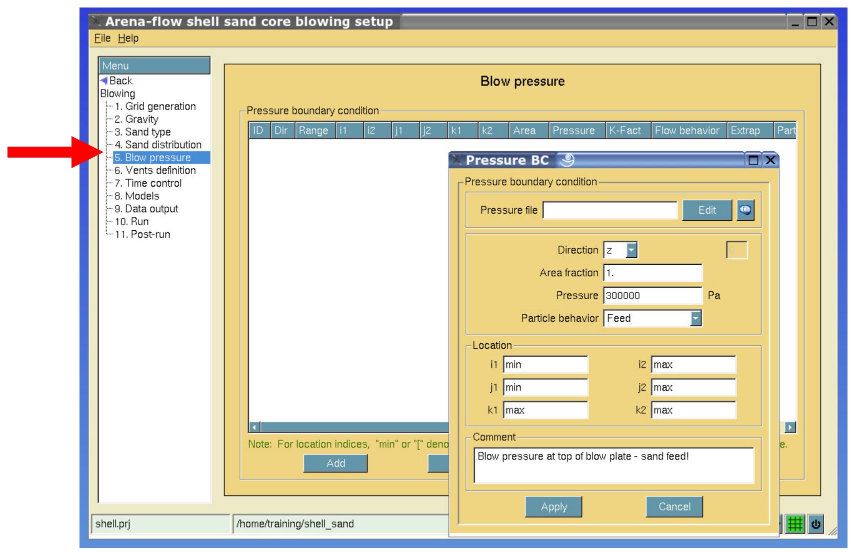
Note: ***Arena-flow***® automatically sets the sand grain computational resolution for you.

Arena-flow® automatically sets the sand grain computational resolution for you. You have control over the "Initial computational particles per cell" setting on the "Sand distribution" page via a slider bar.

> **Tip:** Use the "Automatically calculated", "Medium" or "Medium / High" setting for "Initial computational particles per cell".

To use this, set the "Initial computational particles per cell" via the default, "Automatically calculated" option. Move the "Low / Medium / High" slider to the desired resolution. The default setting is "Medium". For most commercial applications, a "Medium" or "Medium / High" setting is recommended.

Arena-flow® also has advanced capabilities to control the computational modeling of sand grains. This is controlled via the "Manual input" setting. Please only use the "Manual input" option if advised to do so by an ***Arena-flow***® support engineer. The "Automatically calculated" option works well for all typical, commercial applications.



After the "Sand distribution" is set, click on "Blow pressure" to go to the "Blow pressure" page. Here you tell ***Arena-flow***® what blow pressure will be used, whether the pressure is constant or time varying and where the pressure is to be applied.

Click on "Add" to raise the "Pressure BC" window.

Select the "Direction" from the drop-down menu. This tells ***Arena-flow***® upon which faces of the cells the pressure boundary condition is to be applied. If your CAD was designed with the z-axis pointing up then the default directional value of "z" can be left unchanged.

Enter the “Area fraction”. This is the fraction of the cell faces which is open to the pressure. Since the model begins a few cells above the blow tubes for shell sand applications, the “Area-fraction” should remain unchanged at 1.0.

Tip: Many conversion factors to SI units can be found by clicking on “Help”, “Units Reference”.

Enter the “Pressure”. This is the blow pressure in Pa. To determine the appropriate conversion factor when converting from alternate pressure measurements to Pa, click on “Help”, “Units Reference”.

Important: The CFD component of ***Arena-flow***® is an incompressible solver; thus all pressures are gauge pressures with zero (0) representing the vented, atmospheric pressure.

Please note that the CFD component of ***Arena-flow***® is an incompressible solver. As such, all pressure measurements are relative. Calculations are typically set up with a zero (0) pressure at the vents. Thus the blow pressure should be in gauge pressure, relative to the vents.

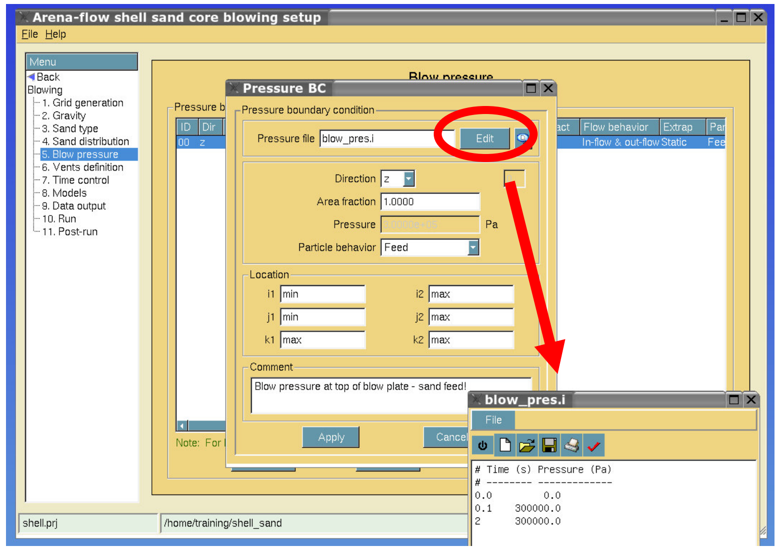
Important: Since you are starting you model in the blow plate, set the “Particle behavior” to “Feed”.

Next specify the “Particle behavior”. Since shell sand core blowing models start a few cells above the blow tubes, change the “Particle behavior” to “feed”. This will permit more particles to enter the computational domain as others leave the cells next to the boundary, simulating a full magazine. ***Arena-flow***® calculations with “feed” activated tend to start quickly and slow down as they run, due to the increased number of computational sand grains as calculation progresses.

Next specify the location of the blow pressure boundary condition. These are boundary cell indices and they must define a plane. You may use the reference grid utility, , to help you determine proper values for the indices. Additionally, the “min” and “max” keywords can also be used to reference the minimum and maximum cell index in each respective direction. Since you are starting your computational model just above the blow tubes, set your blow pressure in the blow plate.

Sometimes multiple planes are required to fully specify the blow pressure inlet location. These boundary conditions can be edited using the “Edit” button, reproduced using the “Copy” button or deleted using the “Delete” button.

Finally, you are permitted to enter a “Comment” for the distribution.

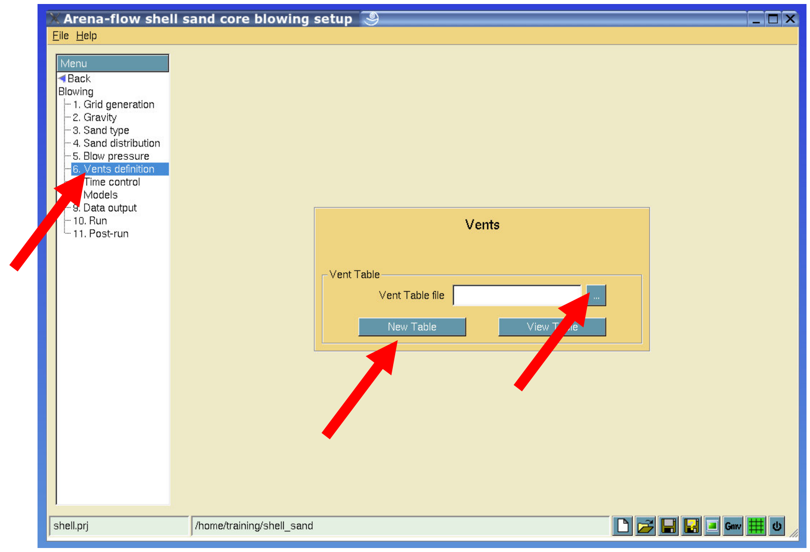


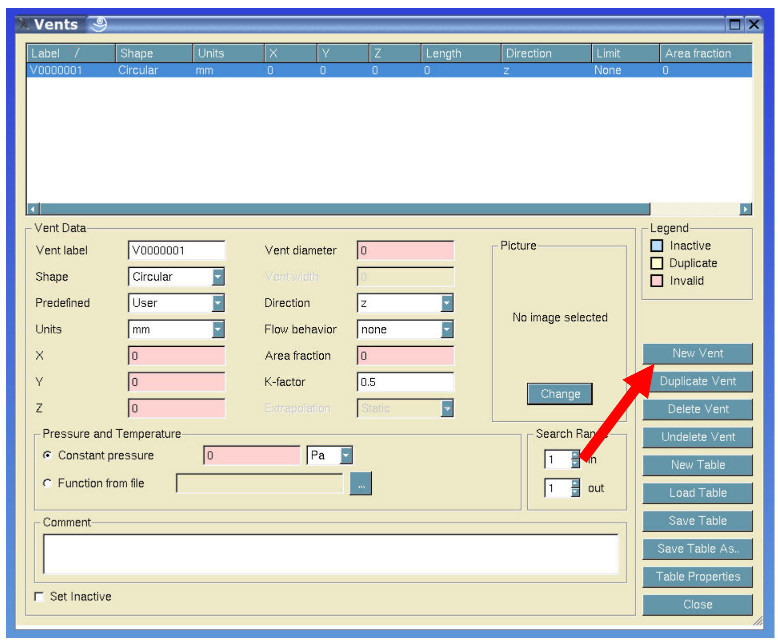
It is not required that the boundary pressure be constant during the simulation. The actual process likely involves a “ramping up” of the pressure over time. To enter a transient pressure, click on “Edit” in the “Pressure BC” window to create or edit a transient pressure file.

The file contains time and pressure entries in two columns, separated by “white space” (spaces, tabs, etc.). Lines beginning with the “#” character are comments and are ignored by the solver. ***Arena-flow***® will linearly interpolate between entries. The example above, ramps up from 0 to 300,000 Pa over 0.1 seconds.

Click on “Vents definition” to go to the “Vents” page. Vents are added through a vent file. To create a vent file, click on “New Table”. To link to an existing vent file, click on the browse, ..., button. Vent files typically are identified with the .tvt extension.

New vent tables contain no information. Click on “New Vent” to begin populating the vent table. Much of the information is defaulted for you.





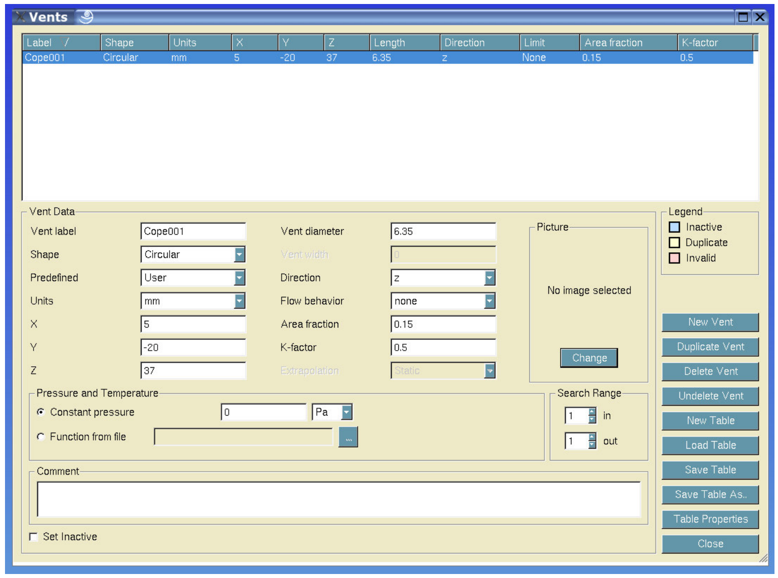
Tip: Enter a “Vent label” to easily identify individual vents.

Enter a “Vent label” if desired. The vent label is a comment that helps to uniquely identify individual vents. This greatly simplifies later design permutations.

Note: The “Units” drop-down list affects both the location and “Vent diameter”

Enter the location of the vent center (“X”, “Y”, “Z”). Note that you may use non-SI units to do so, as indicated by the “Units” drop-down list.

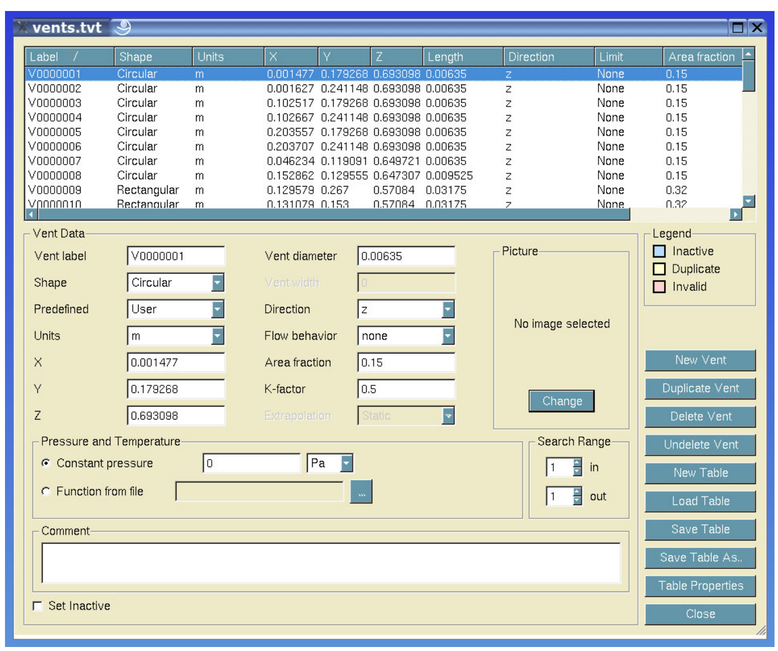
Enter a “Vent diameter”. The “Vent diameter” must also be entered with the same “Units” as the location.



Enter a "Direction". This is the direction normal to the vent face. Typically cope and drag vents are in the z-direction if the model is oriented with the z-axis pointing up.

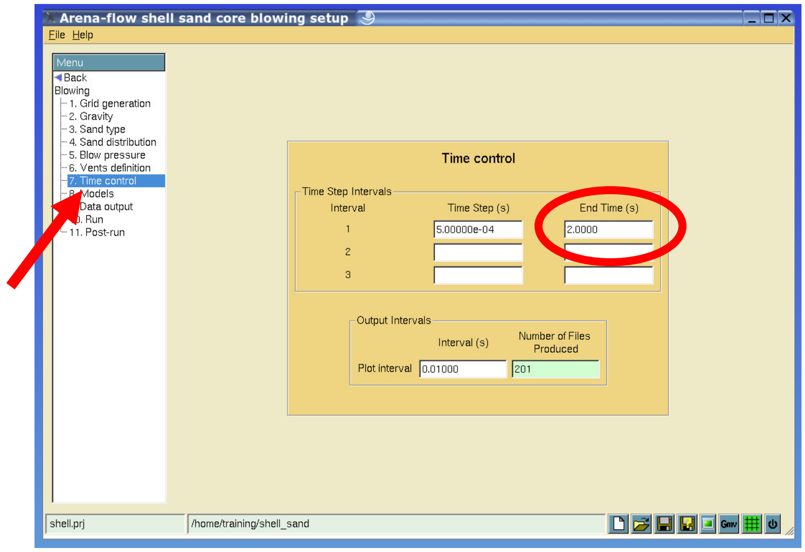
Enter an "Area fraction". This is the fraction of the vent area that is open for the air flow. Typically vent open area fractions are between 0.1 and 0.35.

Usually all other values remain unchanged. To set a vent inactive, click on "Set Inactive". This is a useful feature for design permutations.



An example of a populated vent table is given above. To change any parameters of an existing vent, click on the vent to load the parameters into the display region.

Be sure to save your vent table before exiting.

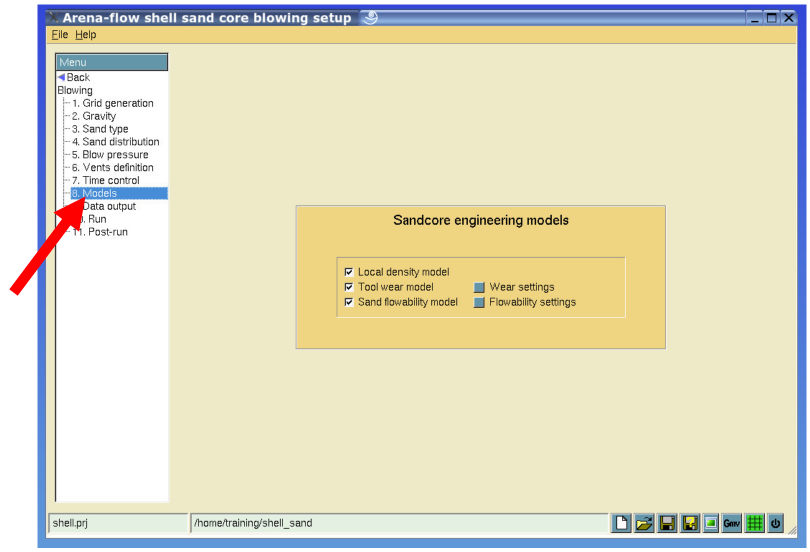


Note: Users only need to change the "End Time" to ensure it corresponds to their blow time. All other parameters on the "Time control" page can remain unchanged.

Click on "Time control" to go to the "Time control" page. Users are only required to change the "End Time". This is the blow time in seconds. Other parameters can be left unchanged.

The "Time Step" is controlled by the solver, and should be left at the default value. Only change the "Time Step" if advised to do so by an ***Arena-flow***® support engineer. The solver will adjust the time step for accuracy and stability purposes, using a robust algorithm.

The "Plot interval" is the frequency of GMV output file production. Lower plot intervals result in more GMV output files. Each GMV file takes time to write and uses a non-trivial amount of disk space. Typically about 100 – 200 GMV files are sufficient to create smooth animations of the transient ***Arena-flow***® results.



Next, click on “Models” to go to the “Sandcore engineering models” page. These models are defaulted for typical shell sand core blowing simulations. It is usually best to leave these unchanged with the exception of the flowability model.

The flowability model is defaulted for simulations without hardening of the core during the blow. The flowability model can be controlled by clicking on “Flowability settings”. The default value is shown in Figure 18.

Tip: The flowability settings are defaulted to simulate the filling of a core where heat-hardening effects are neglected. You can lower the flowability to simulate defects when hardening occurs during the blow.

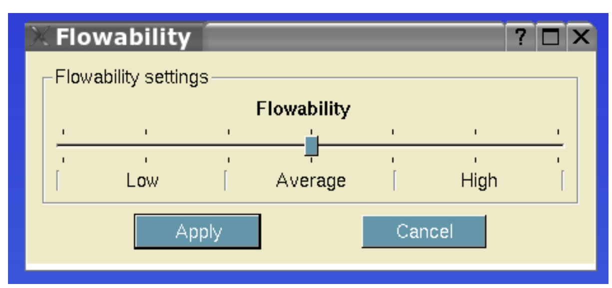


Figure 18: Shell sand flowability settings without core hardening during the blow time

By lowering the flowability of the sand, defects due to sand hardening will become more apparent. Figure 19 shows an extreme case which simulates a great deal of reduced flowability due to core hardening during the blow time.

Tip: By using low flowability settings, you are applying a restrictive force on sand as it slows and stops, simulating simple core hardening.

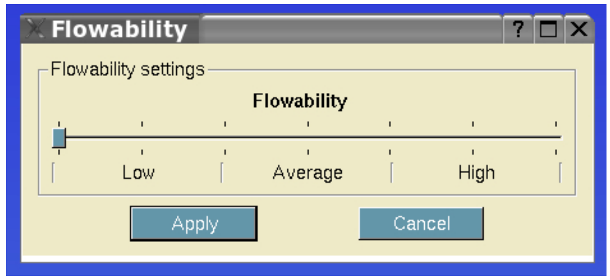
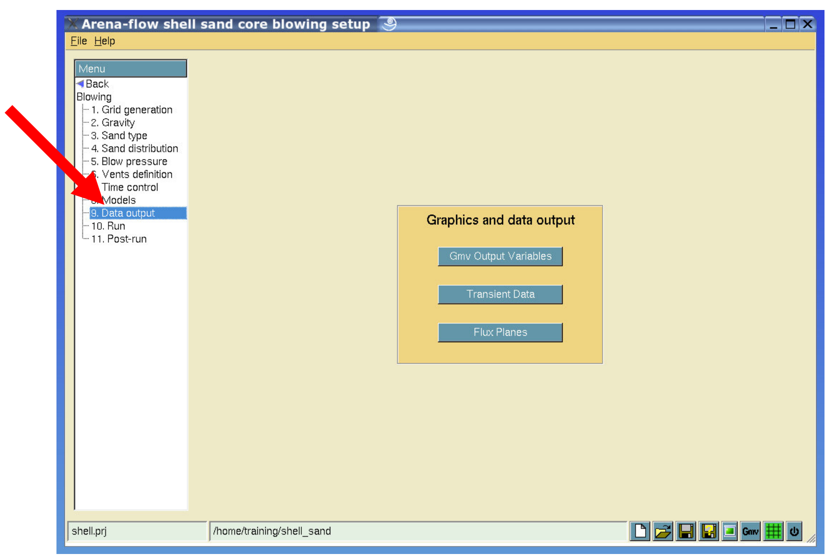


Figure 19: Shell sand flowability settings with core hardening during the blow time

Tip: Always consider the model assumptions when analyzing results.

It should be noted that thermal aspects of the hot tooling are not modeled by ***Arena-flow***®. ***Arena-flow***® models shell sand by setting wall bounce, particle interaction and flowability models appropriate to shell sand. As with any simulation, whenever analyzing results using the "***Arena-flow***® shell sand core blowing setup" interface, be sure to use sound judgment. If the calculation shows sand slowing or stopping before moving again, the hardening effects may be more important than modeled. Further, if this occurs in a thin section, the effects may be more severe still. Always consider the model assumptions when analyzing results.



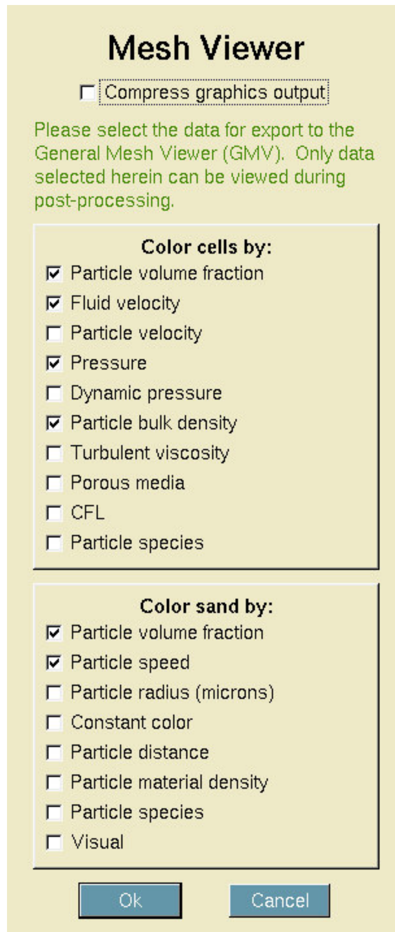
Click on "Data output" to go to the "Graphics and data output" page. The ***Arena-flow***® CPFD™ solver requires a great deal of fluid and particle data to accurately solve sand core engineering problems. The ***Arena-flow***® solver may use 1GB of memory, or more, to run large problems. If all that data were output in a transient manner, a single ***Arena-flow***® calculation could easily fill even a large hard drive.

Important: If you do not tell ***Arena-flow***® to write variables to the GMV output files, the information will not be there!

For efficient post-processing, you need to tell ***Arena-flow***® what information you wish to post-process, before starting your calculation. This is typically done through the "Mesh Viewer" window. Think about your anticipated results before running your calculation - if you do not ask for the data to be written to the GMV files, it will likely not be there!

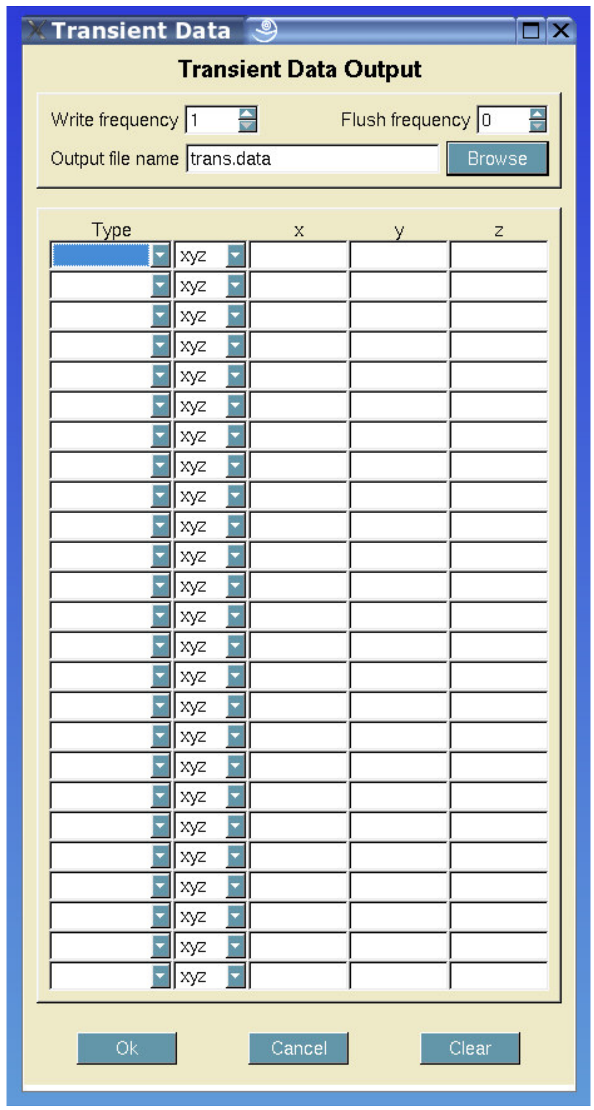
Tip: For cell data, it is good to select "Particle volume fraction", "Fluid velocity", "Pressure" and "Particle bulk density". Be sure to select "Particle species" if you are using different species of sand.

Tip: For sand data, it is good to select "Particle volume fraction" and "Particle speed". Be sure to select "Particle species" if you are using different species of sand. Be sure to select "Particle material density" if you are using sands with different material densities.



Click on "GMV Output Variables" to raise the "Mesh Viewer" window. Here you select what quantities you will wish to view during post-processing. Select the quantities by which you may wish to color your cells in the top of the window. Select the quantities by which you may wish to color your particles in the bottom of the window.

Typically, it is good practice to select “Particle volume fraction”, “Fluid velocity”, “Pressure”, and “Particle bulk density” for cell data and “Particle volume fraction” and “Particle speed” for sand data. If you defined multiple sand species, be sure to select “Particle species” for both the cells and the sand. If you are using sands with different material densities, be sure to select “Particle material density” for the sand data output.



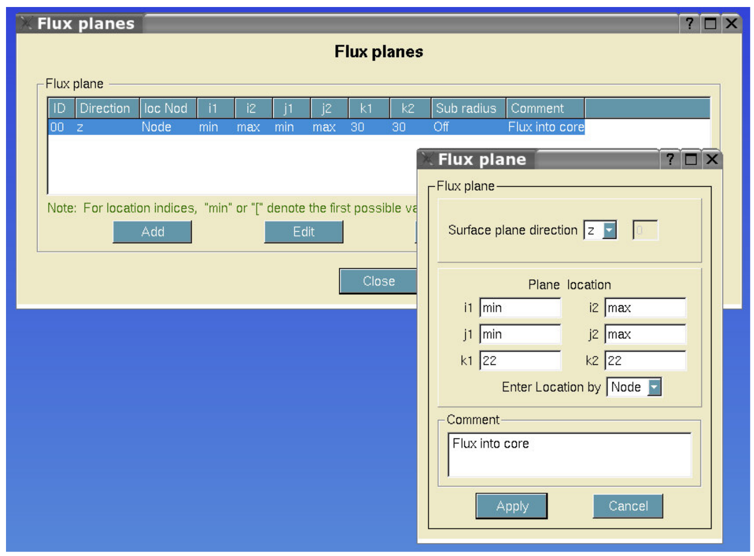
Note: Transient data specification is not required.

