

# SANDIA REPORT

SAND2012-1720

Unlimited Release

Printed March 2012

## Prism Users Guide Version 1.0

V. Gregory Weirs

Prepared by  
Sandia National Laboratories  
Albuquerque, New Mexico 87185 and Livermore, California 94550

Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

Approved for public release; further dissemination unlimited.

Issued by Sandia National Laboratories, operated for the United States Department of Energy by Sandia Corporation.

**NOTICE:** This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government, nor any agency thereof, nor any of their employees, nor any of their contractors, subcontractors, or their employees, make any warranty, express or implied, or assume any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represent that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government, any agency thereof, or any of their contractors or subcontractors. The views and opinions expressed herein do not necessarily state or reflect those of the United States Government, any agency thereof, or any of their contractors.

Printed in the United States of America. This report has been reproduced directly from the best available copy.

Available to DOE and DOE contractors from  
U.S. Department of Energy  
Office of Scientific and Technical Information  
P.O. Box 62  
Oak Ridge, TN 37831

Telephone: (865) 576-8401  
Facsimile: (865) 576-5728  
E-Mail: [reports@adonis.osti.gov](mailto:reports@adonis.osti.gov)  
Online ordering: <http://www.osti.gov/bridge>

Available to the public from  
U.S. Department of Commerce  
National Technical Information Service  
5285 Port Royal Rd.  
Springfield, VA 22161

Telephone: (800) 553-6847  
Facsimile: (703) 605-6900  
E-Mail: [orders@ntis.fedworld.gov](mailto:orders@ntis.fedworld.gov)  
Online order: <http://www.ntis.gov/help/ordermethods.asp?loc=7-4-0#online>



SAND2012-1720  
Unlimited Release  
Printed March 2012

# **Prism Users Guide Version 1.0**

V. Gregory Weirs  
Optimization and Uncertainty Quantification Department  
Sandia National Laboratories  
P.O. Box 5800  
Albuquerque, New Mexico 87185

## **Abstract**

Prism is a ParaView plugin that simultaneously displays simulation data and material model data. This document describes its capabilities and how to use them. A demonstration of Prism is given in the first section. The second section contains more detailed notes on less obvious behavior. The third and fourth sections are specifically for Alegra and CTH users. They tell how to generate the simulation data and SESAME files and how to handle aspects of Prism use particular to each of these codes.



# CONTENTS

1. Prism Demonstration .....	7
2. Notes on Features and Usage.....	11
3. Prism for Alegra Users.....	15
4. Prism for CTH Users .....	19
Distribution .....	21



# 1. PRISM DEMONSTRATION

Prism is a ParaView<sup>1</sup> plugin for simultaneously displaying simulation data and material model data. Simulation data includes element (cell) or node (point) data, or tracer particles (either Lagrangian or Eulerian.) Material model data includes equation of state data, or scalar properties such as opacity or conductivity, in SESAME<sup>2</sup> format. ParaView's filters and selection/extraction capabilities are used to identify the simulation data of interest. The Prism filter accepts this simulation data as in input in the ParaView pipeline, and reads the material model data from a file in ASCII SESAME format. The Prism filter displays the material model data as a surface or a set of contours, and the simulation data as symbols on the surface. This demonstration was developed using the ParaView 3.14.0 release. For this demonstration, the test data includes the three files: SymmetricImpact2.exo, SymmetricImpact2.hscth, and proxium.asc. This data is available in the file ParaViewData-3.14.0.zip at <http://paraview.org/paraview/resources/software.html> in the directory Data/Prism/SymmetricImpact2.

Steps:

- 1.) **Start ParaView.**
- 2.) **Load the Prism plugin.** Go to the Tools menu, and select Manage Plugins. A window will open with two panels, side-by-side. Under "Local Plugins" (right panel) click on PrismPlugin once to highlight it. Then click the button labeled "Load Selected" at the bottom of the panel. If you are connected to a remote server, also load the PrismPlugin on the server side in the same way. If successfully loaded, three icons will appear on the ParaView toolbar: one with a prism and a beam of light being split into its component colors (Prism filter), one with a quarter-circle rainbow (SESAME reader), and one like the first prism but reversed, and with red brackets in the corners (Prism Change View Scale.)
- 3.) **Optional:** If you want, the Prism plugin can be loaded automatically when ParaView starts up. To enable this, go to the Tools menu and select Manage Plugins as in step 2. Double click on PrismPlugin to display more information about the plugin. The last line of this information is "AutoLoad" with a check box. If the checkbox is clicked, it will display a check and ParaView will load the Prism plugin when you start ParaView in the future. There is also a similar check box for the remote server.
- 4.) **Open the exodus file**, SymmetricImpact2.exo. Select all variables to load. Press "Apply".
- 5.) **Apply a threshold filter.** Choose "VOLFR1" from the "Scalars" dropdown menu and enter 0.01 for the "Lower Threshold". Press "Apply". This selects all the elements that contain material 1 as the simulation data.
- 6.) **Apply the Prism filter** by clicking on the Prism icon on the toolbar. A pop-up window will open so that the SESAME data file can be loaded. Choose "proxium.asc"

---

<sup>1</sup> ParaView. <http://paraview.org>

<sup>2</sup> S. P. Lyon and J. D. Johnson, editors. "SESAME: The Los Alamos National Laboratory Equation-Of-State Database", LA-UR-3407, 1992.  
[http://t1web.lanl.gov/doc/SESAME\\_3Ddatabase\\_1992.html](http://t1web.lanl.gov/doc/SESAME_3Ddatabase_1992.html)

by clicking on it; then click “OK”. A second view will open on the main ParaView window. The Prism Filter should also be visible on the pipeline browser, with four filters below it (SESAME Surface, SESAME Curve, SESAME Contours, Simulation Data.) When PrismFilter is highlighted in the Pipeline Browser, the Object Inspector will display the Properties, Display, or Information for the Prism filter, depending on which tab is highlighted.

- 7.) On the “Properties” panel, there are two tabs: “SESAME Data” and “Simulation Data”. First we will go through the properties under the “SESAME Data” tab.
- 8.) **Choose SESAME variables to display.** Choose “Density” as the X-Axis variable and “Temperature” as the Y-Axis variable. (These are probably the default values.) For the z-axis, choose “Total EOS (