The specification of transient data is not required. The “Transient Data Output” window allows you to request that information from pre-identified cells be written to a file during the calculation.

To launch the "Transient Data Output" window, click on "Transient Data" on the "Graphics and data output" page. Specify an "Output file name"; the default is trans.data. Transient data output locations are specified by selecting the data type on the left, and identifying the location at the right. The location can be either spatial coordinates (x, y, z) or cell indices (i, j, k). Tell ***Arena-flow***® which you are using via the xyz / node drop down menu.

Arena-flow® will write the desired data to the output file in columnar form during the run. The output file is in a plain text format, and may be graphed via any plotting utility supporting columnar input. The "Transient Data Output" feature is an excellent way to monitor pressures at discrete locations in your system.

Note: Flux plane specification is not required.

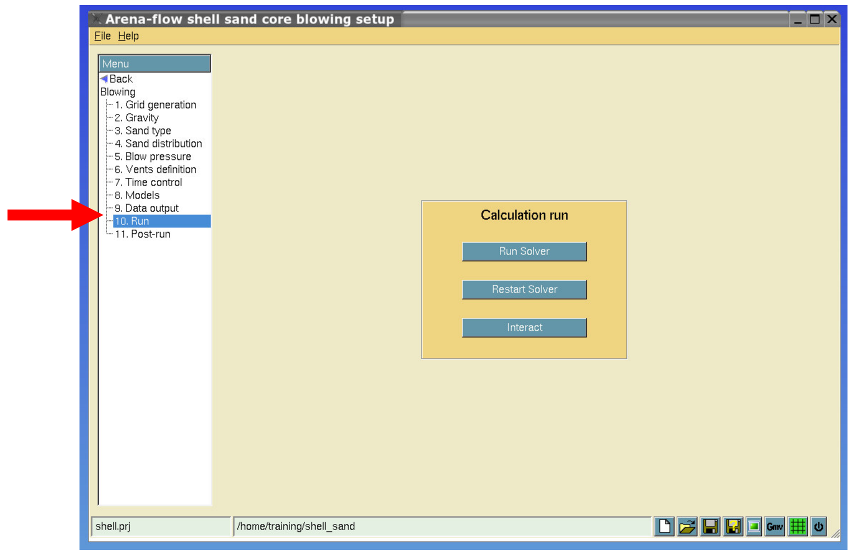


As with transient data, the specification of flux planes is also not required. Flux planes track all fluid and particles which cross them during the calculation.

To launch the "Flux planes" window, click on "Flux Planes" on the "Graphics and data output" page. Individual planes can be defined by clicking "Add".

To define a flux plane, select the "Surface plane direction" and the "Plane location". The "Plane location" can be entered in spatial coordinates (x, y, z) or cell indices (i, j, k – i.e. "Node"). Tell ***Arena-flow***® which you are using via the "Enter Location by" drop down menu.

Each flux plane creates a text file containing columnar data. The files will be named Flux_## where ## is the flux plane "ID". The output file is in a plain text format, and may be graphed via any plotting utility supporting columnar input. The "Flux plane" feature is an excellent way to monitor the flow through blow tubes.



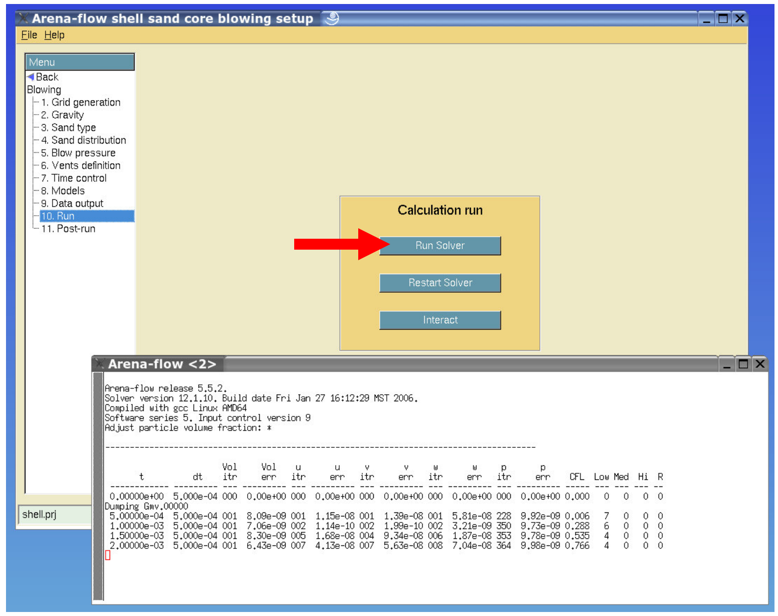
Tip: You can run the ***Arena-flow***® solver by clicking "Run Solver" in the GUI, or by typing arena.x at the command line.

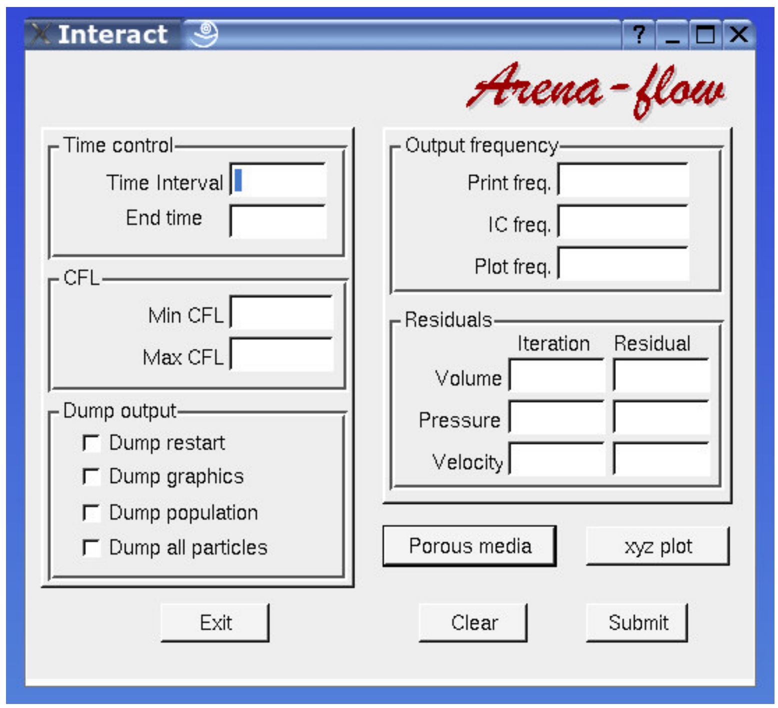
Click on "Run" to go to the "Calculation run" page. Once your calculation is fully set up, click on "Run Solver" to start the simulation. The solver may also be run from the command line with the "arena.x" command to allow for scripting. The argument is the project file name.

When you click on "Run Solver" in the GUI, a new window opens displaying solver information during execution. It is very important to leave this window open during the run. Closing this window will halt solver execution.

The first column of data output in the solver window shows the simulation time. More information regarding solver output is found in section 2.10, Additional Features.

Important: Be sure to leave the ***Arena-flow***® solver window open during the calculation. Closing it will stop the run.





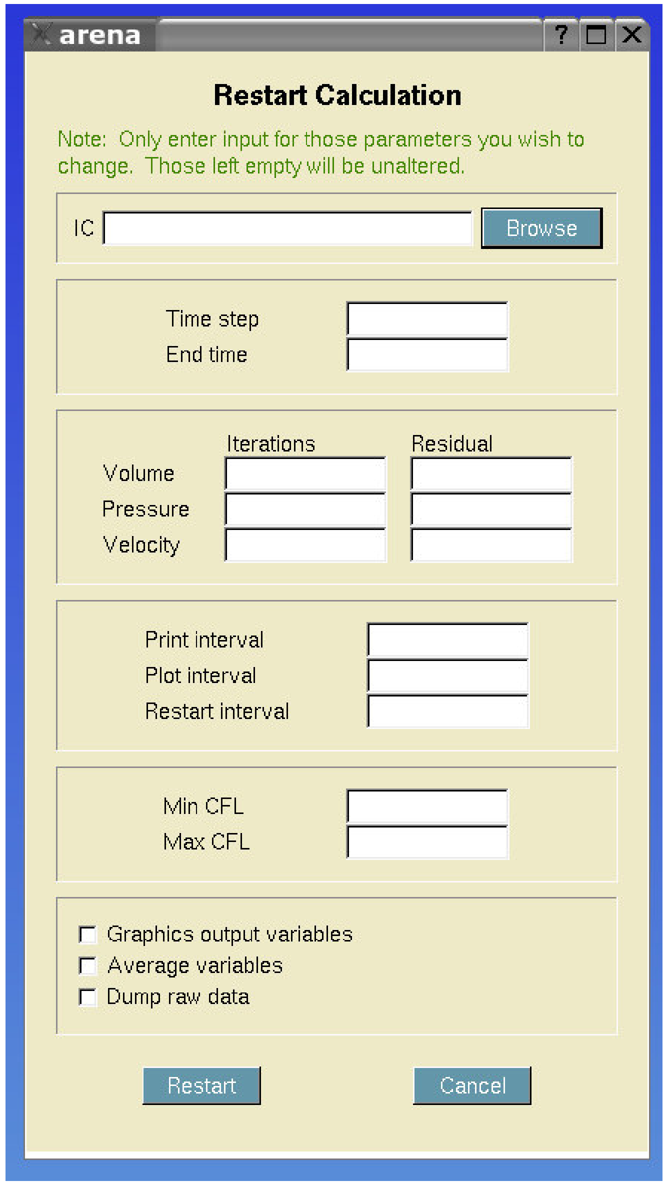
It is possible to change some solver parameters while the calculation is running. Click on “Interact” on the “Calculation run” page to raise the “Interact” window.

A good use of the “Interact” utility is to change the calculation end time during the run. For example, if your core is full, change the “End time” to the current time and click “Submit”. This will end the calculation. Alternately, if you are nearing your originally defined

Tip: Only change parameters (other than the "End time") if you are advised to do so by an ***Arena-flow***® support engineer.

end time and your core is still not full, extend the "End time" by entering a new value and click "Submit".

Only enter the information you wish to change. Use this utility cautiously, and only change other parameters if advised to do so by an ***Arena-flow***® support engineer. This utility is under redevelopment.

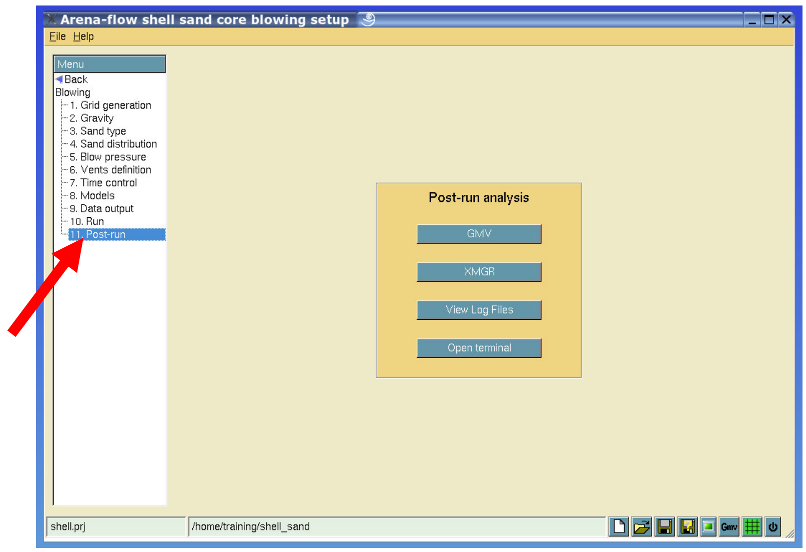


Note: It is possible to restart the solver after completion.

It is also possible to restart your calculation once it has completed. To do so, click on "Restart Solver" on the "Calculation run" page to raise the "Restart Calculation" window.

Tip: Only change parameters (other than the "End time") if you are advised to do so by an ***Arena-flow***® support engineer.

You must enter a restart file (IC). Restart file names start with "IC_" and may contain additional characters. Click on "Browse" to select the file. Only enter the information you wish to change from the original run. If you would like to run the calculation for longer than originally intended, please enter a new "End time". As with the "Interact" utility, please only change other parameters if advised to do so by an ***Arena-flow***® support engineer.



Tip: All of the utilities on the "Post-run analysis" page can be used while the calculation is running.

Click on "Post-run" to open the "Post-run analysis" page. All these features can be used while the calculation is running. The utilities are as follows:

Tip: GMV can be used to view your ***Arena-flow***® results, while the calculation is running!

- "GMV" is the general mesh viewer. This is used to post-process the graphical output from ***Arena-flow***®. The use of GMV is further described in section 2.9, Post-Processing. Using GMV, you can view your results while your calculation is running.

Note: You can use any plotting utility to graph columnar data. Use of XMGR is not required.

- "XMGR" is a plotting utility. It can be used to plot columnar data such as flux plane and transient data output files. XMGR is an open-source program. More information regarding the use of XMGR can be found in a web search. It is not required to use XMGR, please use your favorite plotting utility to graph columnar data.

- “View Log Files” launches a viewer to open text files. This can be used to view various output files.

Tip: Use the terminal window to enter additional Linux commands.

- “Open terminal” launches a Linux command prompt in the working directory to allow for the input of various commands.

Important: Check these things every time you start an ***Arena-flow***® shell sand core blowing calculation.

As soon as you start a shell sand core blowing calculation running, check the following:

- Check that your boundary conditions are applied in the correct location
- Check that your vents are applied correctly
- Check that your flux planes are defined as desired (if used)
- Check that your sand is in the correct location
- Check that your grid extends a few cells into the blow plate and that the particle “Feed” condition is applied to the “Blow pressure” boundary condition
- Check your flowability setting and consider how much core “hardening” you are simulating

Important: Check these things every time you analyze the results from an ***Arena-flow***® shell sand core blowing calculation.

Once your calculation is complete, be sure to view:

- Transient filling of the core
- Variations in the final core density
- Sand speed in regions of poor fill
- Air pressure and flow during filling
- Regions of likely tool wear

With shell sand core blowing, it is particularly important to look for regions where the sand stops (or slows) before moving again. If this happens, early hardening due to heating is possible.

More information regarding analyzing your results is presented in the various ***Arena-flow***® training courses. Please consult your training manual. Contact your sales representative or support engineer for more information regarding course content and schedule.

2.6 Gas Curing – Steady-State

In this chapter you will learn about:

- *Setting up a steady-state gas curing project*
- *Running a steady-state gas curing project*
- *Evaluating the results from a steady-state gas curing project*

Note: The steady-state gas curing module is used to quickly identify regions of poor air flow in filled tooling.

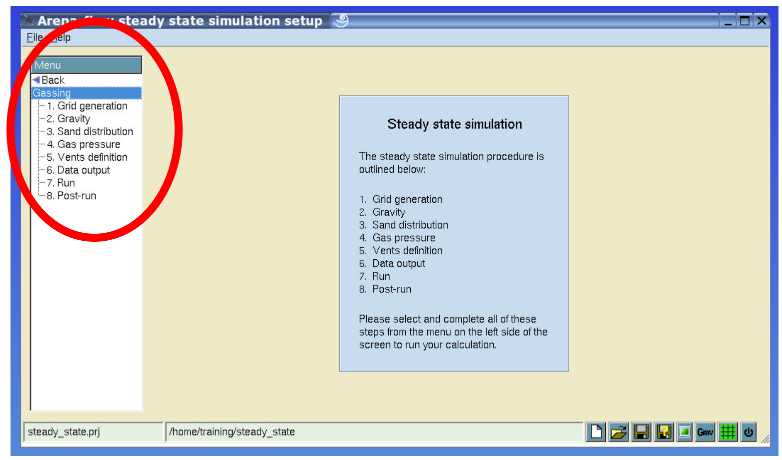
Note: The transient curing module is used to analyze the transient progression of the curing front through the core.

The ***Arena-flow***® gas curing module actually contains two separate analyses interfaces: the steady state gas curing module, and the transient gas curing module. These modules are separate for extreme computational efficiency, but the results are related. The steady-state analysis shows the air flow at the set gas / purge pressure through the filled core. It is used to quickly identify regions of poor air flow, which may be related to regions with a non-optimal curing behavior. The transient curing analysis shows the transient progression of the curing front through the core.

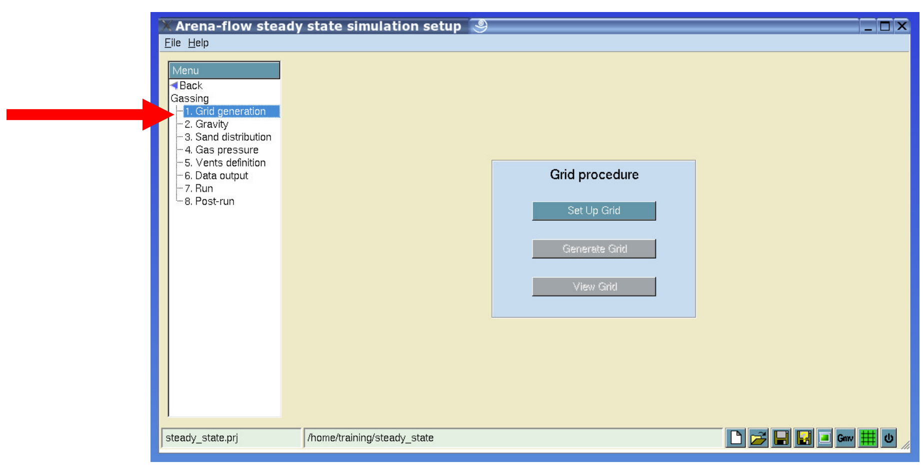
This chapter outlines the use of the steady-state curing module; transient curing is the subject of the next chapter. To model steady-state gas curing with ***Arena-flow***®, click on "Gas curing" as shown:

Select "Steady state gasflow analysis" and create or open a project file in the desired working directory to launch the "***Arena-flow***® steady-state simulation setup" window. To set up and run a steady-state gas curing project, work through the 8 menu items shown in the tree on the left of the window.

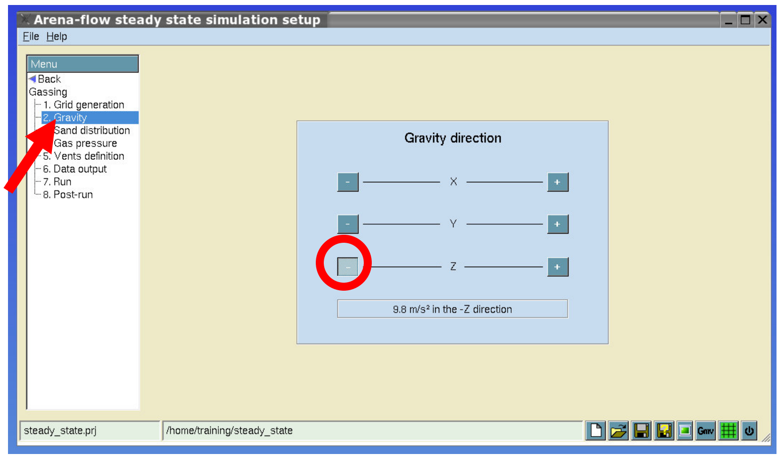
Tip: To set up and run a steady-state gas curing project, work through the 8 menu items shown in the tree on the left of the window.



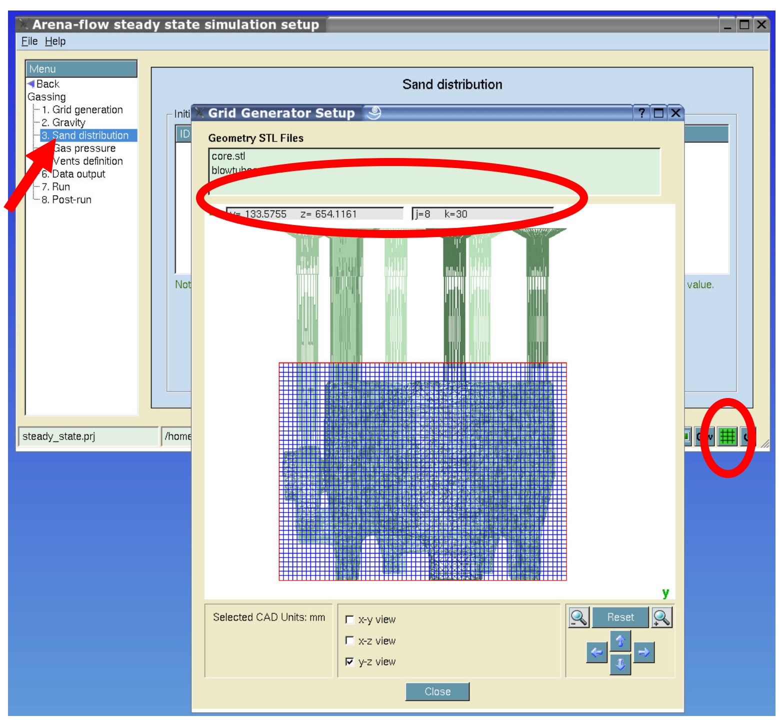
The first step to creating any ***Arena-flow***® model is mesh generation. Click on "Grid generation" to go to the "Grid procedure" page.



On the "Grid procedure" page, create and evaluate the grid by working through the three steps: "Set Up Grid", "Generate Grid" and "View Grid". The use of the ***Arena-flow***® grid generator for steady-state calculations is outlined in section 2.3.3 of this User Guide. Proper grid generation techniques are presented in the various ***Arena-flow***® training classes. Classes are held regularly. Contact your sales representative or support engineer for more information regarding course content and schedule.



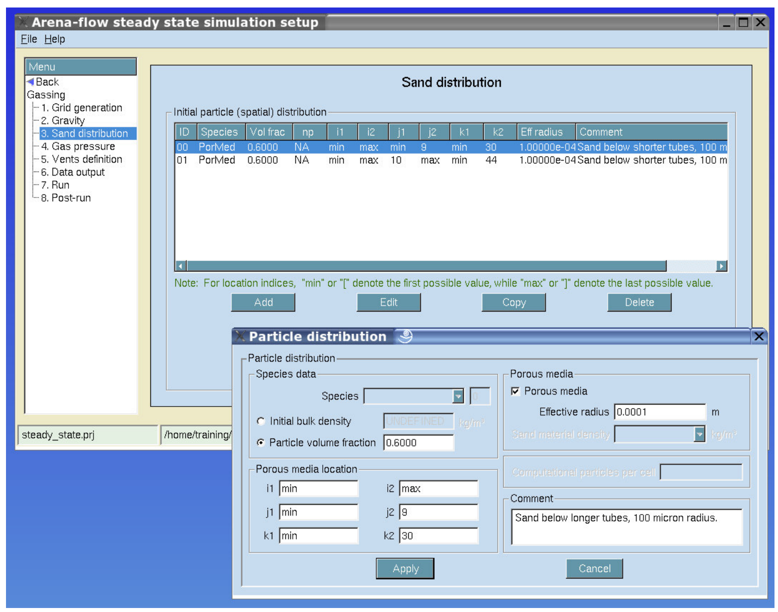
Click on the next menu item, "Gravity". This is where you tell ***Arena-flow***® the orientation of your CAD, relative to gravity. In this example, the CAD is oriented such that the z-axis is up. Thus, the gravity vector should point in the –z direction



> **Tip:** You can always refer to your grid by clicking on the reference grid button - .

Next, click on “Sand distribution” to go to the “Sand distribution” page. This is where you tell ***Arena-flow***® the sand location and its properties.

The location of the sand is related to the grid you defined in step 1. You may reference this grid by clicking on the button on the bottom right of the screen. This opens a non-editable version of the “Grid Generator Setup” screen. As you move your mouse over your grid, the coordinates (x, y, z) and cell indices (i, j, k) are displayed at the top of the window. Note that a two-dimensional plane is shown, thus only two of the three coordinates and grid indices are shown at one time. Determine which three-dimensional block of cells you wish to initialize with particles.



To place the sand in the cells, click on “Add” to raise the “Particle distribution” window. First specify either the “Initial bulk density” or “Particle volume fraction” of the sand. Since the core is full, this will be the fully-packed average core density or volume fraction. A typical “Particle volume fraction” is 0.6.

Next you must define the location of the sand. The “Porous media location” asks for six values, corresponding to the minimum and maximum cell indices in the x, y and z directions defining the three-

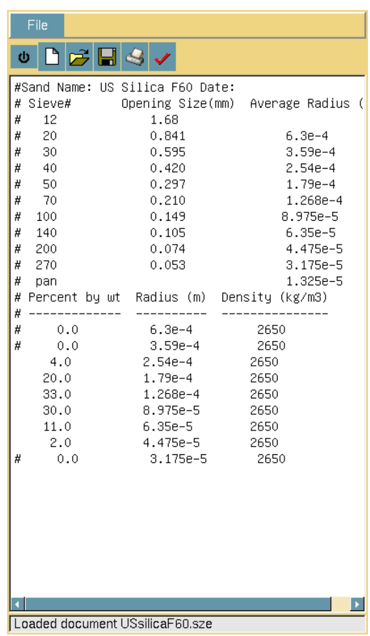
Tip: The “min” and “max” keywords can be used instead of the minimum and maximum cell or face indices whenever defining a boundary or initial condition.

dimensional block of cells containing the sand. These cell indices can be determined by using the reference grid tool, , however, the keywords “min” and “max” can also be used to reference the minimum and maximum cell index in each respective direction. Define the sand to be in the core, but not in the blow tubes / gassing head.

The stead-state simulation does not permit the sand to move; it is used to simulate an already filled core. Since the sand is fixed, a porous media is used to include the effects of the sand in the core.

Tip: Choose an “Effective radius” that is slightly smaller than your average grain size.

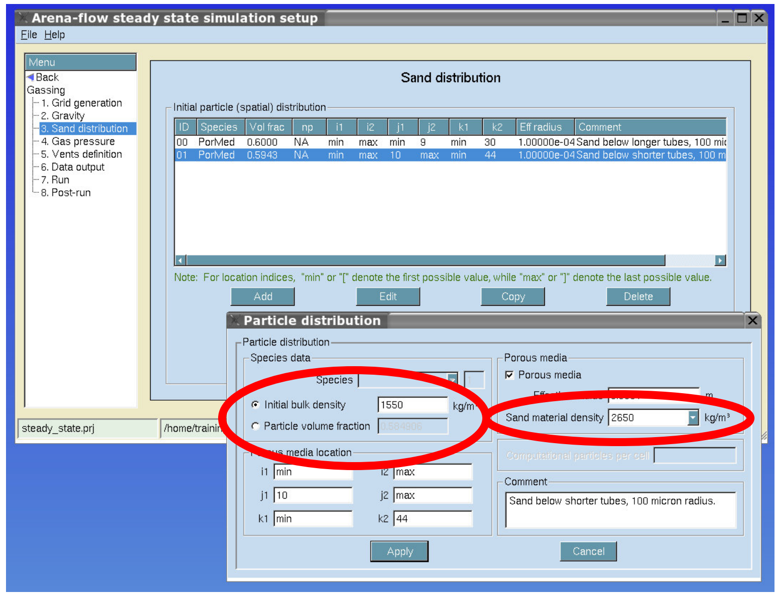
To use the ***Arena-flow***® porous media model, enter an “Effective radius” for the sand. The effective radius is used to control the pressure drop relation for gas flow through the sand. Since sand typically has a size distribution, a single, effective radius must be determined. Experience has shown that the finer sand grains influence the pressure drop more than the coarser ones. It is good practice to use a sand size that is slightly finer than the average grain size. Typically a value that is between 30% – 45% up the cumulative sand size distribution curve is a good choice.


```
#Sand Name: US Silica F60 Date:
# Sieve#       Opening Size(mm)  Average Radius (
#   12             1.68
#   20             0.841              6.3e-4
#   30             0.595              3.59e-4
#   40             0.420              2.54e-4
#   50             0.297              1.79e-4
#   70             0.210              1.268e-4
#  100             0.149             8.975e-5
#  140             0.105              6.35e-5
#  200             0.074              4.475e-5
#  270             0.053              3.175e-5
#  pan                                1.325e-5
# Percent by wt  Radius (m)  Density (kg/m3)
# -------------  ----------  ---------------
#     0.0          6.3e-4        2650
#     0.0          3.59e-4       2650
     4.0         2.54e-4       2650
    20.0         1.79e-4       2650
    33.0         1.268e-4      2650
    30.0         8.975e-5      2650
    11.0         6.35e-5       2650
     2.0         4.475e-5      2650
#     0.0          3.175e-5      2650
```


For example, in the sand size distribution shown above, 43% of the sand has a size of 89.75 µm or less. Thus a value between 80 and 90 µm would be a good choice of an "Effective radius" for this sand.

Tip: Enter comments whenever possible – it will save you time in the long run.

Lastly you are permitted to enter a "Comment" for the distribution. This is not used by the solver, but can be extremely useful when setting up large projects.

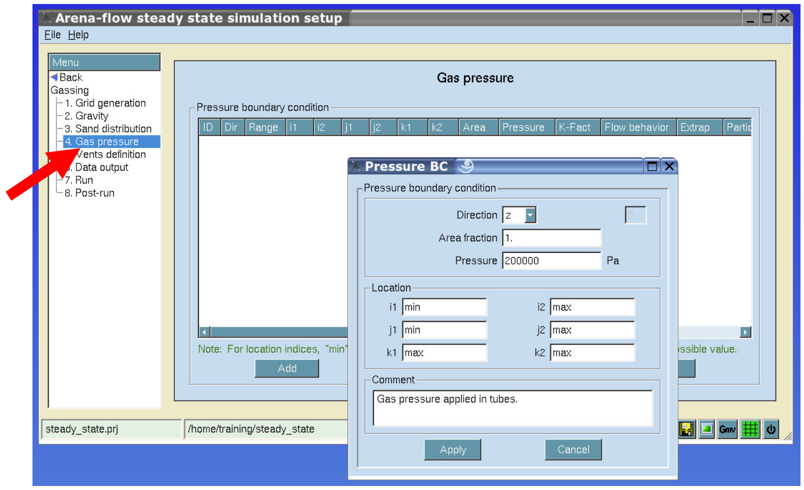


Note: If an "Initial bulk density" is entered, a "Sand material density" is also required to fully specify the problem.

An "Initial bulk density" may be defined instead of a "Particle volume fraction". The "Initial bulk density" is the average density of the core. If an "Initial bulk density" is used, a "Sand material density" must also be entered to fully specify the problem. Choose a predefined sand from the "Sand material density" drop-down list, or type the density into the field.

Once a sand distribution is defined using the "Add" button, it appears in the "Initial particle (spatial) distribution" table on the "Sand distribution" page. These distributions can then be edited using the "Edit" button, reproduced using the "Copy" button or deleted using the "Delete" button.

After the "Sand distribution" is set, click on "Gas pressure" to go to the "Gas pressure" page. Here you tell ***Arena-flow***® what pressure will be used and where the pressure is to be applied.



Click on "Add" to raise the "Pressure BC" window.

Select the "Direction" from the drop-down menu. This tells ***Arena-flow***® upon which faces of the cells the pressure boundary condition is to be applied. If your CAD was designed with the z-axis pointing up and your gas pressure is applied at the top of the model (in the blow tubes) then the default directional value of "z" can be left unchanged.

Enter the "Area fraction". This is the fraction of the cell faces which is open to the pressure. Usually, this is left at the default value of 1.0.

Tip: Many conversion factors to SI units can be found by clicking on "Help", "Units Reference".

Enter the "Pressure". This is the gas pressure in Pa. To determine the appropriate conversion factor when converting from alternate pressure measurements to Pa, click on "Help", "Units Reference". The pressure is constant (not time-varying) since this is a steady-state calculation. Typically this pressure represents the maximum gas or purge pressure of the system.

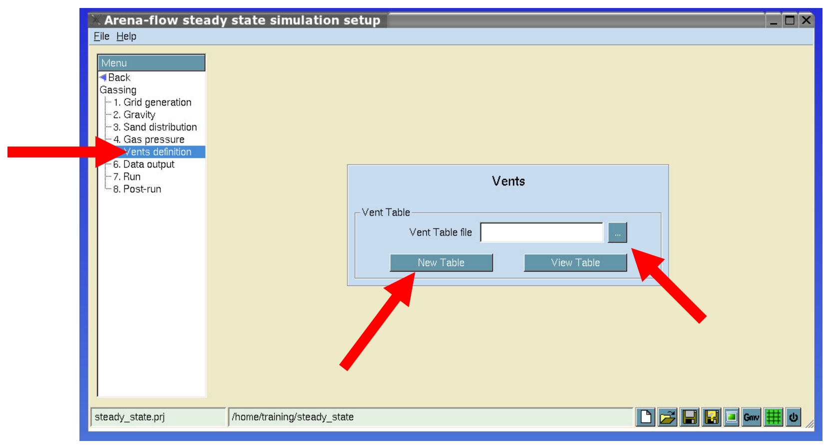
Important: The CFD component of ***Arena-flow***® is an incompressible solver; thus all pressures are gauge pressures with zero (0) representing the vented, atmospheric pressure.

Please note that the CFD component of ***Arena-flow***® is an incompressible solver. As such, all pressure measurements are relative. Calculations are typically set up with a zero (0) pressure at the vents. Thus the gas pressure should be in gauge pressure, relative to the vents.

Next specify the location of the gas pressure boundary condition. These are boundary cell indices and they must define a plane. You may use the reference grid utility, , to help you determine proper values for the indices. Additionally, the “min” and “max” keywords can also be used to reference the minimum and maximum cell index in each respective direction.

Sometimes multiple planes are required to fully specify the gas pressure inlet location. These boundary conditions can be edited using the “Edit” button, reproduced using the “Copy” button or deleted using the “Delete” button.

Finally, you are permitted to enter a “Comment” for the distribution.

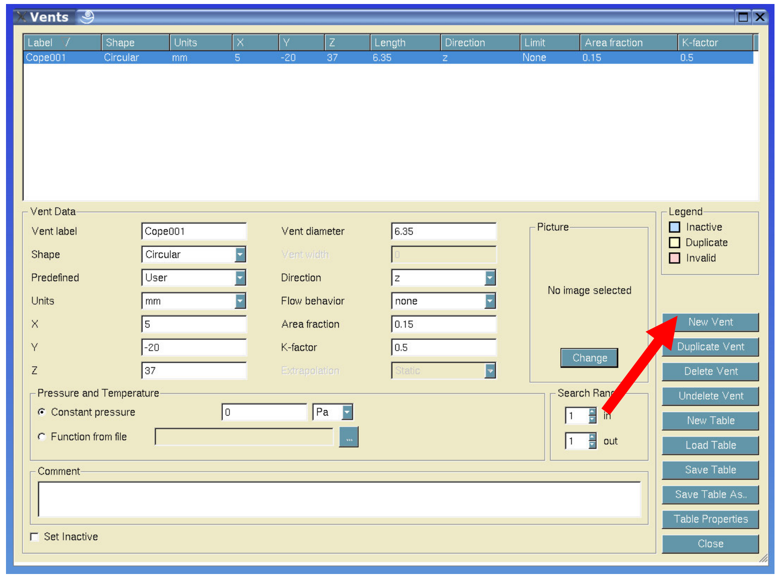


Click on “Vents definition” to go to the “Vents” page. Vents are added through a vent file. To create a vent file, click on “New Table”. To link to an existing vent file, click on the browse, , button. Vent files typically are identified with the .tvt extension.

New vent tables contain no information. Click on “New Vent” to begin populating the vent table. Much of the information is defaulted for you

Tip: Enter a “Vent label” to easily identify individual vents.

Enter a “Vent label” if desired. The vent label is a comment that helps to uniquely identify individual vents. This greatly simplifies later design permutations.



Note: The "Units" drop-down list affects both the location and "Vent diameter"

Enter the location of the vent center ("X", "Y", "Z"). Note that you may use non-SI units to do so, as indicated by the "Units" drop-down list.

Enter a "Vent diameter". The "Vent diameter" must also be entered with the same "Units" as the location.

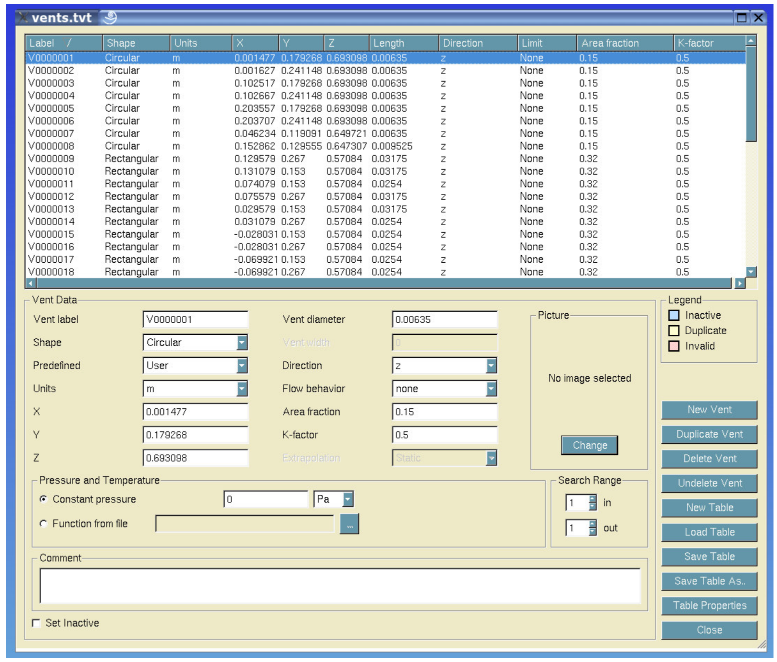
Enter a "Direction". This is the direction normal to the vent face. Typically cope and drag vents are in the z-direction if the model is oriented with the z-axis pointing up.

Enter an "Area fraction". This is the fraction of the vent area that is open for the air flow. Typically vent open area fractions are between 0.1 and 0.35.

Usually all other values remain unchanged. To set a vent inactive, click on "Set Inactive". This is a useful feature for design permutations.

Note: For steady-state calculations only "Constant pressures" are permitted.

Note that time-varying pressures are not permitted with the steady-state solver used by the ***Arena-flow***® steady-state gas curing module. Only constant pressures may be applied to the vents as shown below.



An example of a populated vent table is given above. To change any parameters of an existing vent, click on the vent to load the parameters into the display region.

Remember to save your vent table before exiting.

Note: Steady-state gas curing calculations do not model the amine, but rather the driving air flow.

Note that the steady-state calculation does not directly model the amine, but rather the driving air flow. Thus, it is possible to use the steady-state gas curing module to model the air flow during the blow cycle. The sand motion will not be modeled, but the calculation will give a quick, rough estimate of the air flow through the tooling. To do this, simply set your gas pressure to your blow pressure and using the blowing configuration of your vent file.

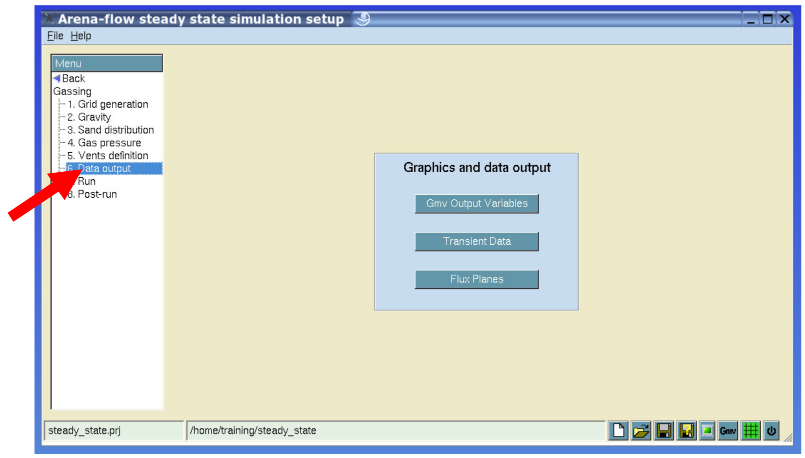
Note: For steady-state calculations, only two GMV files are created – one each at the start and end of the simulation.

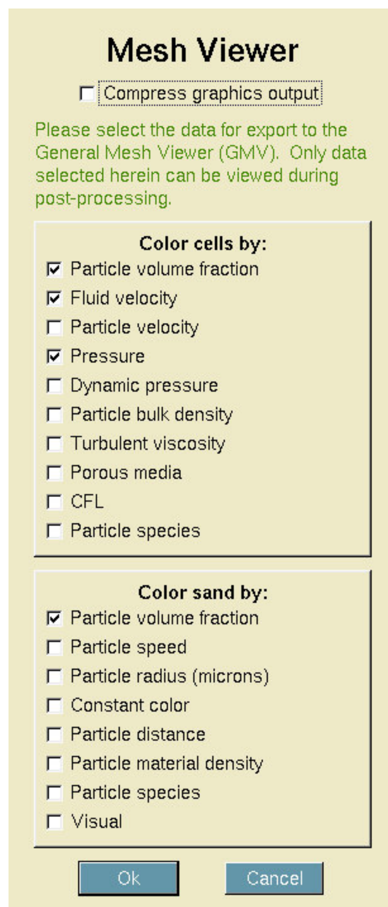
Also note that there is no “Time control” information in the ***Arena-flow***® steady-state gas curing module. The computational time is pre-set and no transient snapshots of the solution are produces. Only two GMV files are created - one at the start of the calculation (Gmv.00000) and one upon completion (Gmv.00001).

Click on “Data output” to go to the “Graphics and data output” page. For efficient post-processing, you need to tell ***Arena-flow***® what information you wish to post-

Important: If you do not tell ***Arena-flow***® to write variables to the GMV output files, the information will not be there!

process, before starting your calculation. This is typically done through the "Mesh Viewer" window. Think about your anticipated results before running your calculation - if you do not ask for the data to be written to the GMV files, it will likely not be there!

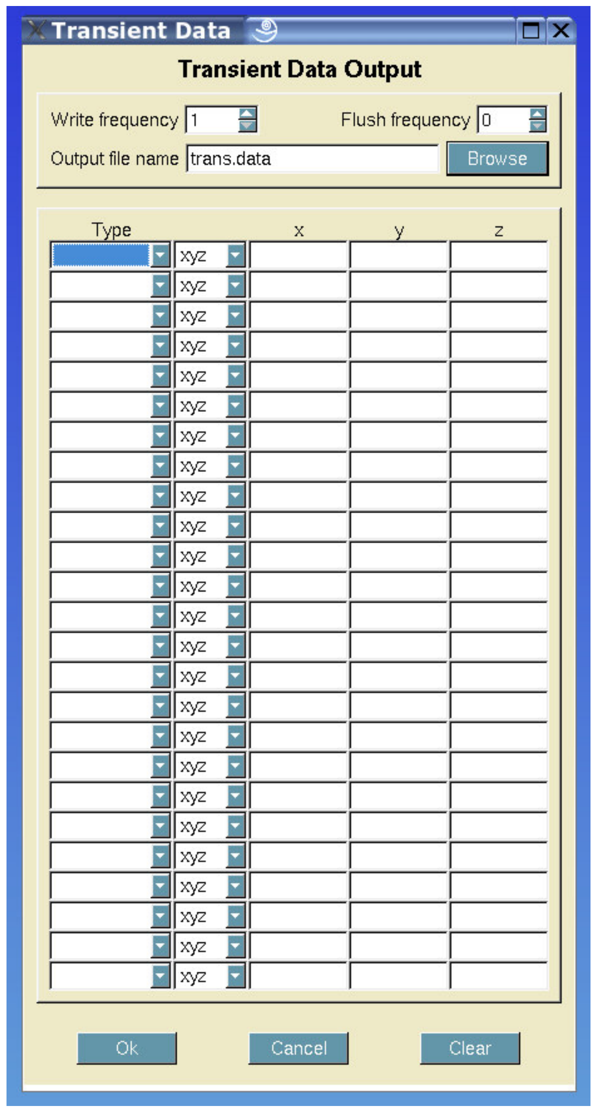




Tip: For cell data, it is good to select "Particle volume fraction", "Fluid velocity" and "Pressure for steady-state calculations.

Click on “GMV Output Variables” to raise the “Mesh Viewer” window. Here you select what quantities you will wish to view during post-processing. Select the quantities by which you may wish to color your cells in the top of the window. Select the quantities by which you may wish to color your particles in the bottom of the window.

Typically, it is good practice to select “Particle volume fraction”, “Fluid velocity” and “Pressure” for cell data and “Particle volume fraction” and for sand data.



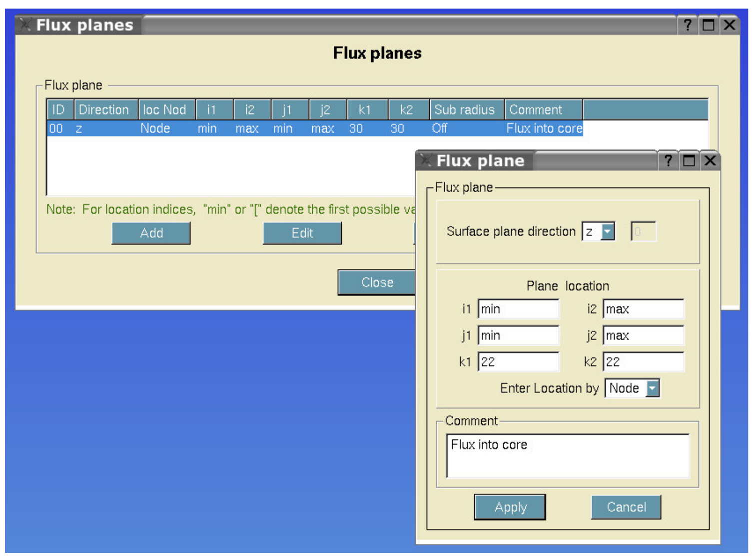
Note: Transient data specification is not required.

The specification of transient data is not required. The “Transient Data Output” window allows you to request that information from pre-identified cells be written to a file during the calculation. Even though you are running a steady-state calculation, you may monitor the convergence of the solution via the “Transient Data Output” feature.

To launch the “Transient Data Output” window, click on “Transient Data” on the “Graphics and data output” page. Specify an “Output file name”; the default is trans.data. Transient data output locations are specified by selecting the data type on the left, and identifying the location at the right. The location can be either spatial coordinates (x, y, z) or cell indices (i, j, k). Tell ***Arena-flow***® which you are using via the xyz / node drop down menu.

Arena-flow® will write the desired data to the output file in columnar format during the run. The output file is in a plain text format, and may be graphed via any plotting utility supporting columnar input. The “Transient Data Output” feature is an excellent way to monitor pressures at discrete locations in your system.

Note: Flux plane specification is not required.

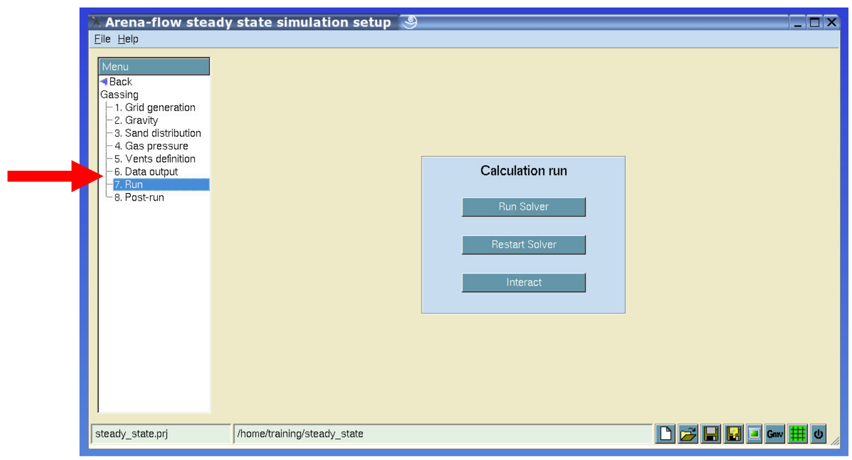


As with transient data, the specification of flux planes is also not required. Flux planes track all fluid and particles which cross them during the calculation. Flux planes are another way to monitor the convergence of the solution. Once the fluxes no longer change, a steady-state has been reached.

To launch the “Flux planes” window, click on “Flux Planes” on the “Graphics and data output” page. Individual planes can be defined by clicking “Add”.

To define a flux plane, select the “Surface plane direction” and the “Plane location”. The “Plane location” can be entered in spatial coordinates (x, y, z) or cell indices (i, j, k – i.e. “Node”). Tell ***Arena-flow***® which you are using via the “Enter Location by” drop down menu.

Each flux plane creates a text file containing columnar data. The files will be named Flux_## where ## is the flux plane “ID”. The output file is in a plain text format, and may be graphed via any plotting utility supporting columnar input. The “Flux plane” feature is an excellent way to monitor the flow through blow tubes.

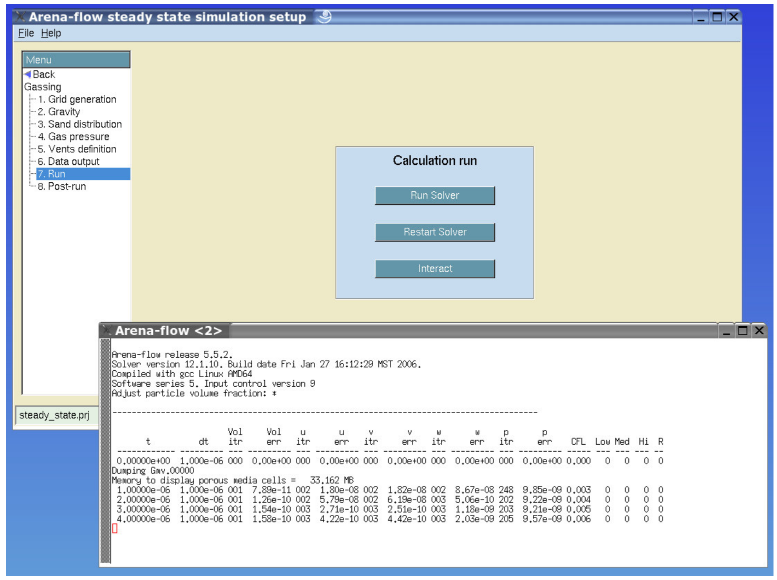


Tip: You can run the ***Arena-flow***® solver by clicking “Run Solver” in the GUI, or by typing arena.x at the command line.

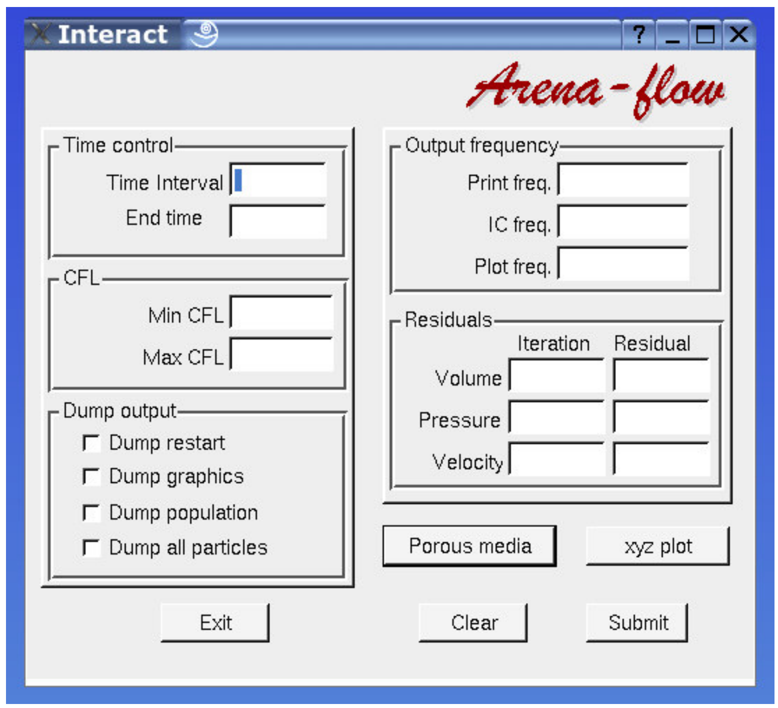
Click on “Run” to go to the “Calculation run” page. Once your calculation is fully set up, click on “Run Solver” to start the simulation. The solver may also be run from the command line with the “arena.x” command to allow for scripting. The argument is the project file name.

When you click on “Run Solver” in the GUI, a new window opens displaying solver information during execution. It is very important to leave this window open during the run. Closing this window will halt solver execution.

The first column of data output in the solver window shows the simulation time. More information regarding solver output is found in section 2.10, Additional Features.



Important: Be sure to leave the ***Arena-flow***® solver window open during the calculation. Closing it will stop the run.

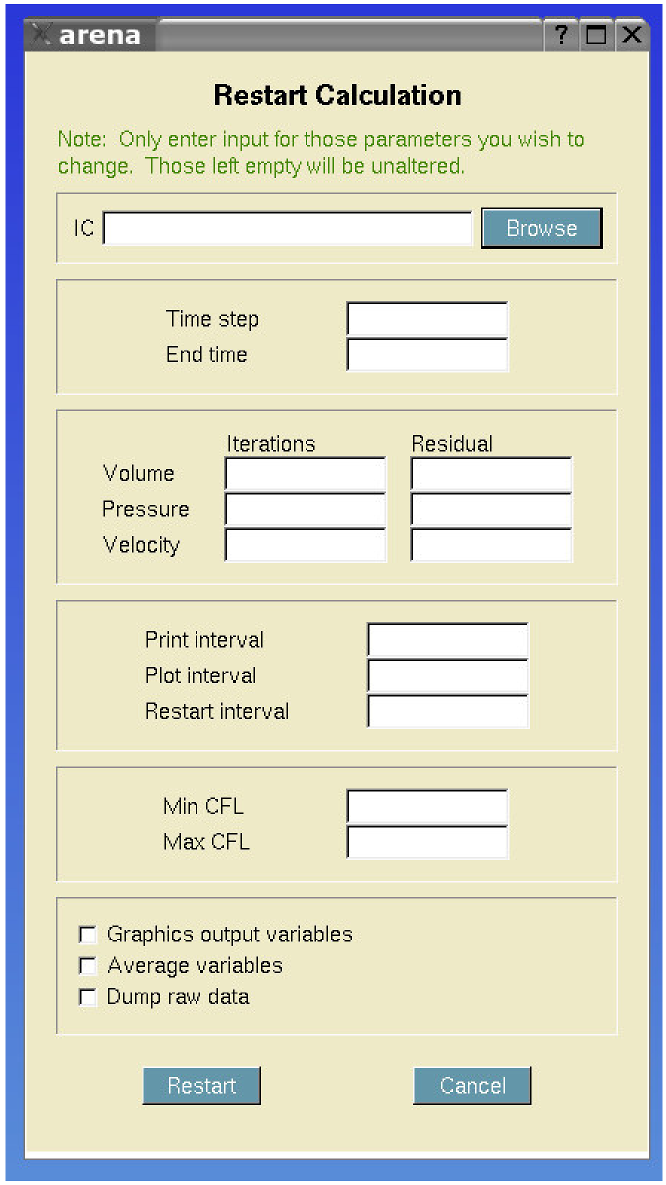


Note: It is possible to change some solver parameters while the calculation is running.

Tip: Only change parameters (other than the "End time") if you are advised to do so by an ***Arena-flow***® support engineer.

It is possible to change some solver parameters while the calculation is running. Click on "Interact" on the "Calculation run" page to raise the "Interact" window.

A good use of the "Interact" utility for steady-state gas flow calculations is to view intermediate GMV results while the calculation is running. To do so, select "Dump graphics" and click on Submit. When using the "Interact" utility, only enter the information you wish to change. Use this utility cautiously, and only change other parameters if advised to do so by an ***Arena-flow***® support engineer. This utility is under redevelopment.

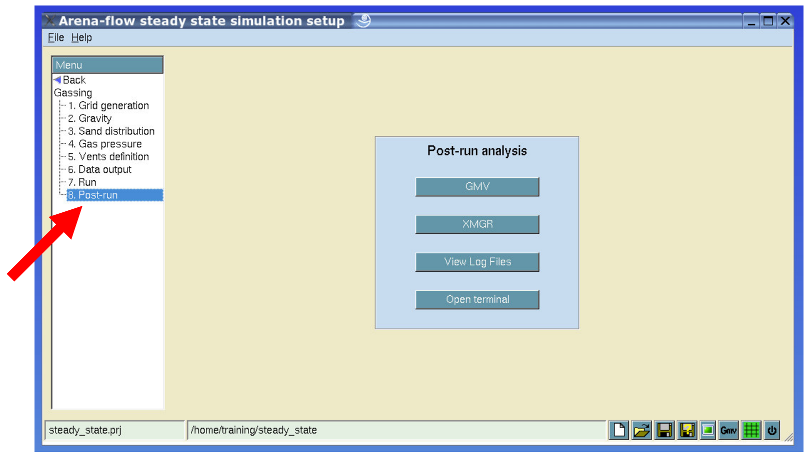


Note: It is possible to restart the solver after completion. This feature is not often used with the steady-state gas curing module.

It is also possible to restart your calculation once it has completed. To do so, click on "Restart Solver" on the "Calculation run" page to raise the "Restart Calculation" window.

You must enter a restart file (IC). Restart file names start with "IC_" and may contain additional characters. Click on "Browse" to select the file. Only enter the information you wish to change from the original run.

The restart feature is not often used with steady-state gas curing calculations. As with the “Interact” utility, please only change other parameters if advised to do so by an ***Arena-flow***® support engineer.



Tip: All of the utilities on the “Post-run analysis” page can be used while the calculation is running.

Click on “Post-run” to open the “Post-run analysis” page. All these features can be used while the calculation is running. The utilities are as follows:

- “GMV” is the general mesh viewer. This is used to post-process the graphical output from ***Arena-flow***®. The use of GMV is further described in section 2.9, Post-Processing. Using GMV, you can view your results while your calculation is running.

Tip: GMV can be used to view your ***Arena-flow***® results, while the calculation is running!

- “XMGR” is a plotting utility. It can be used to plot columnar data such as flux plane and transient data output files. XMGR is an open-source program. More information regarding the use of XMGR can be found in a web search. It is not required to use XMGR, please use your favorite plotting utility to graph columnar data.

Note: You can use any plotting utility to graph columnar data. Use of XMGR is not required.

- “View Log Files” launches a viewer to open text files. This can be used to view various output files.

- “Open terminal” launches a Linux command prompt in the working directory to allow for the input of various commands.

Tip: Use the terminal window to enter additional Linux commands.

Important: Check these things every time you start an ***Arena-flow***® steady-state gas curing calculation.

As soon as you start a steady-state gas curing calculation running, check the following:

- Check that your boundary conditions are applied in the correct location
- Check that your vents are applied correctly
- Check that your flux planes are defined as desired (if used)
- Check that your sand is in the correct location, namely in the core and not in the gassing head

Important: Check these things every time you analyze the results from an ***Arena-flow***® steady-state gas curing calculation.

Once your calculation is complete, be sure to view:

- Regions of low air flow
- Air flow patterns throughout the core
- Pressure distribution throughout the core

More information regarding analyzing your results is presented in the various ***Arena-flow***® training courses. Please consult your training manual. Contact your sales representative or support engineer for more information regarding course content and schedule.

2.7 Gas Curing – Transient Curing

In this chapter you will learn about:

- *Setting up a transient gas curing project*
- *Running a transient gas curing project*
- *Evaluating the results from a transient gas curing project*

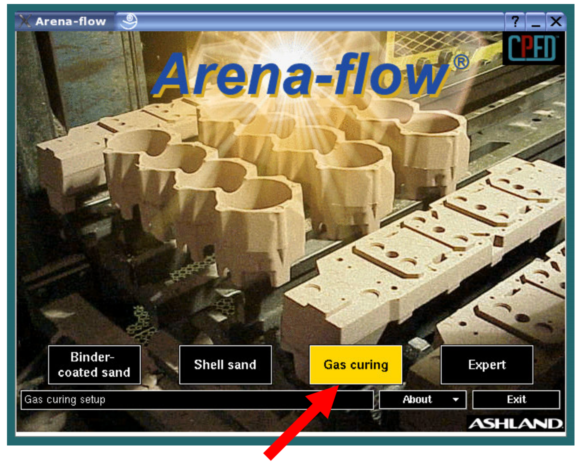
Note: The steady-state gas curing module is used to quickly identify regions of poor air flow in filled tooling.

Note: The transient curing module is used to analyze the transient progression of the curing front through the core.

The ***Arena-flow***® gas curing module actually contains two separate analyses interfaces: the steady state gas curing module, and the transient gas curing module. These modules are separate for extreme computational efficiency, but the results are related. The steady-state analysis is used to quickly identify the air flow patterns through the filled core. The transient curing analysis shows the transient progression of the curing front through the core. By optimizing the progression of the curing front, cores are usually cured in the minimal time with the minimal use of consumables.

This chapter outlines the use of the transient curing module; steady-state analysis is the subject of the previous chapter.

To model transient gas curing with ***Arena-flow***®, click on "Gas curing" as shown:

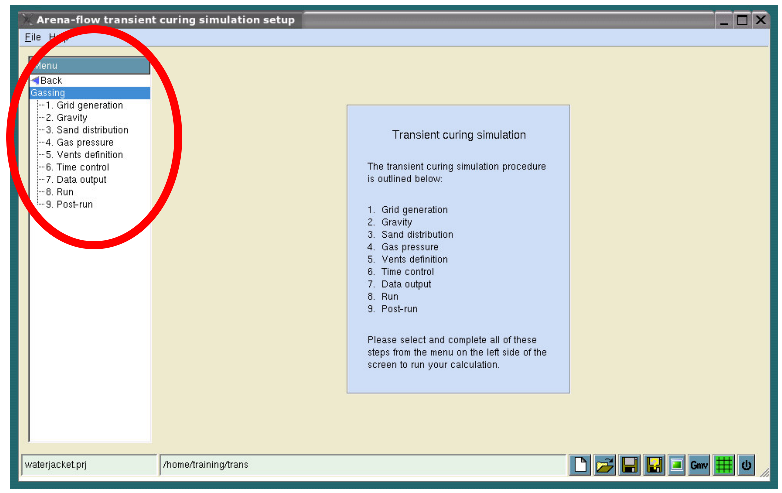


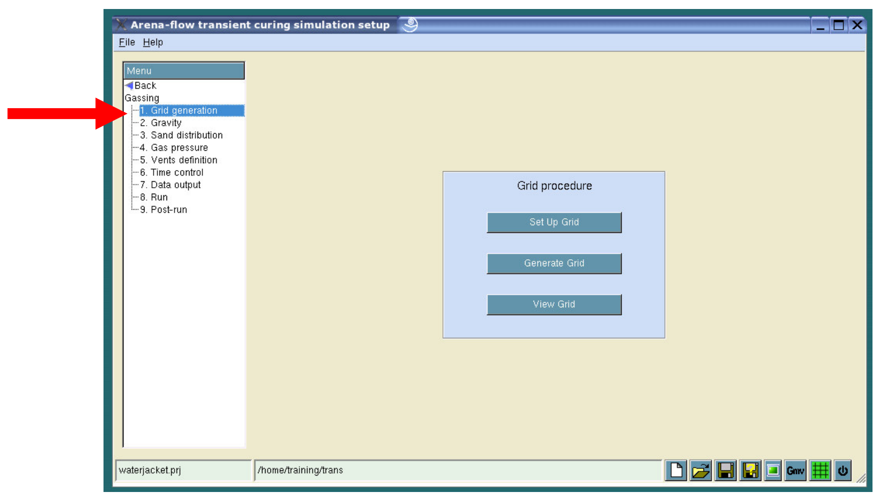
Select "Transient curing analysis" and create or open a project file in the desired working directory to launch the "***Arena-flow***® steady-state simulation setup" window.

To set up and run a transient gas curing project, work through the 9 menu items shown in the tree on the left of the window.

The first step to creating any ***Arena-flow***® model is mesh generation. Click on "Grid generation" to go to the "Grid procedure" page.

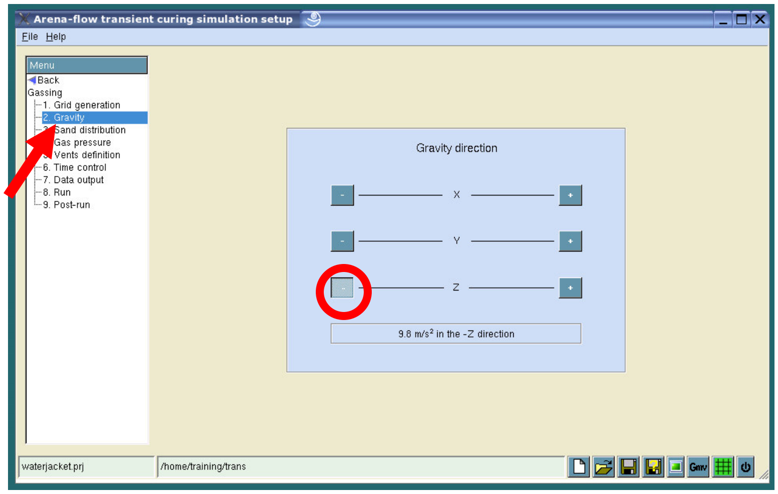
Tip: To set up and run a transient gas curing project, work through the 9 menu items shown in the tree on the left of the window.



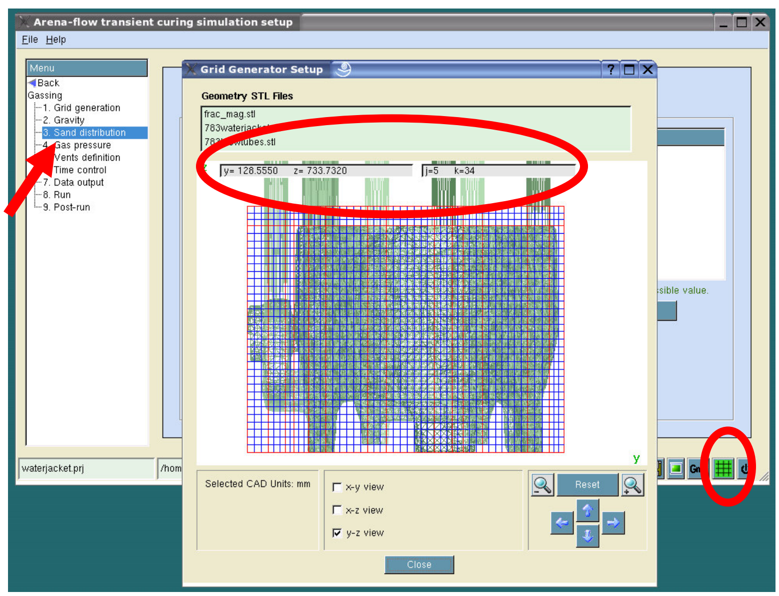


On the “Grid procedure” page, create and evaluate the grid by working through the three steps: “Set Up Grid”, “Generate Grid” and “View Grid”. The use of the ***Arena-flow***® grid generator for steady-state calculations is outlined in section 2.3.3 of this User Guide. Proper grid generation techniques are presented in the various ***Arena-flow***® training classes. Classes are held regularly. Contact your sales representative or support engineer for more information regarding course content and schedule.

Click on the next menu item, “Gravity”. This is where you tell ***Arena-flow***® the orientation of your CAD, relative to gravity. In this example, the CAD is oriented such that the z-axis is up. Thus, the gravity vector should point in the –z direction



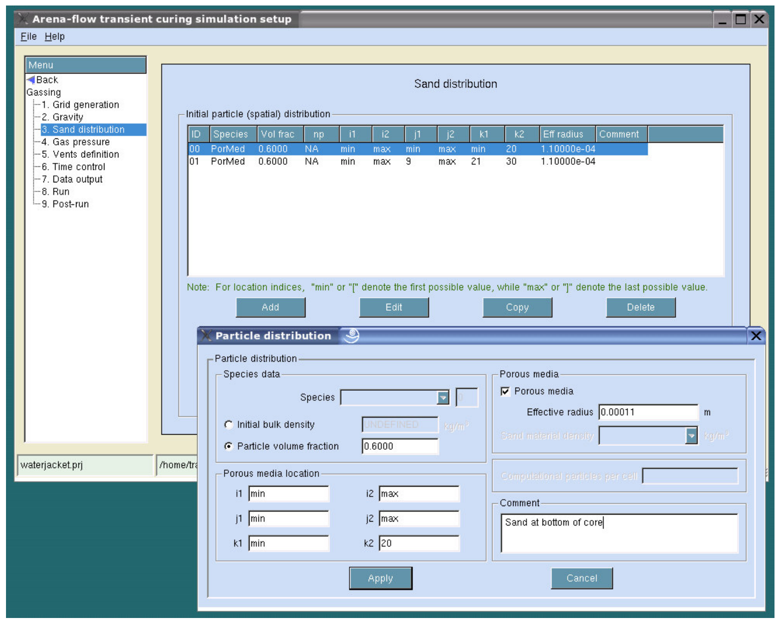
Next, click on “Sand distribution” to go to the “Sand distribution” page. This is where you tell ***Arena-flow***® the sand location and its properties.



Tip: You can always refer to your grid by clicking on the reference grid button - .

The location of the sand is related to the grid you defined in step 1. You may reference this grid by clicking on the button on the bottom right of the screen. This opens a non-editable version of the “Grid Generator Setup” window. As you move your mouse over your grid, the coordinates (x, y, z) and cell indices (i, j, k) are displayed at the top of the window. Note that a two-dimensional plane is shown, thus only two of the three coordinates and grid indices are shown at one time. Determine which three-dimensional block of cells you wish to initialize with particles.

To place the sand in the cells, click on “Add” to raise the “Particle distribution” window. First specify either the “Initial bulk density” or “Particle volume fraction” of the sand. Since the core is full, this will be the fully-packed average core density or volume fraction. A typical “Particle volume fraction” is 0.6.



Next you must define the location of the sand. The “Porous media location” asks for six values, corresponding to the minimum and maximum cell indices in the x, y and z directions defining the three-dimensional block of cells containing the sand. These cell indices can be determined by using the reference grid tool, ▦, however, the keywords “min” and “max” can also be used to reference the minimum and maximum cell index in each respective direction. Define the sand to be in the core, but not in the blow tubes / gassing head.

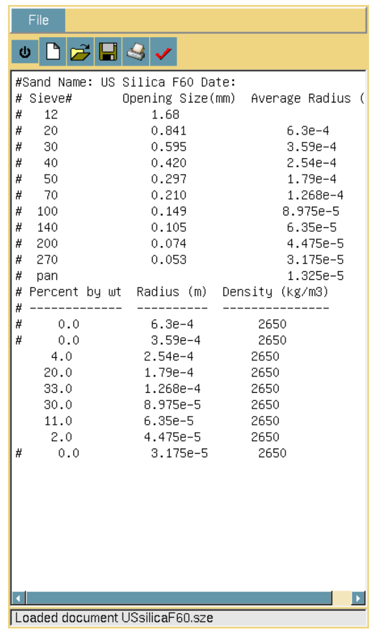
Tip: The “min” and “max” keywords can be used instead of the minimum and maximum cell or face indices whenever defining a boundary or initial condition.

The transient curing simulation does not permit the sand to move; it is used to simulate an already filled core. Since the sand is fixed, a porous media is used to include the effects of the sand in the core.

To use the ***Arena-flow***® porous media model, enter an “Effective radius” for the sand. The effective

Tip: Choose an "Effective radius" that is slightly smaller than your average grain size.

radius is used to control the pressure drop relation for gas flow through the sand. Since sand typically has a size distribution, a single, effective radius must be determined. Experience has shown that the finer sand grains influence the pressure drop more than the coarser ones. It is good practice to use a sand size that is slightly finer than the average grain size. Typically a value that is between 30% – 45% up the cumulative sand size distribution curve is a good choice.



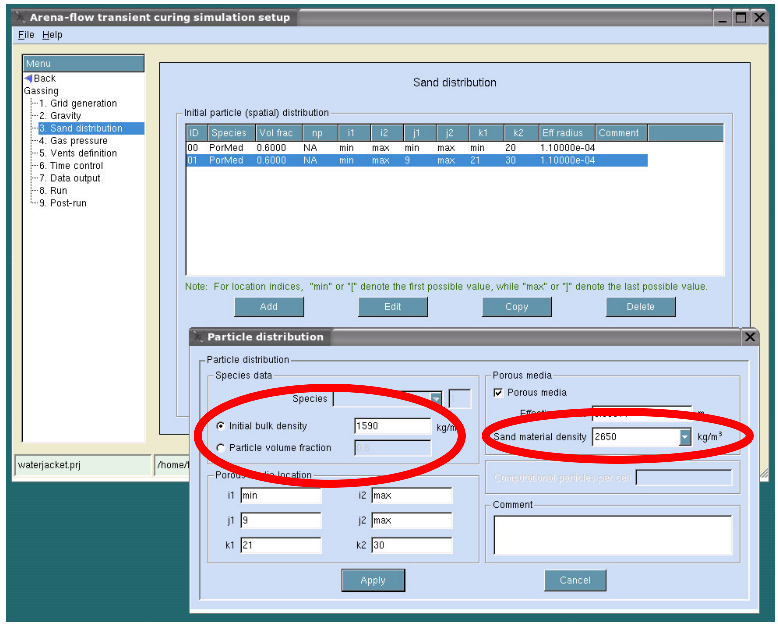
For example, in the sand size distribution shown above, 43% of the sand has a size of 89.75 µm or less. Thus a value between 80 and 90 µm would be a good choice of an "Effective radius" for this sand.

Tip: Enter comments whenever possible – it will save you time in the long run.

Lastly you are permitted to enter a "Comment" for the distribution. This is not used by the solver, but can be extremely useful when setting up large projects.

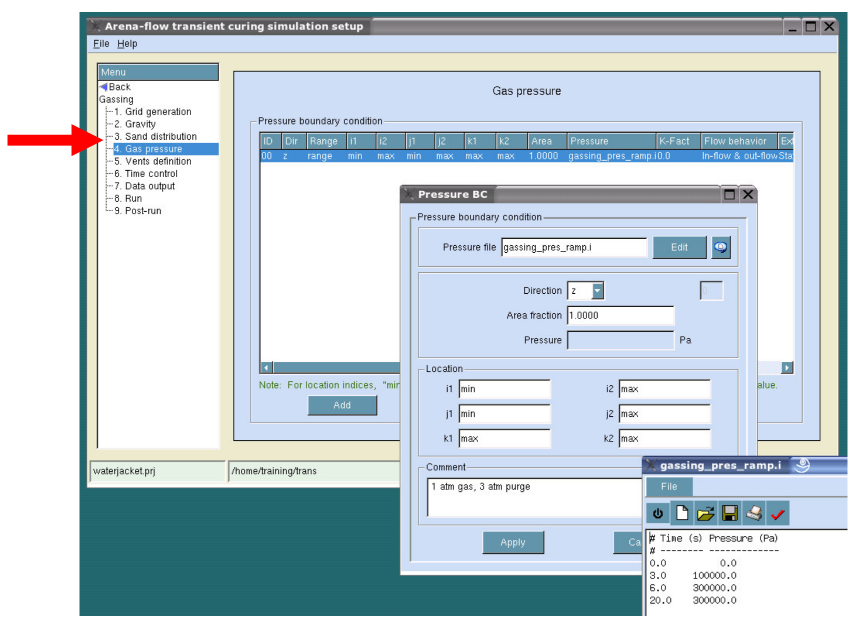
Note: If an "Initial bulk density" is entered, a "Sand material density" is also required to fully specify the problem.

An "Initial bulk density" may be defined instead of a "Particle volume fraction". The "Initial bulk density" is the average density of the core. If an "Initial bulk density" is used, a "Sand material density" must also be entered to fully specify the problem. Choose a predefined sand from the "Sand material density" drop-down list, or type the density into the field.



Once a sand distribution is defined using the "Add" button, it appears in the "Initial particle (spatial) distribution" table on the "Sand distribution" page. These distributions can then be edited using the "Edit" button, reproduced using the "Copy" button or deleted using the "Delete" button.

After the "Sand distribution" is set, click on "Gas pressure" to go to the "Gas pressure" page. Here you tell ***Arena-flow***® what pressure will be used and where the pressure is to be applied.



Click on “Add” to raise the “Pressure BC” window.

Typically the gassing cycle involves a time-varying pressure profile. Transient pressures are entered through a “Pressure File”. Click on “Edit” in the “Pressure BC” window to create or edit a transient pressure file.

The “Pressure File” contains time and pressure entries in two columns, separated by “white space” (spaces, tabs, etc.). Lines beginning with the “#” character are comments and are ignored by the solver. ***Arena-flow***® will linearly interpolate between entries.

Tip: Many conversion factors to SI units can be found by clicking on “Help”, “Units Reference”.

In the example above, the pressure is ramped up from 0 to 100,000 Pa over 3.0 seconds. From 3.0 to 6.0 seconds, the pressure increases further to 300,000 Pa, and is then held constant until 20.0 seconds. To determine the appropriate conversion factor when converting from alternate pressure measurements to Pa, click on “Help”, “Units Reference”.

Important: The CFD component of ***Arena-flow***® is an incompressible solver; thus all pressures are gauge pressures with zero (0) representing the vented, atmospheric pressure.

Please note that the CFD component of ***Arena-flow***® is an incompressible solver. As such, all pressure measurements are relative. Calculations are typically set up with a zero (0) pressure at the vents. Thus the gas pressure should be in gauge pressure, relative to the vents.

After saving the transient pressure file, the file name appears in the “Pressure file” field. Next select the “Direction” from the drop-down menu. This tells ***Arena-flow***® upon which faces of the cells the pressure boundary condition is to be applied. If your CAD was designed with the z-axis pointing up and your gas pressure is applied at the top of the model (in the blow tubes) then the default directional value of “z” can be left unchanged.

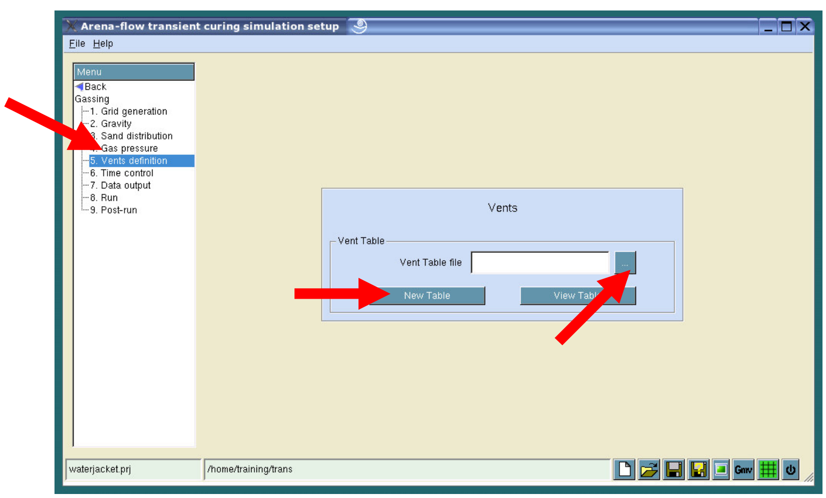
Enter the “Area fraction”. This is the fraction of the cell faces which is open to the pressure. Usually, this is left at the default value of 1.0.

If the system pressure is constant throughout the gas / purge cycle, this may be input in the “Pressure” field. The “Pressure” field is not available if a “Pressure file” is used.

Next specify the location of the gas pressure boundary condition. These are boundary cell indices and they must define a plane. You may use the reference grid utility, , to help you determine proper values for the indices. Additionally, the “min” and “max” keywords can also be used to reference the minimum and maximum cell index in each respective direction.

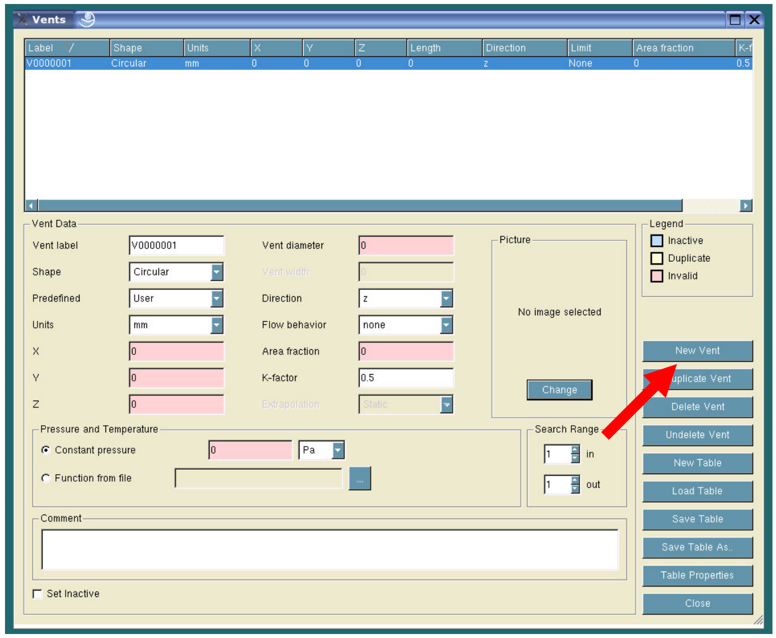
Sometimes multiple planes are required to fully specify the gas pressure inlet location. These boundary conditions can be edited using the “Edit” button, reproduced using the “Copy” button or deleted using the “Delete” button.

Finally, you are permitted to enter a “Comment” for the distribution.



Click on “Vents definition” to go to the “Vents” page. Vents are added through a vent file. To create a vent file, click on “New Table”. To link to an existing vent file, click on the browse, , button. Vent files typically are identified with the .tvt extension.

New vent tables contain no information. Click on “New Vent” to begin populating the vent table. Much of the information is defaulted for you



Tip: Enter a "Vent label" to easily identify individual vents.

Enter a "Vent label" if desired. The vent label is a comment that helps to uniquely identify individual vents. This greatly simplifies later design permutations.

Note: The "Units" drop-down list affects both the location and "Vent diameter"

Enter the location of the vent center ("X", "Y", "Z"). Note that you may use non-SI units to do so, as indicated by the "Units" drop-down list.

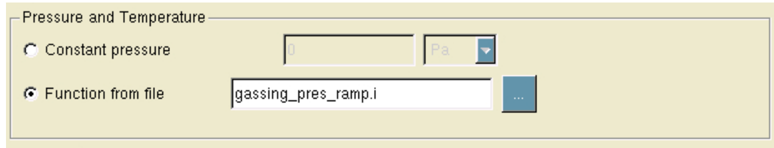
Enter a "Vent diameter". The "Vent diameter" must also be entered with the same "Units" as the location.

Enter a "Direction". This is the direction normal to the vent face. Typically cope and drag vents are in the z-direction if the model is oriented with the z-axis pointing up.

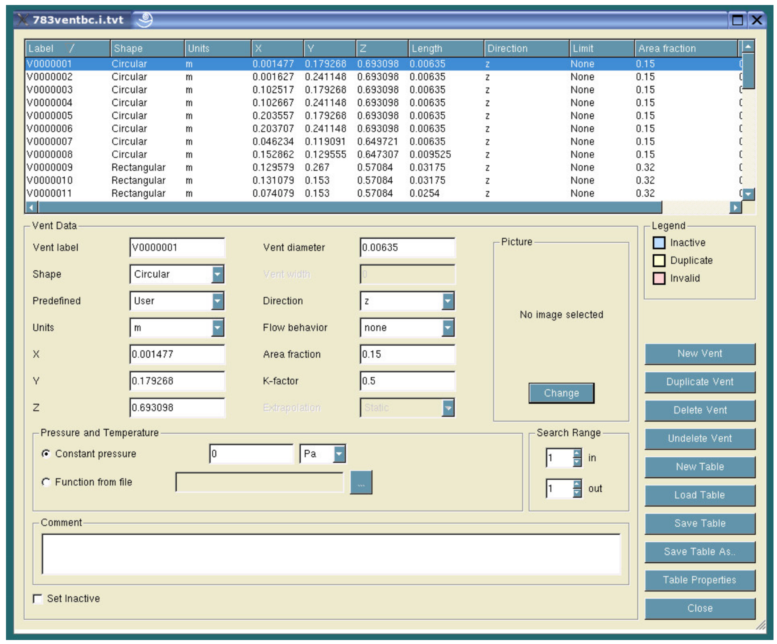
Enter an "Area fraction". This is the fraction of the vent area that is open for the air flow. Typically vent open area fractions are between 0.1 and 0.35.

Note: Often the transient pressure is applied through cope vents. This is easily modeled with ***Arena-flow***®. Simply select a "Function from file" and link to your transient gas pressure file.

Often the curing pressure is applied through cope-side venting, as well as at the gassing head. This can easily be modeled with ***Arena-flow***®. Simply, set the "Pressure" of the vent to be read from a "Function from file". Use the same transient pressure file you created on the "Gas pressure" page.



Usually all other values remain unchanged. To set a vent inactive, click on “Set Inactive”. This is a useful feature for design permutations.

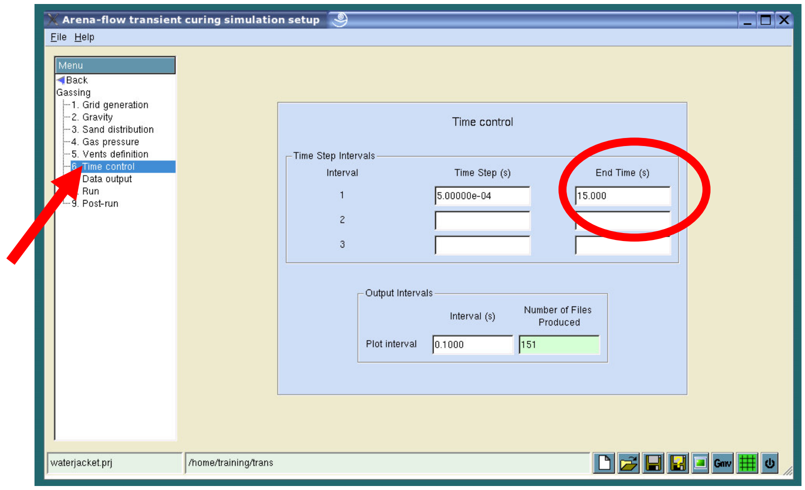


An example of a populated vent table is given above. To change any parameters of an existing vent, click on the vent to load the parameters into the display region. Remember to save your vent table before exiting.

Note: Users only need to change the “End Time” to ensure it corresponds to their blow time. All other parameters on the “Time control” page can remain unchanged.

Click on “Time control” to go to the “Time control” page. Users are only required to change the “End Time”. This is the curing cycle time in seconds. Other parameters can be left unchanged.

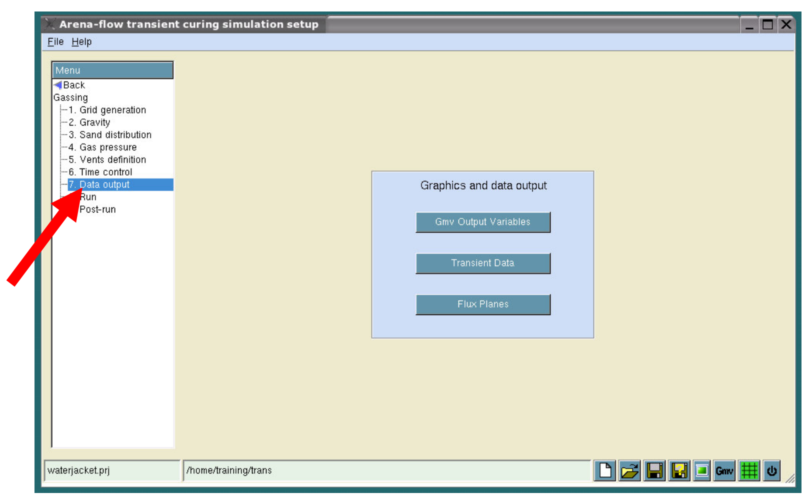
The “Time Step” is controlled by the solver, and should be left at the default value. Only change the “Time Step” if advised to do so by an ***Arena-flow***® support engineer. The solver will adjust the time step for accuracy and stability purposes, using a robust algorithm.

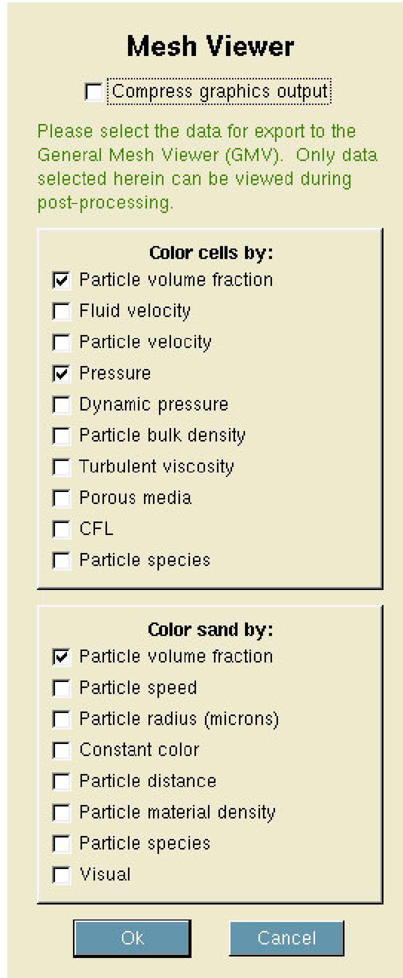


The “Plot interval” is the frequency of GMV output file production. Lower plot intervals result in more GMV output files. Each GMV file takes time to write and uses a non-trivial amount of disk space. Typically about 100 – 200 GMV files are sufficient to create smooth animations of the transient ***Arena-flow***® results.

Important: If you do not tell ***Arena-flow***® to write variables to the GMV output files, the information will not be there!

Click on “Data output” to go to the “Graphics and data output” page. For efficient post-processing, you need to tell ***Arena-flow***® what information you wish to post-process, before starting your calculation. This is typically done through the “Mesh Viewer” window. Think about your anticipated results before running your calculation - if you do not ask for the data to be written to the GMV files, it will likely not be there!





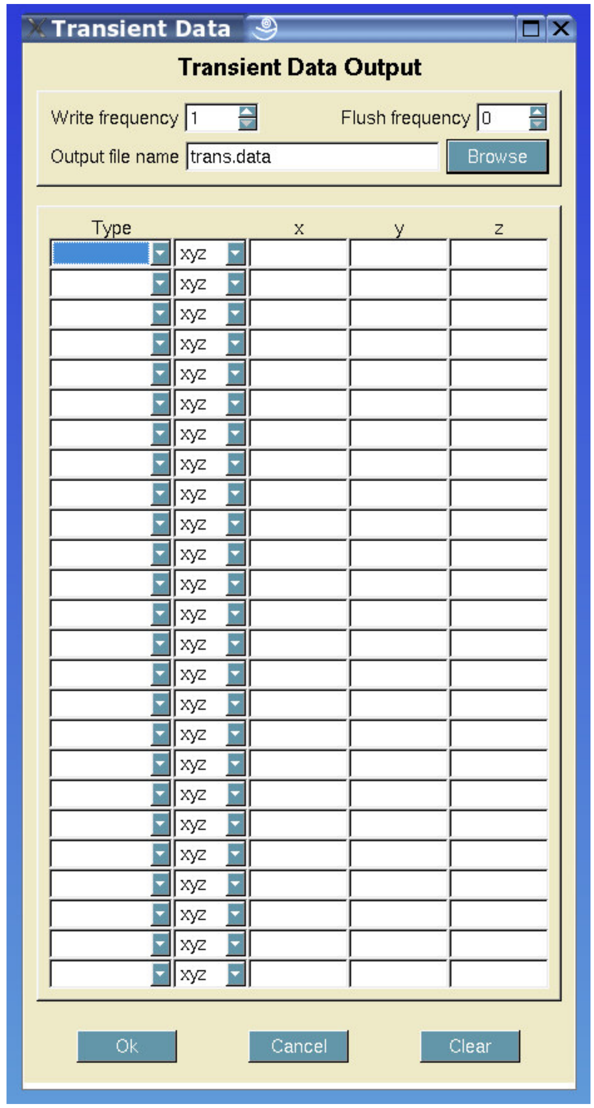
Note: Core curing is always output to the GMV files for ***Arena-flow***® transient curing calculations.

Click on "GMV Output Variables" to raise the "Mesh Viewer" window. Here you select what quantities you will wish to view during post-processing. Select the quantities by which you may wish to color your cells in the top of the window. Select the quantities by which you may wish to color your particles in the bottom of the window.

Notice there is no option to view the curing front. For ***Arena-flow***® transient curing calculations this is always output to the GMV files.

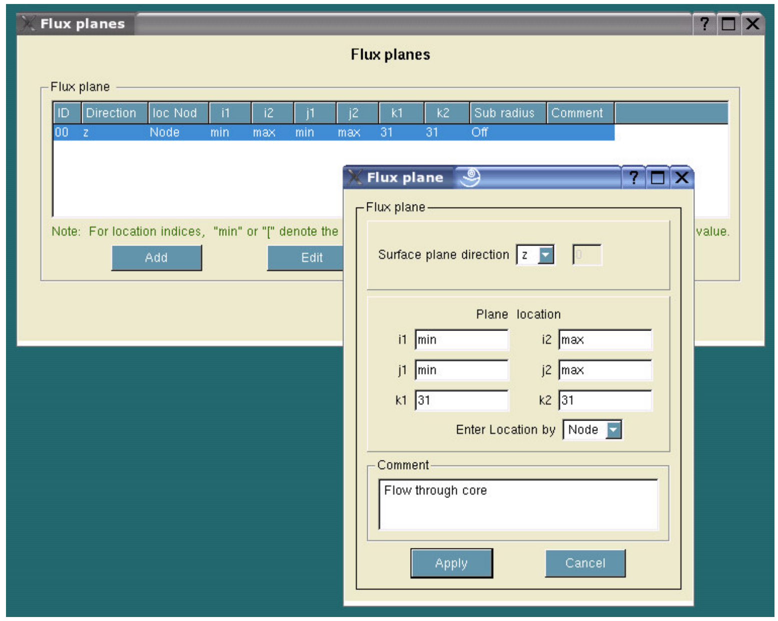
The specification of transient data is not required. The "Transient Data Output" window allows you to request that information from pre-identified cells be written to a file during the calculation.

Note: Transient data specification is not required.



To launch the “Transient Data Output” window, click on “Transient Data” on the “Graphics and data output” page. Specify an “Output file name”; the default is trans.data. Transient data output locations are specified by selecting the data type on the left, and identifying the location at the right. The location can be either spatial coordinates (x, y, z) or cell indices (i, j, k). Tell ***Arena-flow***® which you are using via the xyz / node drop down menu.

Arena-flow® will write the desired data to the output file in columnar form during the run. The output file is in a plain text format, and may be graphed via any plotting utility supporting columnar input. The “Transient Data Output” feature is not typically used for transient curing calculations.



Flux planes track all fluid and particles which cross them during the calculation.

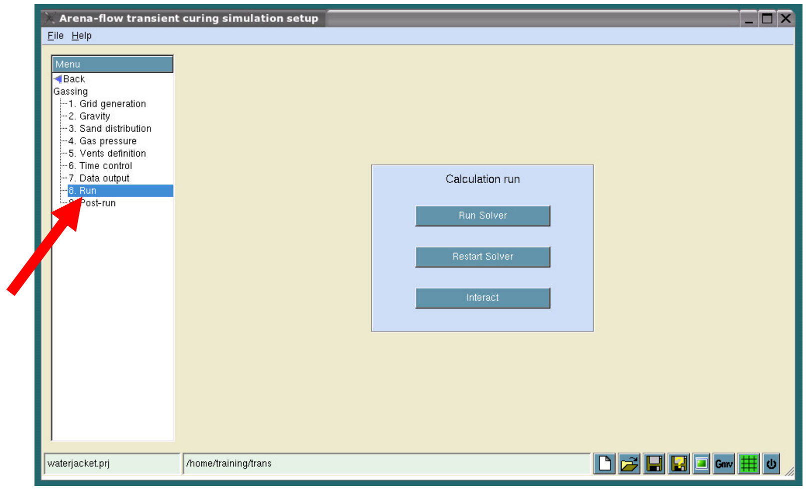
To launch the "Flux planes" window, click on "Flux Planes" on the "Graphics and data output" page. Individual planes can be defined by clicking "Add".

To define a flux plane, select the "Surface plane direction" and the "Plane location". The "Plane location" can be entered in spatial coordinates (x, y, z) or cell indices (i, j, k – i.e. "Node"). Tell ***Arena-flow***® which you are using via the "Enter Location by" drop down menu.

Each flux plane creates a text file containing columnar data. The files will be named Flux_## where ## is the flux plane "ID". The output file is in a plain text format, and may be graphed via any plotting utility supporting columnar input.

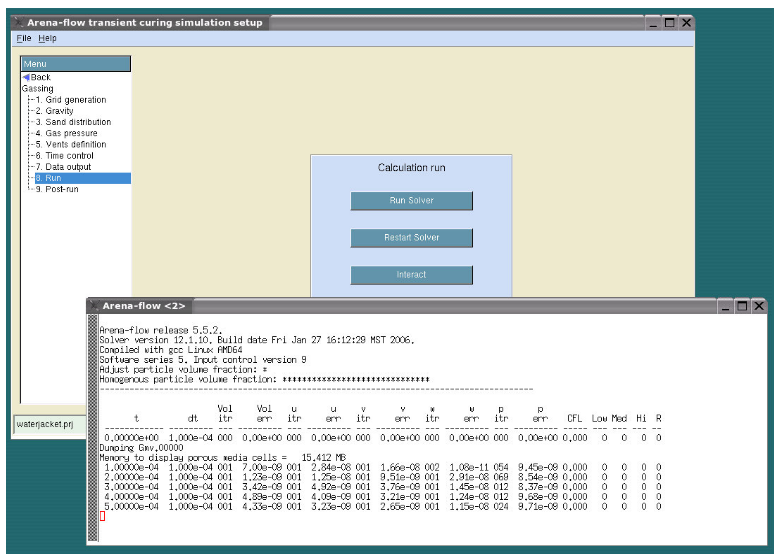
Note: A flux plane will be created automatically by ***Arena-flow***® for transient curing calculations. This is used internally by the solver.

A Flux plane is required when running an ***Arena-flow***® transient curing calculation and is used internally by the solver. For typical calculations, the location of the plane is not important. If no flux plane is defined, ***Arena-flow***® will automatically create a default plane.



Tip: You can run the ***Arena-flow***® solver by clicking "Run Solver" in the GUI, or by typing arena.x at the command line.

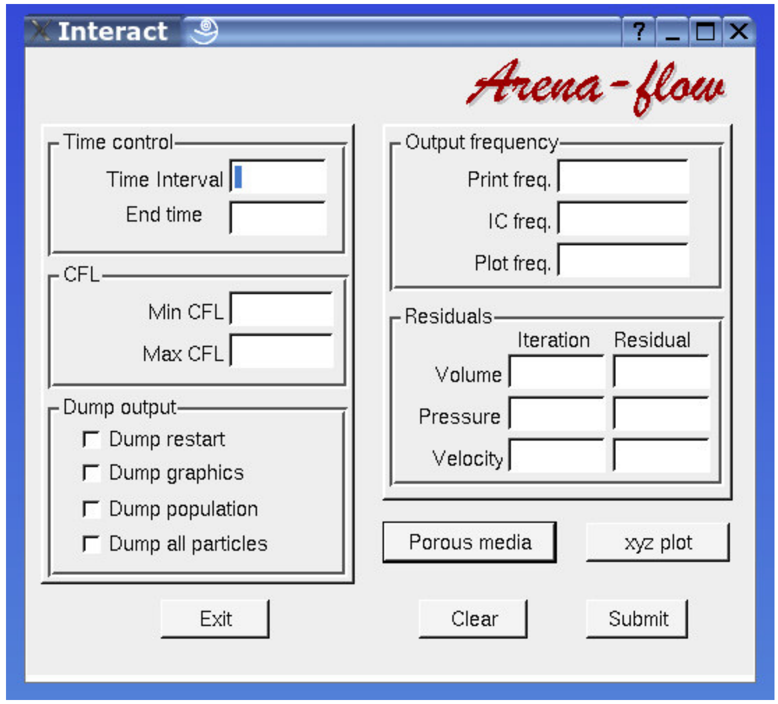
Click on "Run" to go to the "Calculation run" page. Once your calculation is fully set up, click on "Run Solver" to start the simulation. The solver may also be run from the command line with the "arena.x" command to allow for scripting. The argument is the project file name.



Important: Be sure to leave the ***Arena-flow***® solver window open during the calculation. Closing it will stop the run.

When you click on "Run Solver" in the GUI, a new window opens displaying solver information during execution. It is very important to leave this window open during the run. Closing this window will halt solver execution.

The first column of data output in the solver window shows the simulation time. More information regarding solver output is found in section 2.10, Additional Features.

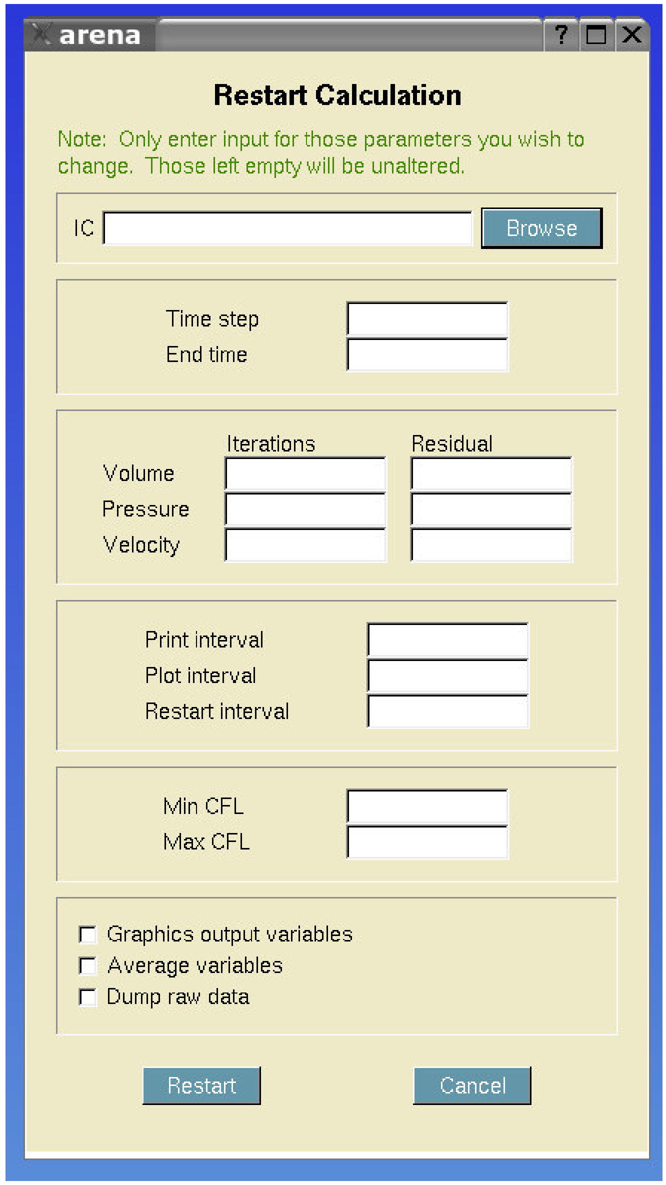


It is possible to change some solver parameters while the calculation is running. Click on "Interact" on the "Calculation run" page to raise the "Interact" window.

A good use of the "Interact" utility is to change the calculation end time during the run. For example, if your core is fully cured, change the "End time" to the current time and click "Submit". This will end the calculation. Alternately, if you are nearing your originally defined end time and your core is still not fully cured, extend the "End time" by entering a new value and click "Submit".

Tip: Only change parameters (other than the "End time") if you are advised to do so by an ***Arena-flow***® support engineer.

Only enter the information you wish to change. Use this utility cautiously, and only change other parameters if advised to do so by an ***Arena-flow***® support engineer. This utility is under redevelopment.

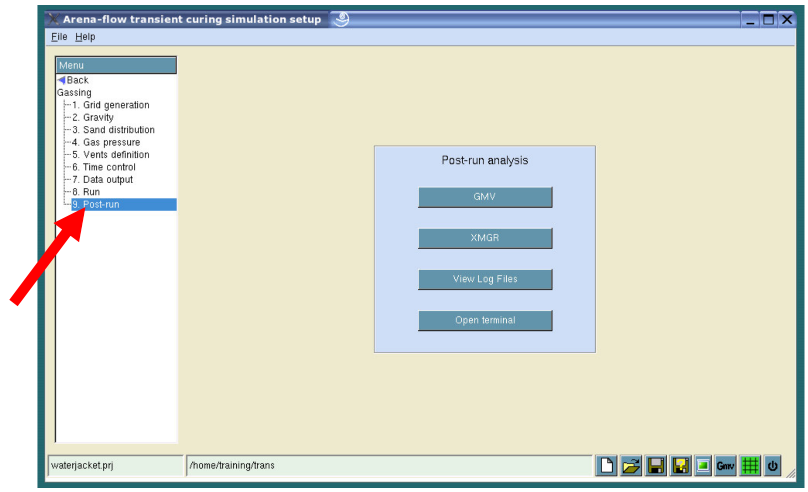


Note: It is possible to restart the solver after completion.

It is also possible to restart your calculation once it has completed. To do so, click on "Restart Solver" on the "Calculation run" page to raise the "Restart Calculation" window.

Tip: Only change parameters (other than the "End time") if you are advised to do so by an ***Arena-flow***® support engineer.

You must enter a restart file (IC). Restart file names start with "IC_" and may contain additional characters. Click on "Browse" to select the file. Only enter the information you wish to change from the original run. If you would like to run the calculation for longer than originally intended, please enter a new "End time". As with the "Interact" utility, please only change other parameters if advised to do so by an ***Arena-flow***® support engineer.



Tip: All of the utilities on the "Post-run analysis" page can be used while the calculation is running.

Click on "Post-run" to open the "Post-run analysis" page. All these features can be used while the calculation is running. The utilities are as follows:

Tip: GMV can be used to view your ***Arena-flow***® results, while the calculation is running!

- "GMV" is the general mesh viewer. This is used to post-process the graphical output from ***Arena-flow***®. The use of GMV is further described in section 2.9, Post-Processing. Using GMV, you can view your results while your calculation is running.

Note: You can use any plotting utility to graph columnar data. Use of XMGR is not required.

- "XMGR" is a plotting utility. It can be used to plot columnar data such as flux plane and transient data output files. XMGR is an open-source program. More information regarding the use of XMGR can be found in a web search. It is not required to use XMGR, please use your favorite plotting utility to graph columnar data.

- "View Log Files" launches a viewer to open text files. This can be used to view various output files.

Tip: Use the terminal window to enter additional Linux commands.

- "Open terminal" launches a Linux command prompt in the working directory to allow for the input of various commands.

Important: Check these things every time you start an ***Arena-flow***® transient gas curing calculation.

As soon as you start a transient gas curing calculation running, check the following:

- Check that your boundary conditions are applied in the correct location
- Check that your vents are applied correctly
- Check that your flux planes are defined as desired (if used)
- Check that your sand is in the correct location, namely in the core and not in the gassing head

Important: Check these things every time you analyze the results from an ***Arena-flow***® transient gas curing calculation.

Once your calculation is complete, be sure to view:

- Animation of core curing
- Last-to-cure region

Note: Optimizing the progression of the curing front through the core usually also optimizes the curing cycle time and amine usage!

It should be noted that transient curing calculations do not compute the complex chemistry involved in the cure of the core, but rather the progression of the curing front through the core. The shape and progression of the front is expected to be very accurate, however the timing and extent of cure is dependent upon many variables which are not modeled (amine type, binder type, binder concentration, temperature, humidity, etc.). By optimizing the progression of the curing front, cores are usually cured in the minimal time with the minimal use of consumables.

More information regarding analyzing your results is presented in the various ***Arena-flow***® training courses. Please consult your training manual. Contact your sales representative or support engineer for more information regarding course content and schedule.

2.8 Expert Interface

In this chapter you will learn about:

- *The purpose of the expert interface*
- *Basic layout of the expert interface*
- *Proper use of the expert interface*

The previous four chapters have described the process-oriented ***Arena-flow***® interfaces. These are used to model:

- Binder-coated sand core blowing
- Shell sand core blowing

- Steady-state gas curing
- Transient gas curing

Note: The process-oriented interfaces in **Arena-flow**® utilize default model parameters which have been extensively validated. This frees the user to simply employ **Arena-flow**® as a useful foundry tool, rather than revalidate it for each new project.

While the CPFD™ technology in ***Arena-flow***® uses many mathematical models to describe the complex particle / fluid physics of sand core blowing and curing, most of these model settings are hidden from the user. Each of the four process-oriented interfaces utilizes default settings for many models which have undergone extensive validation for numerous test cases. Users are not tasked with the burden of understanding and re-validating each model for each new project; rather the ***Arena-flow***® interfaces may simply be employed as useful foundry engineering tools.

Important: Only use the expert interface if advised to do so by a support engineer.

Arena-flow® allows users to access many of these model parameters through the expert module. Users must exercise caution when operating the expert interface, since an unphysical model setting can yield unphysical results. While it is possible to vary a model setting to improve agreement with measured data for one particular simulation by an unphysical model setting, agreement may be hindered for another project. For this reason, the expert interface should only be used upon consultation with a support engineer. Other use of the expert interface is not fully supported.

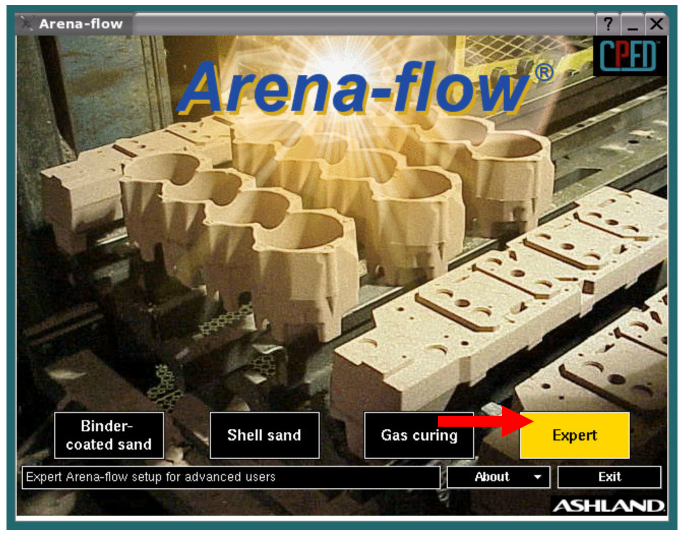
Note: A typical use of the expert interface is to change a model parameter in an existing project set up via a process-oriented interface.

The purpose of the expert interface is to allow ***Arena-flow***® users further control of a particular engineering model. It is a support tool, allowing you access to more of ***Arena-flow***®'s advanced features. Typically, projects are created and set up using one of the process-oriented interfaces. Once set up, the project file may be opened by the expert interface to vary a model parameter.

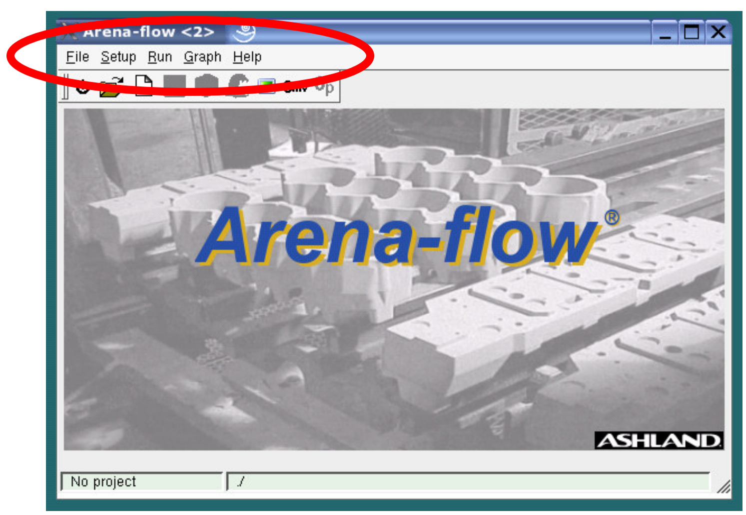
For example, consider the gravity-driven filling of a magazine with sand. Such a project may be initially created using one of the core blowing interfaces (binder-coated sand or shell sand). The core blowing interface will assume a pressurized air is used to drive the sand and automatically set the air properties for the user. Such a project may be opened using the expert interface to modify the fluid properties to reflect those of non-pressurized air.

Note: The expert interface can open any ***Arena-flow***® project file.

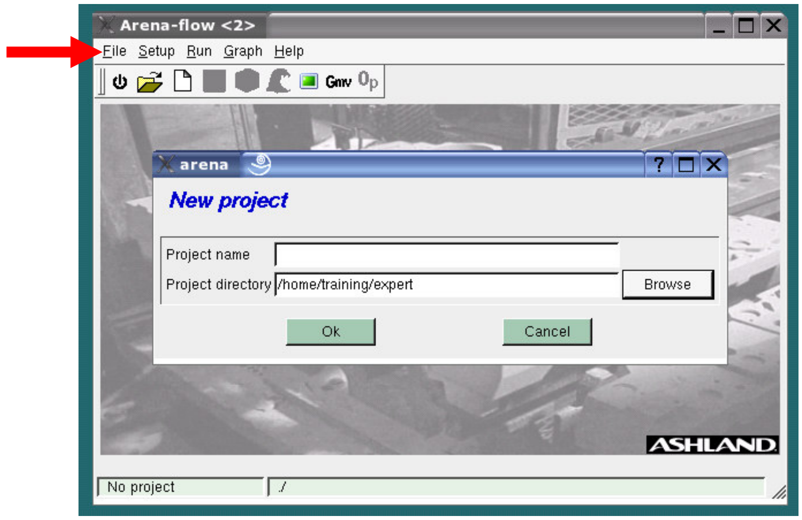
The expert interface can open any ***Arena-flow***® project file. The process-oriented modules can only open project files last modified by either that same interface, or the expert interface.



To open the ***Arena-flow***® expert interface, click on "Expert" as shown.



All controls of the expert interface can be found in the menu items across the top.



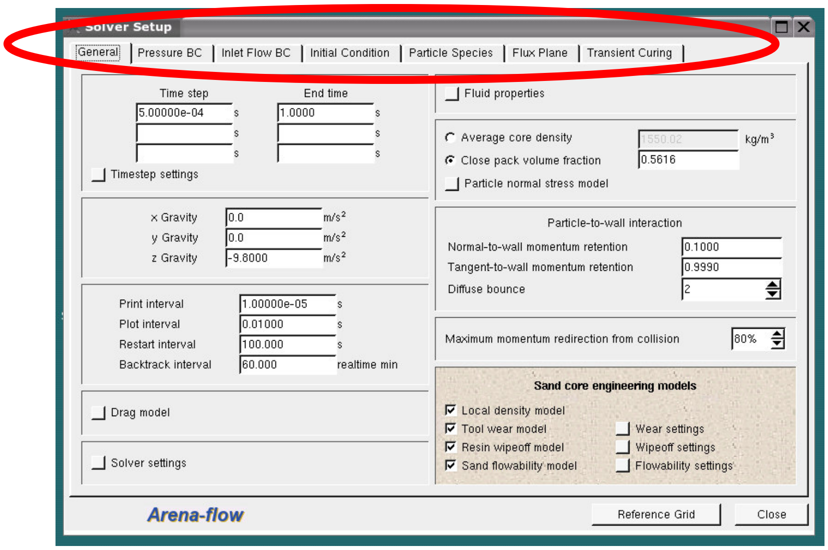
As with all other ***Arena-flow***® interfaces, all information is tied to a project file. You can either create a new project or open an existing project. Project file access is through the "File" menu.

> **Note:** The expert interface accesses the same "Grid generator setup" window as the process-oriented interfaces. The expert interface defaults the grid generator options to those required for core blowing calculations.

Once a project file is accessed many commonly performed tasks are related to either grid generation or project setup. The "Grid generator setup" window is accessed through the "Setup" menu or by clicking on the button. This opens the grid generator setup window with the same functionality as described in section 2.3 of this Users Guide. However, the expert interface does not know the intended use of the project, so the grid generator options are not customized to your (unknown) intended purpose. The expert interface defaults the "Advanced grid generation options" to those appropriate for core blowing.

The "Solver setup" window is also accessed through the "Setup" menu or by clicking on the button. This opens the "Solver Setup" window as shown.

The "Solver Setup" window consists of several pages access by the tabs across the top. You may move from page to page by clicking on these tabs.



May of the controls found in the process-oriented interfaces are found in the expert interface as well. Typically the expert interface allows users more control over model parameters than do the process-oriented modules.

Before using the expert interface please contact a support engineer. The engineer will evaluate the intended use against the known capabilities of ***Arena-flow***® and instruct you how to best proceed. Typical uses of the expert interface may be to:

- model different fluid properties than the default (pressurized air)
- model a project where the gravitational vector does not directly line up with an axis of the CAD model
- vary the wall bounce of the sand to customize it for your particular particles

These are only examples; your support engineer can further advise you of the expert interface capabilities.

2.9 Post-Processing

In this chapter you will learn about:

- *GMV – the **Arena-flow**® post-processor*
- *How to use GMV to check your grid*
- *How to use GMV to check your project setup*

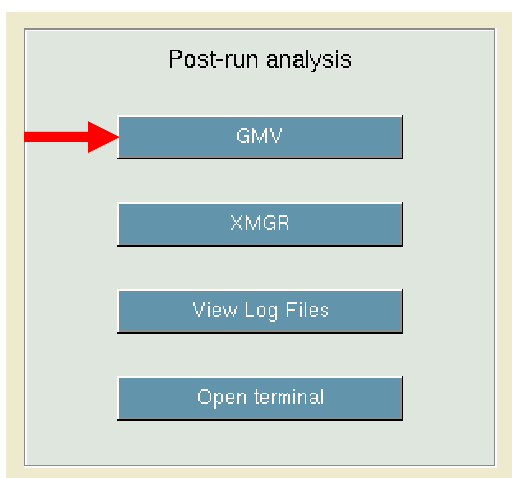
- *How to use GMV to post-process a core blowing calculation*
- *How to use GMV to post-process a steady-state gas flow calculation*
- *How to use GMV to post-process a transient gas curing calculation*
- *Where to find additional information regarding GMV and its full capabilities*

All ***Arena-flow***® projects process enormous amounts of air and sand data as the calculations run. This information is output periodically to files which are snapshots of the data of interest at an instant in time. The filenames begin with "Gmv." and are written to the working directory.

The utility used to post-process these files is the General Mesh View (GMV). GMV is an easy to use, 3-D scientific visualization tool designed to view simulation data from any type of structured or unstructured mesh[20]. The GMV executable is included on the ***Arena-flow***® installation disk. Full documentation and new versions are available on the GMV web site:

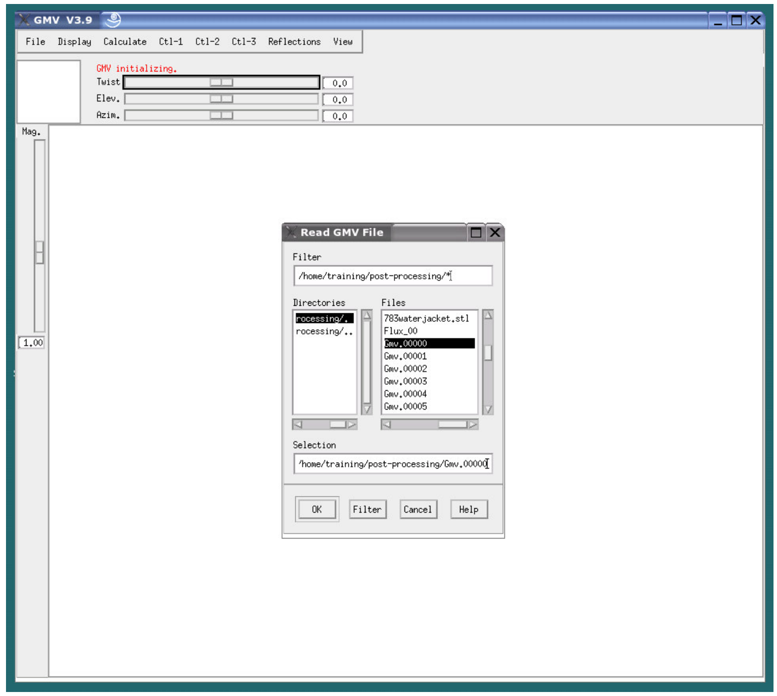
http://www.x-div.lanl.gov/XCM/gmv/GMVHome.html

The use of GMV with ***Arena-flow***® is covered extensively the various ***Arena-flow***® training courses. Please consult your training manual. Contact your sales representative or support engineer for more information regarding course content and schedule.



Note: GMV may be launched from the “post-processing” menu, by clicking on the Gmv button or by typing “gmv” at the command line.

To launch GMV, click on the Gmv button or on the “GMV” option from the post-processing menu of any process-oriented interface. GMV may also be launched from the command line with the “gmv” command; no input argument is necessary.



Note: Gmv.00000 contains the initial condition information. Subsequent GMV files are named Gmv.00001, Gmv.00002, etc.

When GMV is launched, the “Read GMV File” window is active prompting users for an input file. ***Arena-flow***® files for use with GMV begin with the prefix “Gmv”. “Gmv.00000” is always the initial conditions file at “time zero”. Subsequent GMV files are named “Gmv.00001”, “Gmv.00002”, etc.

GMV also requires several other files to view the information contained in the Gmv.##### files. These are:

- 00cells.gmv
- 00drawcells.gmv
- 00grid.grd
- 00gridstl.gmv
- 00mat.gmv
- 00nodes.gmv
- 00poly.gmv

These files are created by running the ***Arena-flow***® grid generator and may be modified when running the

Important: When storing result files, always be sure to store the 00* files as well; without these files your Gmv.##### files will be unreadable.

Arena-flow® solver. These files must be located in the run directory with the Gmv.##### files. When storing result files, always be sure to store the 00* files as well; otherwise your Gmv.##### files will be unreadable.

Common uses of GMV with ***Arena-flow***® can be subdivided into three categories: evaluating the grid, checking the project setup and post-processing the results. The remainder of this chapter discusses common GMV tasks for ***Arena-flow***® calculations.

2.9.1 Evaluating the grid

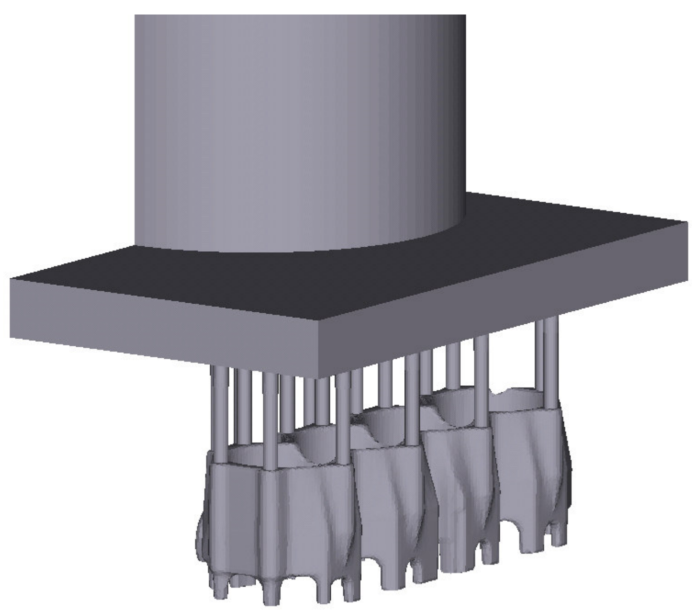
All ***Arena-flow***® projects require a grid. Grid generation is described in section 2.3 of this User Guide. Whenever generating a grid, be sure to perform the following tasks:

- View the CAD
- View the grid with grid lines
- View the grid as a transparent model
- Compare the grid to the original CAD

Viewing the CAD

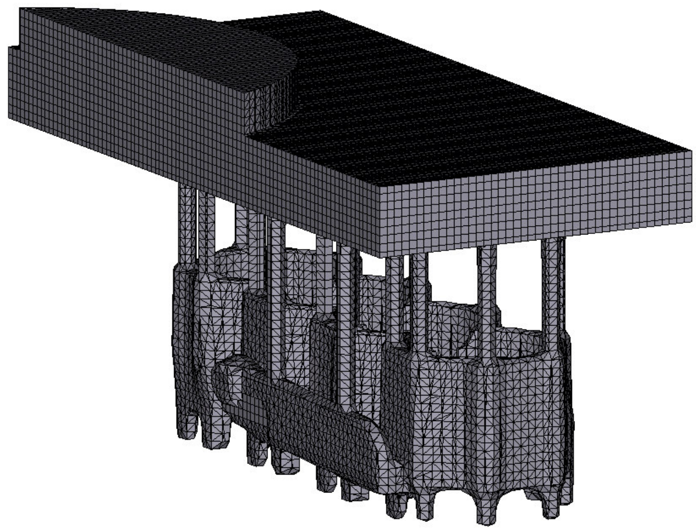
Viewing the original CAD (STL) file will help you understand the geometry to which your grid should conform. Use sound judgment – consider what level of resolution is required for the process you are modeling.

Tip: Viewing the CAD will help you understand the geometry you are modeling.



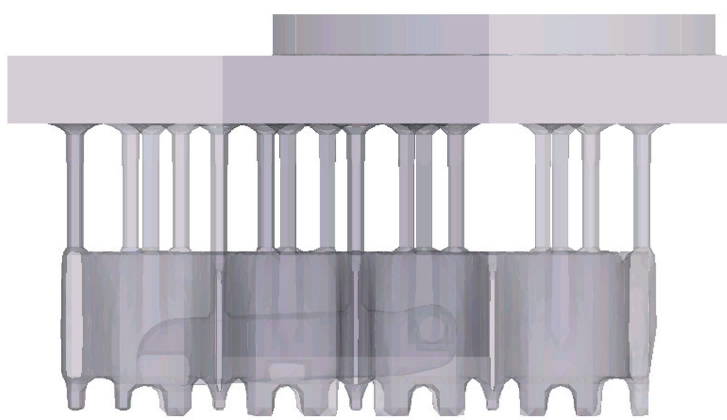
Viewing the grid with grid lines
Viewing the grid with grid lines will help you understand the foundation of the CPFD™ model you have created. Check for mesh uniformity, cell aspect ratio, number of cells, etc. More information on proper grid generation may be found in section 2.3 of this User Guide.

Tip: Always check your as-meshed grid.



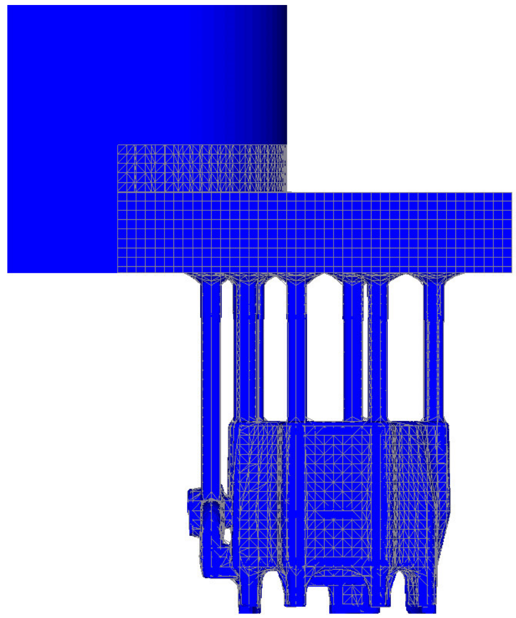
Viewing a transparent grid
Viewing a transparent grid will show you the connectivity of the meshed region. Any internal holes in the grid are visible in the transparent mode as they reflect light.

Tip: Viewing your transparent model will reveal any holes or other internal structures.



Comparing the grid to the CAD
Comparing the grid directly to the CAD will show you regions where the mesh deviates from the original input geometry.

Tip: Always compare your grid directly to the original CAD.



2.9.2 Checking the project setup

As soon as you start your ***Arena-flow***® calculation running, it is important to check your project setup. By catching setup errors early, you can correct them before needlessly spending time and computational resources on a simulation that does not reflect your actual foundry process. Whenever you run the ***Arena-flow***® solver, be sure to perform the following tasks:

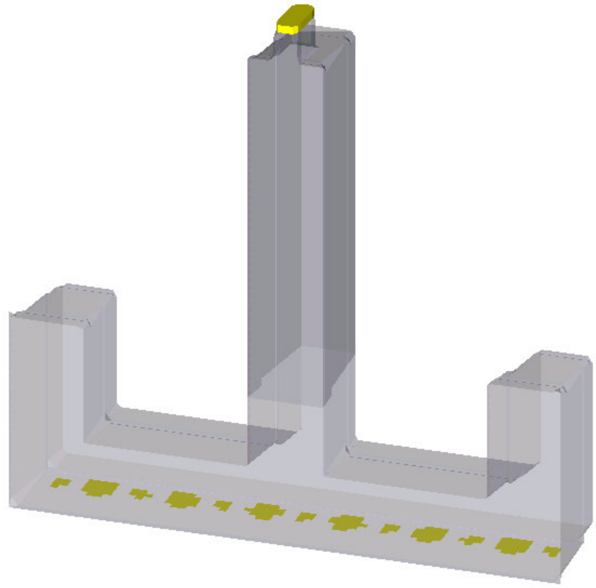
- View the boundary conditions
- View the vents
- View the flux plane locations (if used)
- Verify the initial sand location

Additionally, ensure the Gmv.00000 file contains all the information required for post-processing.

Important: When starting the solver, ensure the Gmv.00000 file contains all the information required for post-processing.

Viewing the boundary conditions
Viewing your boundary conditions will ensure they are applied to the correct part of your model. Boundary conditions could be blow pressures, gassing pressures or vents.

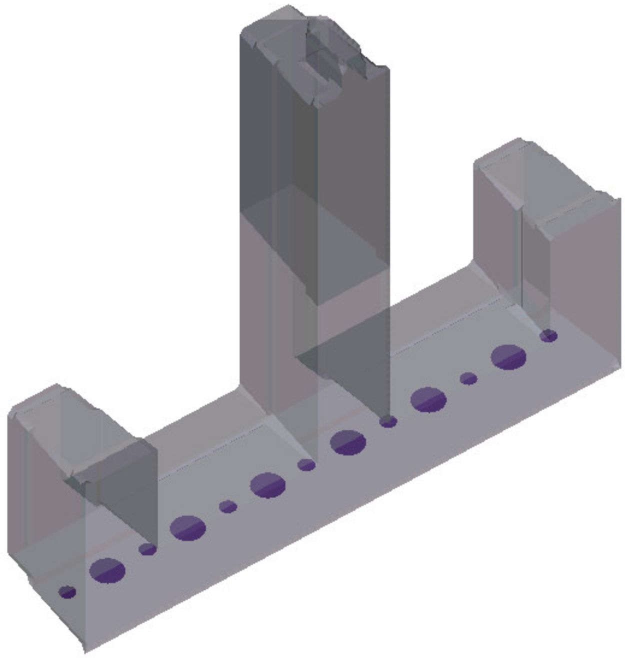
Tip: Always check the locations of the boundary conditions on your model.



Viewing the vents

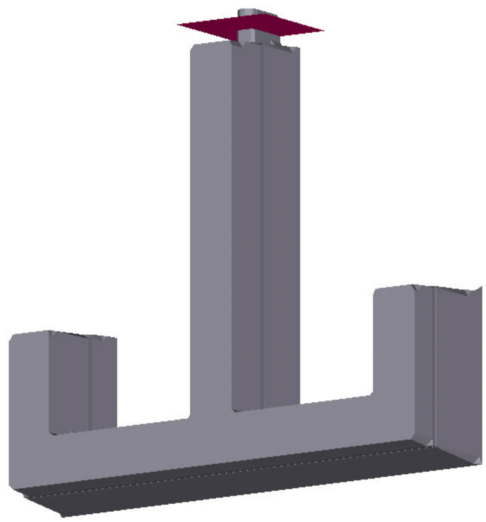
It is important to view your vents directly, even after checking your boundary conditions in general. Viewing the vents provides you with information as to how well the vents were applied to the model, and will highlight vents which were not found on the model surface.

Tip: Viewing the vents on your model will quickly show you any vents which were specified incorrectly.



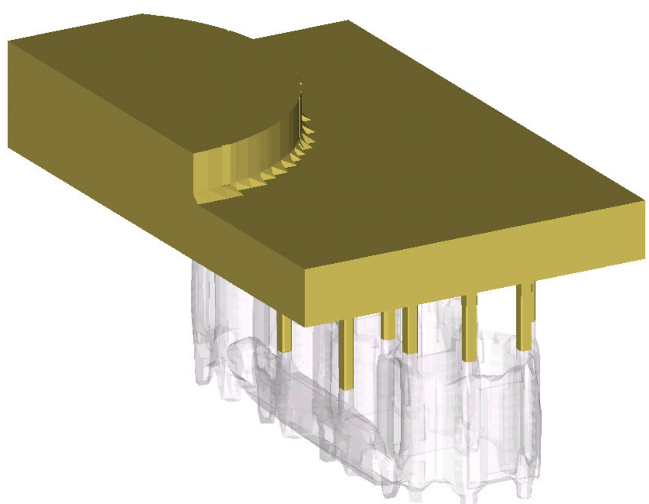
Viewing the flux planes
If you used ***Arena-flow***®'s flux plane feature, it is important to view the flux planes to ensure they are located where you intended.

Tip: If you ***Arena-flow***®'s flux plane feature, always check the flux plane location on your model.



Viewing the initial sand location
Viewing your initial sand location will ensure the sand is in the correct place at the start of your calculation. For core blowing calculations, the sand should be initialized in the blow plate and sometimes in the blow tubes. For steady-state or transient gas curing calculations, the sand should be located in the core and not in the blow tubes.

Tip: Always check the initial sand location in your model.



2.9.3 Post-processing core blowing calculations

Post-processing is one of the most important parts of any numerical simulation process. Getting an answer is easy – finding meaning in the answer and applying it practically can be hard work.

While it is not possible provide a recipe for post-processing every scenario, when analyzing ***Arena-flow***® core blowing results it is important to consider the following:

- Does the sand fill the tooling?
- Are there variations in the final density of the filled core?
- How does the sand fill the tooling?
- How balanced is the fill?
- Does the transient filling animation explain the density variations in the filled core?
- At any point in time is there an unfilled region in the core with no direct path for sand to take between a blow tube and that region?
- What regions have the potential for problems?
- What causes those regions to have the potential for problems?
- Does the colored sand show the filling more clearly?
- How is the air flowing during the core filling process?
- Where do I expect tool wear?
- If this is a shell process, where could the filling be even worse than predicted due to premature hardening of the core?
- Would I have expected these results?

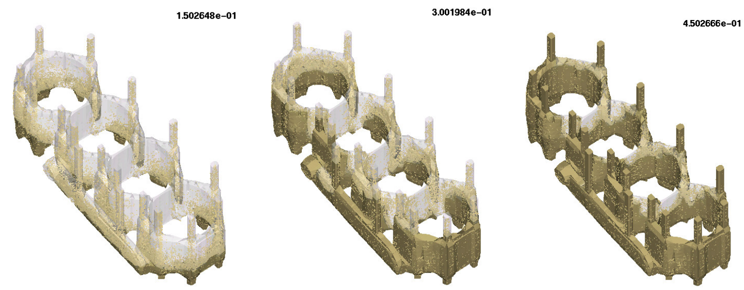
Some of these questions can be answered by performing the following tasks:

- Viewing the sand filling of the core
- Viewing the filling of the core (sand colored by volume fraction)
- Viewing the density variations in the filled core
- Viewing the sand speed as it fills sections of the core
- Viewing regions of likely tool wear
- Viewing air flow through the system
- Viewing pressures through the system

- Viewing colored sand information

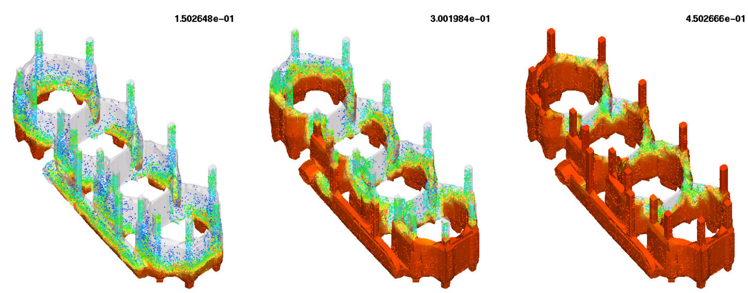
Viewing the sand filling of the core
Arena-flow® is very useful to visualize the sand filling behavior of the core. Animations of the sand motion give you a feel for the overall core filling behavior. This is how the sand filling would look, if the tooling was transparent.

Tip: Animations of sand motion help you visualize the transient sand filling pattern, as though the tooling was transparent.



Viewing the filling of the core
While the sand filling allows you to numerically "look inside your tooling", coloring the sand by the local volume fraction (density) provides you with more information about the transient filling behavior.

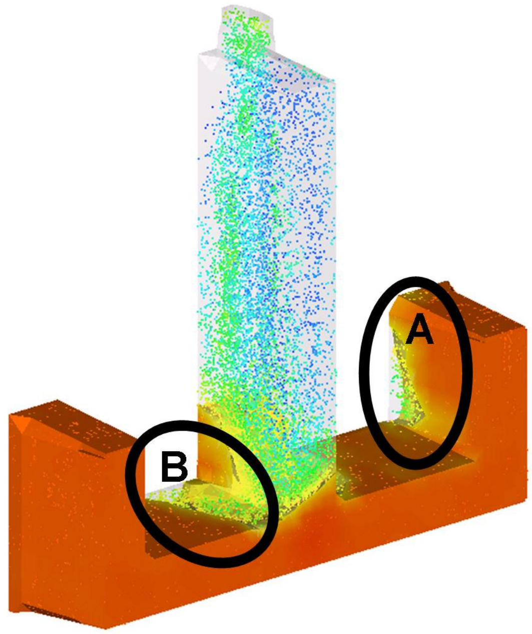
Tip: Coloring the sand by the local volume fraction or density provides you with more information about the transient filling behavior.



In particular, look for regions which are not fully compacted, yet have no direct, low-density path between the uncompacted region and a blow tube. These regions are often related to core defects.

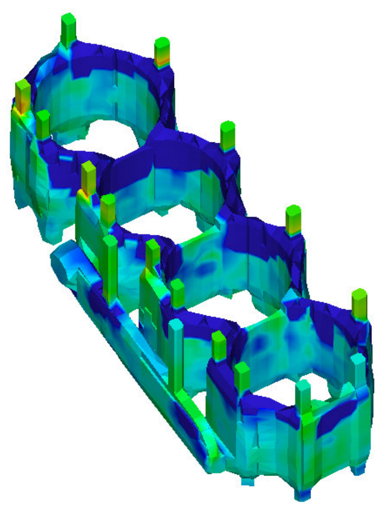
In the example below:
- Region A is not fully compacted and the sand has no direct path to take from the blow slot to reach the region. Here a defect is possible.
- Region B is also not fully compacted, but sand can still flow directly from the blow slot to the region. No blowing defect is anticipated here.

Important: Regions which are not fully compacted, yet have no direct, low-density path between the uncompacted region and the blow tubes are likely to have defects.



Viewing the density variations of the filled core
Arena-flow® not only computes core filling and non-filling behavior, but also models density variations in fully compacted cores. Check your core density variations. If there are regions with too-low or too-high density, always check the filling animations to see why these occur. Understanding your process is the first step toward improving it.

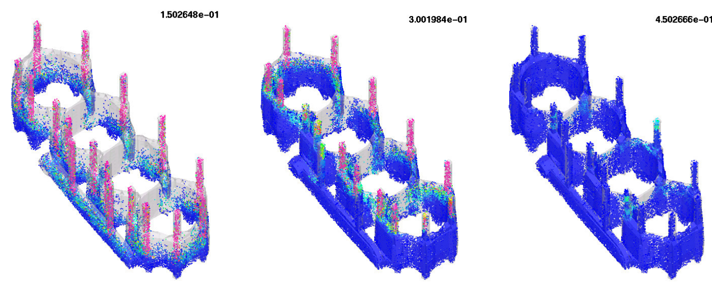
Tip: Always compare the information from the density variation plot with the information from the filling animations. By understanding your process you can make informed decisions to improve it.



Viewing the sand speed during filling

The filling animations or core density variation plots may identify regions with the potential for a problem. When studying these regions, it is often useful to also study the sand speed during the compaction process. Regions that are filled by higher-velocity sand will often obtain a higher level of compaction than will regions that are filled by lower-velocity sand.

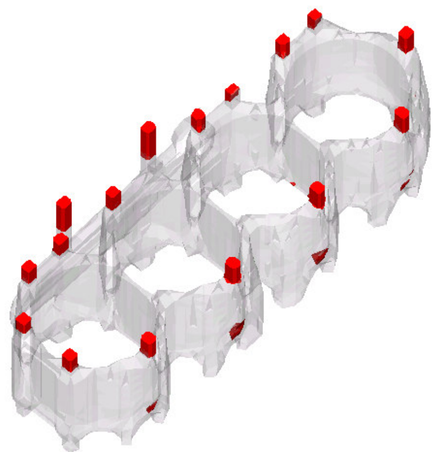
> **Tip:** Coloring the sand by speed will give you insight into the transient compaction behavior as well.



Viewing regions of likely tool wear

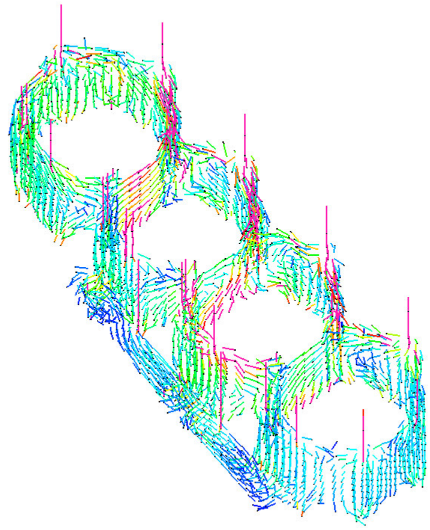
Arena-flow®'s tool wear model accounts for the interaction of every sand grain with the tooling surfaces. While the amount of wear is dependent on many parameters such as tooling material, sand properties, etc., the regions of likely wear are determined by the transient sand filling behavior. View these regions when post-processing your results. Direct comparisons between different designs are very meaningful, allowing you to maximize the life of your tooling by making informed, up-front design decisions.

> **Tip:** Viewing regions of likely tool wear helps you extend the life of your tooling with informed, up-front design decisions.



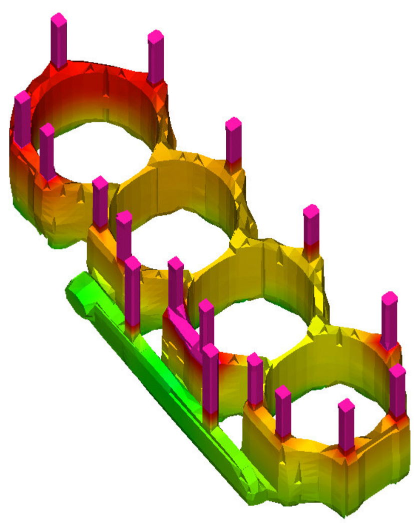
Viewing air flow
Since ***Arena-flow***® is a CPFD™ solver, both the air and sand are important. Always consider the air flow in order to understand the sand flow behavior.

Tip: Viewing the air flow will help you understand the sand motion.



Viewing air pressure
The air pressures through the system often vary by several atmospheres from the magazine through the vents. These air pressures directly affect the air flow which in turn affects the sand motion. Viewing the air pressure helps you understand the air flow through the system, enabling informed decisions regarding intelligent vent placement.

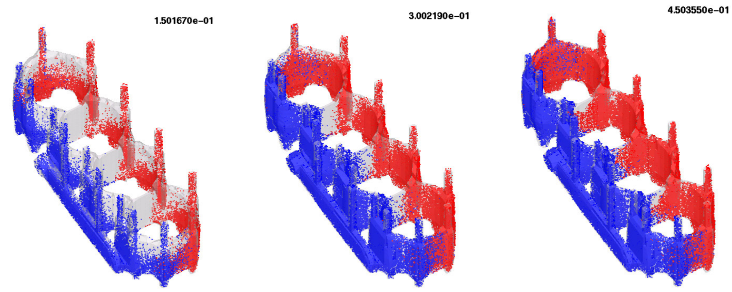
Tip: Viewing the air pressure gives you insight into the air flow, enabling you to make informed decisions regarding intelligent vent placement.



Tip: Viewing the colored sand information gives you insight into the sand flow from various blow tubes.

Viewing colored sand information

If multiple sand species were used in the project setup, then the sand can be colored by species. Viewing the colored sand information gives you quick insight into the sand flow from various blow tubes.



2.9.4 Post-processing steady-state gas curing calculations

Note: The steady-state gas curing module is used to quickly identify regions of poor air flow in filled tooling.

Arena-flow® gas curing calculations are split between two separate analyses methods: steady-state gas flow and transient curing. These modules are separate for extreme computational efficiency, but the results are related. The steady-state analysis shows the air flow at the set gas / purge pressure through the filled core. It is used to quickly identify regions of poor air flow, which may be related to regions with a non-optimal curing behavior. The transient curing analysis shows the transient progression of the curing front through the core.

When post-processing steady-state gas curing results, it is important to remember that the gas flow through the filled core is being studied. Be sure to perform the following tasks:

- View the regions of poor air flow
- View the air flow through the core
- View the pressure distribution through the core

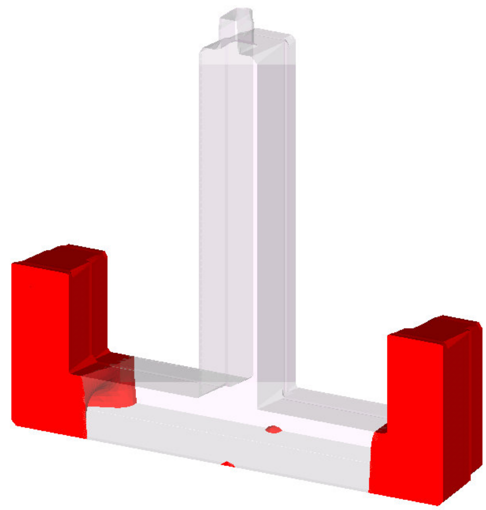
Note: The air flow moves the amine through the core during a gas curing cycle.

Viewing the regions of poor air flow

It is the air flow which moves the amine through the core during a gas curing cycle. While the steady-state gas curing calculation does not model the transport of amine directly, the regions of poor air flow typically will also be the regions of poor or late curing.

Be sure to look for sections of the core experiencing poor air flow. In general, consider regions with air flow less than 0.2 m/s or 0.5 m/s. While the velocity of the air required to produce an efficient cure varies based on geometry, amine, binder, temperature, etc., if the air flow is optimized, the curing is often optimized as well.

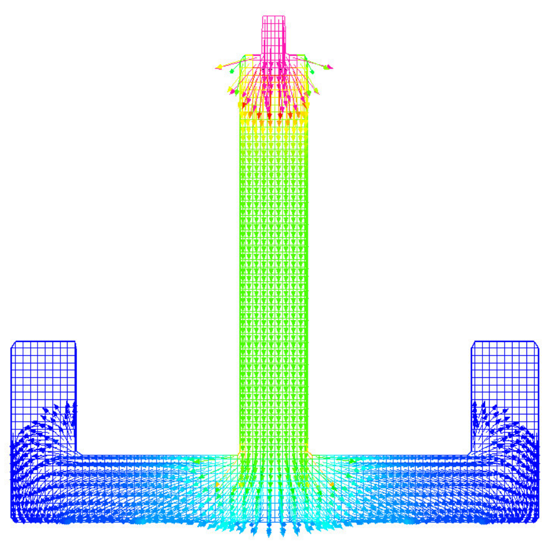
Tip: Be sure to look for sections of the core experiencing poor air flow. If the air flow is optimized, the curing is often optimized as well.



Viewing the air flow

Viewing regions of poor air flow allows you to quickly identify potential problem areas. After doing so, be sure to view the air flow itself to understand its path through the filled core, and how it influences the amine progression.

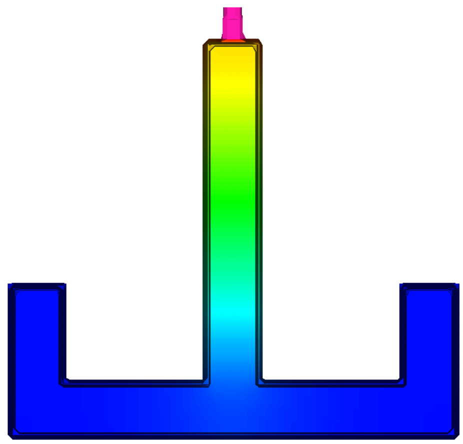
Tip: Viewing the air flow through the tooling will help you understand the amine progression.



Viewing the air pressure

While the air flow is of primary interest when analyzing ***Arena-flow***® steady-state gas curing results, viewing the air pressure give additional information regarding the air field in the core. If the air flow is good, the air pressure need not be studied extensively; however, if poor air flow is observed, then studying the air pressure can yield insight into the process.

Tip: Viewing the pressure can give you further insight into regions with poor air flow.



It should also be noted that the steady-state gas flow analyses can also be used to quickly and crudely analyze core blowing. Performing a steady-state air flow calculation on a core blowing configuration will show regions of poor air flow in the filled tooling. While this does not directly simulate a core blowing situation, if the poor air flow occurs on the cope side of the tooling, then it can lead you to question the assumptions regarding sand fill. If you run such calculations, consider whether the air flow would be sufficient to fill the core.

Note: Steady-state air flow calculations are tools which can also be used to crudely simulate the air flow during the core blowing process. These calculations are quick design iteration tools and never replace core blowing and transient curing analyses.

Steady-state gas flow calculations are often used early in the design phase to provide a quick indication of the effect of blow tubes and vents. Although these calculations can be used as fast design iteration tools, they never replace core blowing and transient curing analyses.

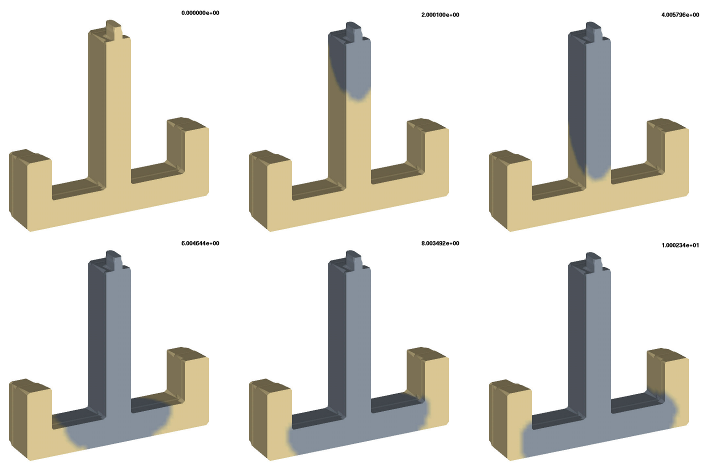
2.9.5 Post-processing transient gas curing calculations

Note: Optimizing the progression of the curing front through the core usually also optimizes the curing cycle time and amine usage!

Transient curing calculations do not compute the complex chemistry involved in the cure of the core, but rather the progression of the curing front through the core. The shape and progression of the front is expected to be very accurate, however the timing and extent of cure is dependent upon many variables which are not modeled (amine type, binder type, binder concentration, temperature, humidity, etc.). By optimizing the progression of the curing front, cores are usually cured in the minimal time with the minimal use of consumables.

Be sure to view the transient progression of the curing front through the filled core when post-processing transient gas curing calculations. In some cases it is apparent that parts of the core will not cure. However, even if your entire core cures, check the "last-to-cure" or "slow-to-cure" regions; in your foundry these regions may result in intermittent defects.

Tip: Viewing the transient progression of the curing front shows regions that will not cure, or regions that are "slow-to-cure".



2.10 Additional Features

In this chapter you will learn about:

- *Various **Arena-flow**® help menus*
- *Customizing the **Arena-flow**® GUI*
- *Compatibility between project types*
- *Solver screen output*
- *Additional output files*

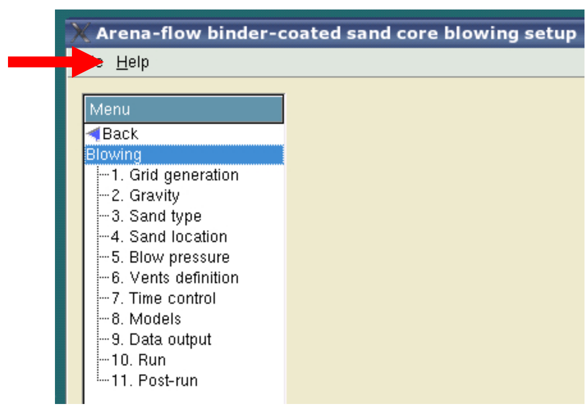
- *Plotting data*
- *Obtaining additional support*

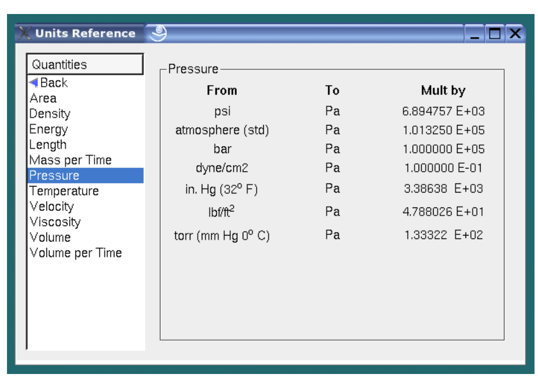
This User Guide describes the sand core engineering and modeling process, as well as details the ***Arena-flow***® software package. ***Arena-flow***® has many additional features beyond those discussed in the previous chapters. This chapter outlines several of the most useful additional features.

Help menus

Regardless of the ***Arena-flow***® interface used, various help menu items are available with more information about your ***Arena-flow***® installation. Click on "Help" to reveal the menu options:

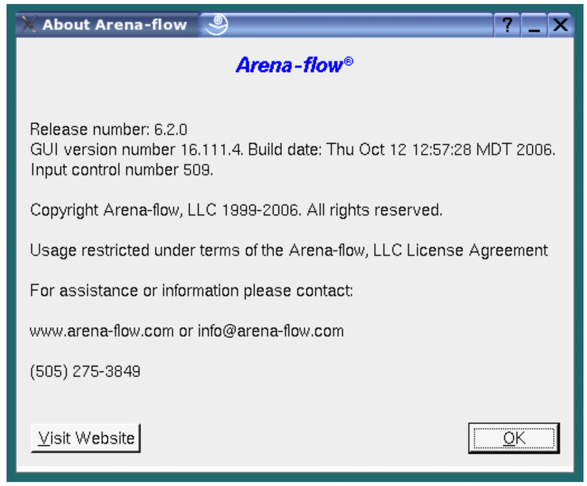
- Units Reference
- About ***Arena-flow***®
- Check License
- Read License Agreement
- About Ashland



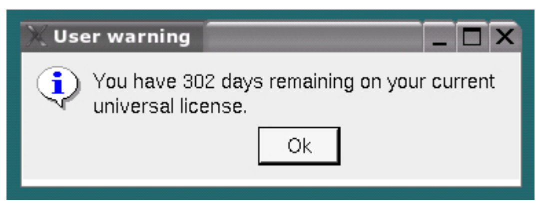


Tip: Use the "Units Reference" tool to find conversion factors to SI units.

Selecting the "Units Reference" option raises the "Units Reference" window as shown above. All input into ***Arena-flow***® is in SI units unless specified to be otherwise. The "Units Reference" tool provides you with handy conversion factors from many common units to SI units.

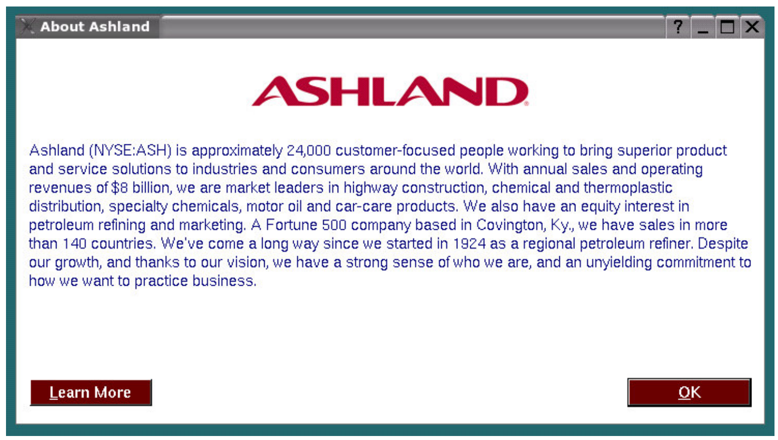


Selecting the "About ***Arena-flow***®" option raises the "About ***Arena-flow***®" window. This window contains information about your ***Arena-flow***® release and contact information for additional support.



Selecting the "Check License" option raises a window with license information. It is important to note that the license time remaining corresponds to the current license control device (software file and/or hardware activation key) on your computer, not the actual license period remaining. These may be identical or different.

Selecting the "Read License Agreement" option raises the "***Arena-flow***® license agreement" window. This is the license agreement in force which was agreed to when installing ***Arena-flow***® or activating a new license.



Selecting the "About Ashland" option raises the "About Ashland" window. ***Arena-flow***® is marketed and distributed world wide by Ashland Inc.

Customizing the *Arena-flow*® GUI

Some parts of the ***Arena-flow***® GUI are customizable. Font size is an example. When ***Arena-flow***® is launched, it is executed with a default system font. As such, ***Arena-flow***® may look different on different computers with different screen resolution settings.

> **Note:** The ***Arena-flow***® GUI can be launched with a custom font size.

To customize the GUI, ***Arena-flow***® may be launched with a font size flag. To do so, use the "-f" option and specify a desired font size when launching the GUI (arena) at the command line. Figure 20 shows the "Grid Generator Setup" window when launched with a font size of 9. In contrast, Figure 21 shows the same window when launched with a font size of 14.

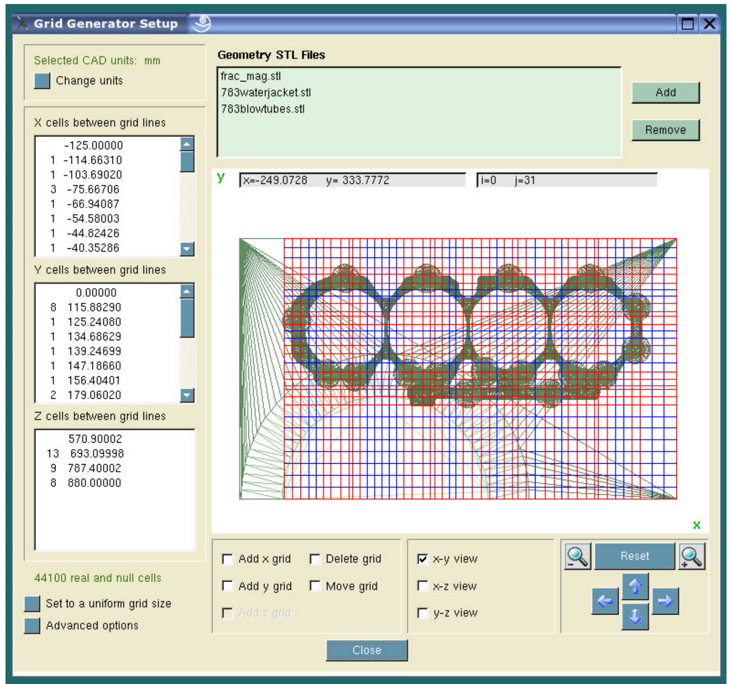


Figure 20: *Arena-flow*® "Grid Generator Setup" window launched by "arena –f9" command

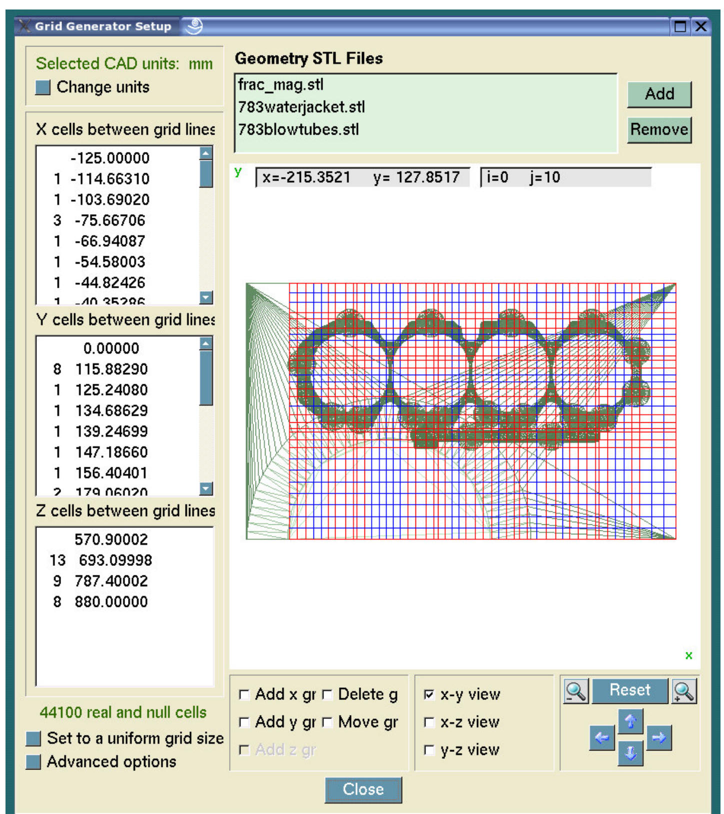
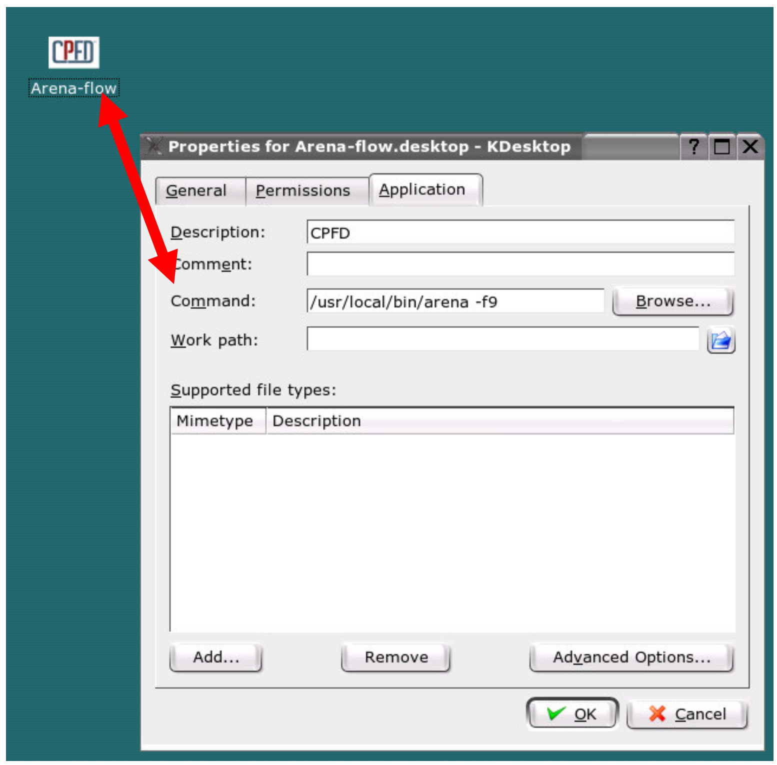


Figure 21: *Arena-flow*® "Grid Generator Setup" window launched by the "arena –f14 command

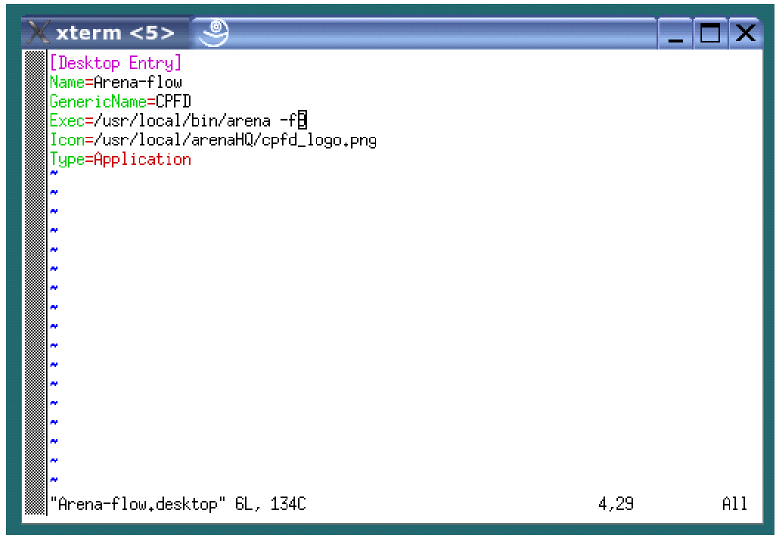
Notice that some of the text in Figure 21 does not fit in the GUI window. If this occurs, be sure to customize your ***Arena-flow***® GUI with the –f option.

Once you find a preferred font size, you may modify the ***Arena-flow***® icon to always use the preferred font size. To do so you must change the command executed when the icon is clicked. The method of changing this command varies from system to system. On most newer SUSE systems, you can right-click on the ***Arena-flow***® icon, select "Properties" and change the "Command" under the "Application" tab as shown below.

Tip: To customize the default font size, edit the command issued by the ***Arena-flow***® desktop icon.



If you do not have permissions to change the icon properties, you can either execute the command as the "root" user, or edit the icon file directly. This file is called "Arena-flow.desktop" located in the user's "Desktop" directory. To edit this file, change the "Exec" line as shown below to include the desired default font size.



Compatibility between project types

A project saved by any of ***Arena-flow***®'s process-oriented modules (binder-coated sand core blowing, shell sand core blowing, steady-state gas curing or transient gas curing) cannot be opened by another process-oriented module. Each project type is created with all models defaulted for use with that particular type of calculation.

Note: A project saved by a process-oriented interface cannot be opened by another process-oriented interface.

However not all project parameters must be recreated when simulating several processes with the same geometry. Many input files such as vent tables and sand size distributions can be reused.

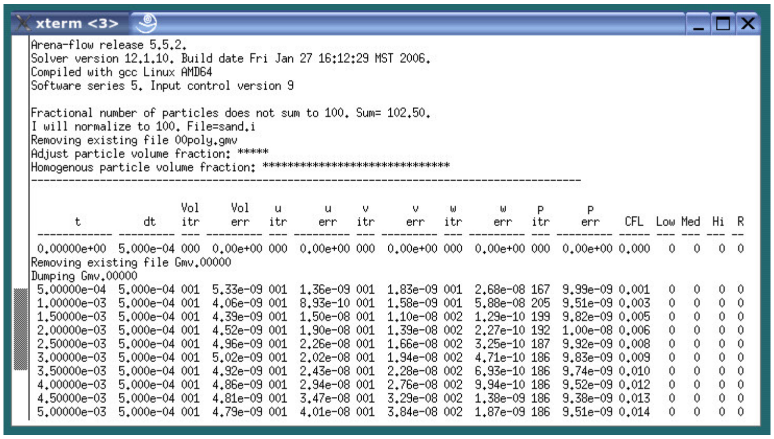
Note: You can reuse some input files (such as sand size distributions and vent tables) when simulating different processes.

For example, if a steady-state gas curing simulation was completed and a transient gas curing simulation was desired, a new project file must be created using the transient gas curing module. However, the vent table originally created for the steady-state calculation can be reused in the transient gas curing interface. If a transient pressure is to be applied to some vents, only those vents need to be edited – the vent table need not be recreated.

If the vent table is created using GMV's "Query Data" feature, it is good practice to create the table using a fine grid. Thus, if both blowing and curing calculations are intended, it is advisable to create the vent table with one of the curing interfaces, since finer meshes are typically used for curing projects.

Tip: If using GMV to determine vent locations, it is advisable to create the vent table using a fine grid.

Solver screen output
As an ***Arena-flow***® calculation runs, some information is output to the screen. An example is shown below:



```
xterm <3>
Arena-flow release 5.5.2.
Solver version 12.1.10. Build date Fri Jan 27 16:12:29 MST 2006.
Compiled with gcc Linux AMD64
Software series 5. Input control version 9

Fractional number of particles does not sum to 100. Sum= 102.50.
I will normalize to 100. File=sand.i
Removing existing file 00poly.gmv
Adjust particle volume fraction: *****
Homogenous particle volume fraction: ******************************
--------------------------------------------------------------------------------------

                        Vol    Vol     u      u       v      v       w      w       p      p
     t           dt     itr    err     itr    err     itr    err     itr    err     itr    err     CFL  Low Med  Hi  R
 ------------ --------- --- --------- --- --------- --- --------- --- --------- --- --------- ----- --- --- --- --
 0.00000e+00  5.000e-04 000  0.00e+00 000  0.00e+00 000  0.00e+00 000  0.00e+00 000  0.00e+00 0.000   0   0   0  0
Removing existing file Gmv.00000
Dumping Gmv.00000
 5.00000e-04  5.000e-04 001  5.33e-09 001  1.36e-09 001  1.83e-09 001  2.68e-08 167  9.99e-09 0.001   0   0   0  0
 1.00000e-03  5.000e-04 001  4.06e-09 001  8.93e-10 001  1.58e-09 001  5.88e-08 205  9.51e-09 0.003   0   0   0  0
 1.50000e-03  5.000e-04 001  4.39e-09 001  1.50e-08 001  1.10e-08 002  1.29e-10 199  9.82e-09 0.005   0   0   0  0
 2.00000e-03  5.000e-04 001  4.52e-09 001  1.90e-08 001  1.39e-08 002  2.27e-10 192  1.00e-08 0.006   0   0   0  0
 2.50000e-03  5.000e-04 001  4.96e-09 001  2.26e-08 001  1.66e-08 002  3.25e-10 187  9.92e-09 0.008   0   0   0  0
 3.00000e-03  5.000e-04 001  5.02e-09 001  2.02e-08 001  1.94e-08 002  4.71e-10 186  9.83e-09 0.009   0   0   0  0
 3.50000e-03  5.000e-04 001  4.92e-09 001  2.43e-08 001  2.28e-08 002  6.93e-10 186  9.74e-09 0.010   0   0   0  0
 4.00000e-03  5.000e-04 001  4.86e-09 001  2.94e-08 001  2.76e-08 002  9.94e-10 186  9.52e-09 0.012   0   0   0  0
 4.50000e-03  5.000e-04 001  4.81e-09 001  3.47e-08 001  3.29e-08 002  1.38e-09 186  9.38e-09 0.013   0   0   0  0
 5.00000e-03  5.000e-04 001  4.79e-09 001  4.01e-08 001  3.84e-08 002  1.87e-09 186  9.51e-09 0.014   0   0   0  0
```

Note that the "Release" number, "Solver version" and "Build date" are written at the top of the screen. Following these, data are written in columns as the calculation progresses.

> **Note:** Users are encouraged to monitor the first column of solver screen output – time. When the time reaches the "End time", the calculation is complete.

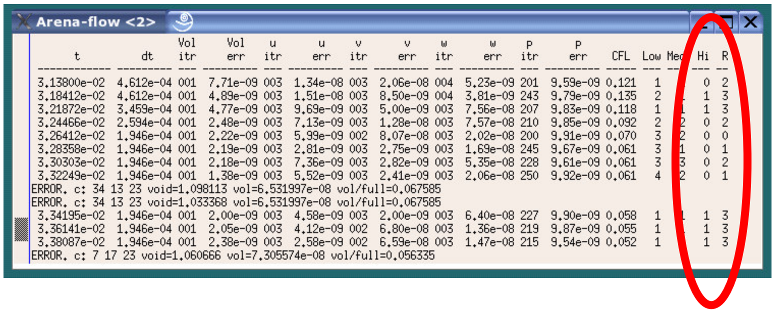
The first column (t) is time. This is the amount of time that has been simulated. When this time reaches the "End time", the calculation is complete.

The rest of the columns are provided for support purposes and are not intended for most users. These columns are described below:

- Time step (dt) – this is used by ***Arena-flow***®'s CPFD™ solver. It is automatically controlled by a sophisticated algorithm to maintain the accuracy and stability of the solution as the calculation progresses

- Fluid solver information (Vol itr, Vol err, u itr, u err, v itr, v err, w itr, w err, p itr, p err, CFL) – these are related to the fluid solver in ***Arena-flow***®. These display the number of iterations and residual information for the volume fraction, x-component of fluid velocity, y-component of fluid velocity, z-component of fluid velocity and the pressure equation. CFL is the non-dimensional time step used by the solver.

- Particle solver information (Low, Med, Hi, R) – these refer to the success of convergence of the sand stress model. Zero values are ideal. Non-zero values in the Hi and R columns indicate that the solver is having difficulty with the particle solver as shown below.



```
Arena-flow <2>
                          Vol     Vol    u        u     v        v     w        w     P        P
      t           dt      itr     err    itr      err   itr      err   itr      err   itr      err      CFL  Low Med  Hi  R
------------ --------- --- --------- --- --------- --- --------- --- --------- --- --------- ----- --- --- --- --
 3.13800e-02  4.612e-04 001  7.71e-09 003  1.34e-08 003  2.06e-08 004  5.23e-09 201  9.59e-09 0.121   1 [illegible] 0  2
 3.18412e-02  4.612e-04 001  4.89e-09 003  1.51e-08 003  8.50e-09 004  3.81e-09 243  9.79e-09 0.135   2 [illegible] 1  3
 3.21872e-02  3.459e-04 001  4.77e-09 003  9.69e-09 003  5.00e-09 003  7.56e-08 207  9.83e-09 0.118   1   1   1  3
 3.24466e-02  2.594e-04 001  2.48e-09 003  7.13e-09 003  1.28e-08 003  7.57e-08 210  9.85e-09 0.092   2   2   0  2
 3.26412e-02  1.946e-04 001  2.22e-09 003  5.99e-09 002  8.07e-08 003  2.02e-08 200  9.91e-09 0.070   3   2   0  0
 3.28358e-02  1.946e-04 001  2.19e-09 003  2.81e-09 003  2.75e-09 003  1.69e-08 245  9.67e-09 0.061   3   1   0  1
 3.30303e-02  1.946e-04 001  2.18e-09 003  7.36e-09 003  2.82e-09 003  5.35e-08 228  9.61e-09 0.061   3   3   0  2
 3.32249e-02  1.946e-04 001  1.38e-09 003  5.52e-09 003  2.41e-09 003  2.06e-08 250  9.92e-09 0.061   4   2   0  1
ERROR. c: 34 13 23 void=1.098113 vol=6.531997e-08 vol/full=0.067585
ERROR. c: 34 13 23 void=1.033368 vol=6.531997e-08 vol/full=0.067585
 3.34195e-02  1.946e-04 001  2.00e-09 003  4.58e-09 003  2.00e-09 003  6.40e-08 227  9.90e-09 0.058   1   1   1  3
 3.36141e-02  1.946e-04 001  2.05e-09 003  4.12e-09 002  6.80e-08 003  1.36e-08 219  9.87e-09 0.055   1 [illegible] 1  3
 3.38087e-02  1.946e-04 001  2.38e-09 003  2.58e-09 002  6.59e-08 003  1.47e-08 215  9.54e-09 0.052   1 [illegible] 1  3
ERROR. c: 7 17 23 void=1.060666 vol=7.305574e-08 vol/full=0.056335
```

Note: To prevent false confidence in inaccurate results, your calculation will halt if the final accuracy of your solution may be compromised.

If the particle component of the CPFD™ solver has even more difficulty converging, error messages may appear. Your calculation will halt if the final accuracy of your solution may be compromised. This helps you avoid having false confidence in inaccurate results.

If the calculation halts due to an error, you will see messages similar to the following.

```
ERROR. c: 34 13 23 void=1.098113 vol=6.531997e-08 vol/full=0.067585
ERROR. c: 34 13 23 void=1.033368 vol=6.531997e-08 vol/full=0.067585
```

Tip: Try these if you get errors during your calculation run.

In most cases when this occurs it can be avoided by performing the following:

- Make sure your cells are of a fairly uniform size and aspect ratio. This is more important for CPFD™ solvers than for CFD solvers. Ensure that the length of your largest Cartesian cell in any linear direction (x, y, or z) is no more than 2 or 2.5 times the length of your smallest Cartesian cell in the same direction.
- Use more particles. Try an "Initial computational particles per cell" of "medium / high" or more.

Contact support if the error still occurs; be sure to include the error information.

Additional output files

The primary ***Arena-flow***® output consists of GMV post-processing files (Gmv.* and 00*). Many additional output files are created for support purposes. These can be classified broadly as "log" files, "otp" files, IC_* files and data files.

Files with the ".log" extension are typically summaries of the project run. The "history.log" file is a permanent record of much of the screen and error information during the solver execution. The "info.log" file contains information regarding project initialization. The "POPUL*.log" files contain information regarding the particles in the calculation domain. The "grid.log" file contains grid generator information. Please include the history.log and info.log files when contacting support for help with a specific project, unless directed to do otherwise.

Important: Please include the history.log and info.log files when contacting support.

File with the ".otp" extension contain more detailed information than log files. Only send these to support if requested to do so.

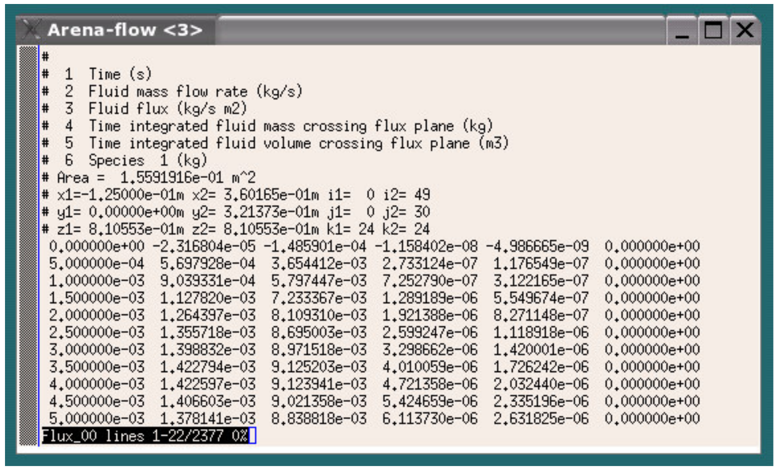
Files which begin with "IC_" are restart files. The solver may be restarted using any of these files. The "IC_" file with no trailing numbers is a temporary restart, or "backtrack" file. This file is overwritten frequently to prevent loss of data during an unscheduled computer shutdown (i.e. crash). If the "IC_" file contains trailing numbers, it represents a permanent restart file at the instance in time represented by the trailing numbers.

There are two main types of data files created by ***Arena-flow***®: transient data and flux data. Transient data files are written if requested in the "Transient Data Output" window. The filename is specified in the window; "trans.data" is the default filename. Transient data files are columnar text files and may be viewed with any text editor. The format is given in the first few lines of the file. The transient data may be plotted with any plotting utility which reads columnar data.

Flux files are written if flux planes are defined during the project set up. Flux files are named "Flux_##" where "##" represents the flux plane "ID". The output file is in a plain text format, and may be graphed via any plotting utility supporting columnar input. The file format is contained in the first few lines.

Plotting data

Examples of columnar data files used by ***Arena-flow***® are flux, transient data, transient boundary condition and sand size distribution files. A sample columnar data file (flux file format) is shown below.



Arena-flow <3>

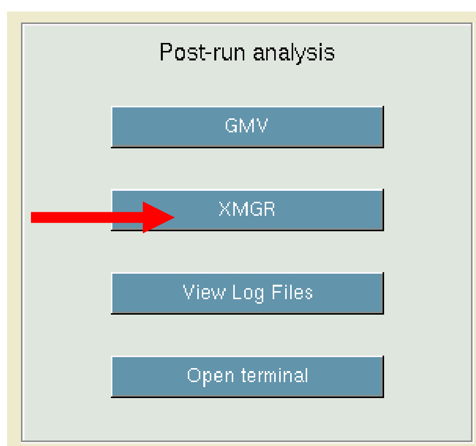
```
#
#  1  Time (s)
#  2  Fluid mass flow rate (kg/s)
#  3  Fluid flux (kg/s m2)
#  4  Time integrated fluid mass crossing flux plane (kg)
#  5  Time integrated fluid volume crossing flux plane (m3)
#  6  Species  1 (kg)
# Area =  1.5591916e-01 m^2
# x1=-1.25000e-01m x2= 3.60165e-01m i1=  0 i2= 49
# y1= 0.00000e+00m y2= 3.21373e-01m j1=  0 j2= 30
# z1= 8.10553e-01m z2= 8.10553e-01m k1= 24 k2= 24
 0.000000e+00 -2.316804e-05 -1.485901e-04 -1.158402e-08 -4.986665e-09  0.000000e+00
 5.000000e-04  5.697928e-04  3.654412e-03  2.733124e-07  1.176549e-07  0.000000e+00
 1.000000e-03  9.039331e-04  5.797447e-03  7.252790e-07  3.122165e-07  0.000000e+00
 1.500000e-03  1.127820e-03  7.233367e-03  1.289189e-06  5.549674e-07  0.000000e+00
 2.000000e-03  1.264397e-03  8.109310e-03  1.921388e-06  8.271148e-07  0.000000e+00
 2.500000e-03  1.355718e-03  8.695003e-03  2.599247e-06  1.118918e-06  0.000000e+00
 3.000000e-03  1.398832e-03  8.971518e-03  3.298662e-06  1.420001e-06  0.000000e+00
 3.500000e-03  1.422794e-03  9.125203e-03  4.010059e-06  1.726242e-06  0.000000e+00
 4.000000e-03  1.422597e-03  9.123941e-03  4.721358e-06  2.032440e-06  0.000000e+00
 4.500000e-03  1.406603e-03  9.021358e-03  5.424659e-06  2.335196e-06  0.000000e+00
 5.000000e-03  1.378141e-03  8.838818e-03  6.113730e-06  2.631825e-06  0.000000e+00
Flux_00 lines 1-22/2377 0%
```

Note: Lines in columnar data files beginning with a "#" character are comments.

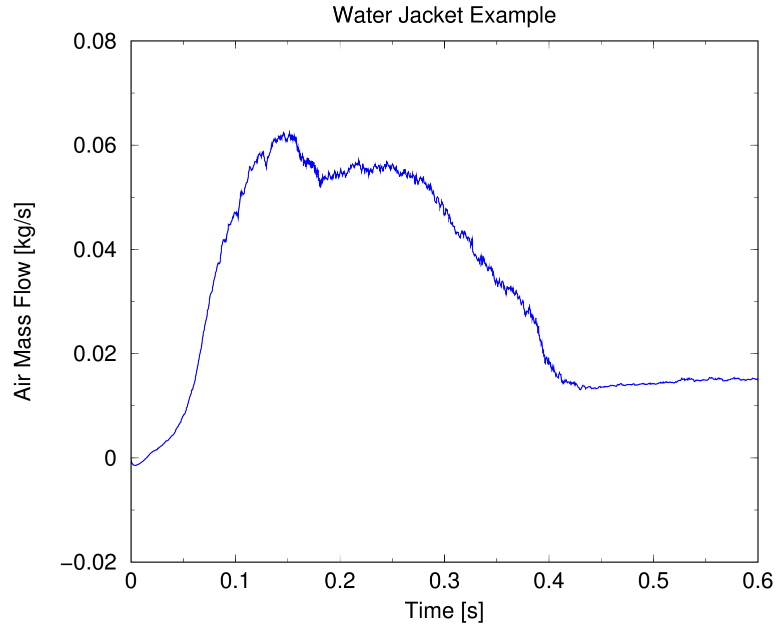
Lines beginning with a "#" character are comments. The file format is listed in the first few lines. In this example the first column is time; the second is the fluid mass flow rate and so forth.

Note: Columnar data files may be plotted using any graphing utility supporting columnar data formats such as xmgr, Excel or OpenOffice.

Columnar data files produced by ***Arena-flow***® may be plotted using any graphing utility supporting columnar data formats. Examples of graphing utilities are xmgr, Excel and OpenOffice. ***Arena-flow***® contains a link to xmgr on the "Post-run" page of any process-oriented interface.

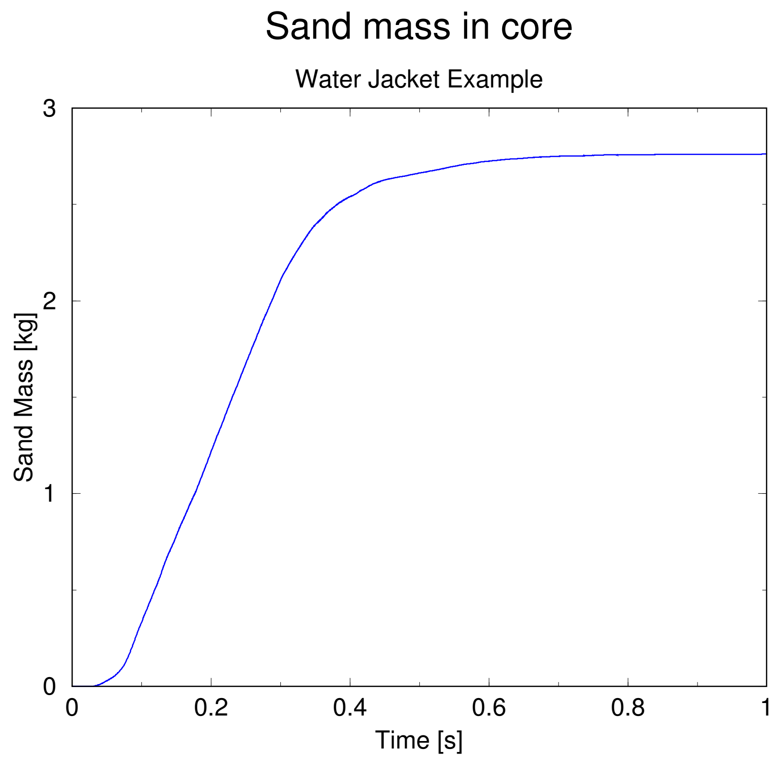


Air mass flow into core



The plots of columnar data produced by ***Arena-flow***® provide users with quantitative information regarding core blowing and curing. The above example plots the air mass flow into the core during the blow cycle on the vertical axis against time on the horizontal axis. This information is contained in columns 1 and 2 of the above flux file. Notice the air flow ramping up with blow pressure. Also, notice the decrease around 0.4 seconds. Quantitative plots can yield additional information about the complex, transient core filling behavior.

The example below is a plot of the mass of sand that has entered the core vs. time. This information is contained in columns 1 and 6 of the above flux file. Notice that most of the sand has entered the core by 0.4 seconds. Also, the slow “roll-over” at the top of the curve indicates that all the blow tubes are not filled at the same time. This can be an indication of suboptimal filling behavior.



> **Important:** Always compare the data plots with images and animations and consider what causes the observed behavior.

Always use ***Arena-flow***®'s quantitative graphical information in conjunction with the images and animations created by GMV. Be sure to consider how a phenomenon observed in the quantitative information relates to the images or animations. Understanding your process is the first step toward improving it.

Additional help using *Arena-flow*®

This User Guide is a first step in applying ***Arena-flow***® toward systematic sand core engineering at your foundry. Time spent reviewing the information contained herein is spent well. However, this User Guide is just a beginning and a reference toward fully utilizing ***Arena-flow***®'s full potential.

More information regarding project setup and analysis of results is presented in the various ***Arena-flow***® training courses. Contact your sales representative or support engineer for more information regarding course content and scheduling.

Additionally, all ***Arena-flow***® users who have attended an ***Arena-flow***® training class and have current maintenance terms may contact user support:

Arena-flow LLC, attn User Support
10899 Montgomery Blvd. NE, Suite B
Albuquerque, NM 87111 USA
Phone: +1.505.275.3849
Fax: +1.505.275.3346
email: support@arena-flow.com

Tip: When contacting user support, you may be required to send these file.

Often the support engineer will require the following files:

- the project file (*.prj)
- the CAD (*.stl or *.STL, unless linked)
- the boundary conditions (*.i)
- the sand size distributions (*.sze)
- the vent file (*.tvt)
- history.log
- info.log

The support engineer may also request additional information such as:

- ***Arena-flow***® release number
- ***Arena-flow***® grid generator version number
- ***Arena-flow***® solver version and build date
- additional files

Finally, ***Arena-flow***® continues to be improved through the feedback of our important user base. We welcome your thoughts – please do not hesitate to contact us with your feedback and suggestions.

3 Index

T

U

V

W

X

Z

4 References

[1] Schleg, P. 2003., *Technology of Metalcasting*, American Foundry Society, Des Plaines, IL, p.1.

[2] Blaser, P., and Yeomans, N., 2006, "Sand Core Engineering & Process Modeling," Japan Foundry Society, Vol. 2, No. 2, February 2006, pp. 420-427.

[3] Yeomans, N., and Blaser, P., 2006, "Predicting the Process," Foundry Management & Technology, January 2006, pp 48-49.

[4] Winartomo, B., Vroomen, U., and Buhrig-Polaczek, A., Pelzer, M. 2005, "Multiphase modeling of core shooting processes," International Journal of Cast Metals Research, Vol. 18, No. 1.

[5] Lefebvre, D., Mackenbrock, A., Vidal, V., and Haigh, P., 2005, "Development and use of simulation in the design of blown cores and moulds," Foundry Trade Journal, February 2005.

[6] CPFD™ is a trademark of Arena-flow, LLC.

[7] Snider, D. M., 2001, "An Incompressible Three-Dimensional Multiphase PIC Model for Dense Phase Flows," Journal of Computational Physics, **170**, pp. 523-549.

[8] Snider, D. M., Williams, K. A., and Johnson, R. A., 2004, "Mulitphase Particle-in-Cell Simulations of Dense-Phase Flows in Cyclone Separators," Paper HT-FED2004-56665, American Society of Mechanical Engineers, New York, NY.

[9] Williams, K. A., Snider, D. M., Torczynski, J. R., Trujillo, S. M., and O'Hern, T. J., 2004, "Multiphase Particle-in-Cell Simulations of Flow in a Gas-Solid Riser," Paper HT-FED2004-56594, American Society of Mechanical Engineers, New York, NY.

[10] Blaser, P. J., Snider, D. M., Williams, K. A., Cook, A. E., and Hoover, M., 2004, "Process Modeling: Lost-Foam Pattern Filling," Paper IMECE2004-60483, American Society of Mechanical Engineers, Anaheim, CA.

[11] ***Arena-flow***® is a registered trademark of Arena-flow, LLC. ***Arena-flow***® is marketed and distributed world wide by Ashland Inc.

[12] ***Seefoam***™ is a trademark of Arena-flow, LLC.

[13] ***Barracuda***™ is a trademark of Arena-flow, LLC.

[14] Williams, K. A., Snider, D. M., Walker, M., and Palczewski, S., 2002, "Process Modeling: Sand Core Blowing," AFS Transactions 02-141.

[15] Walker, M., Palczewski, S., Snider, D., and Williams, K., 2002, "Modeling Sand Core Blowing: Simulation's next Challenge". Modern Casting, April 2002, pp. 41-43.

[16] Buchholz, A., Showman, R., Snider, D., and Williams, K. "Experiences with Modeling the Core Blowing Process."

[17] Blaser, P., 2006, "Arena-flow® Grid Resolution of Blow Tubes," Technical Note 77, Arena-flow, LLC, Albuquerque, NM.

[18] Schleg, P. 2003., *Technology of Metalcasting*, American Foundry Society, Des Plaines, IL, p.39.

[20] http://www.x-div.lanl.gov/XCM/gmv/GMVHome.html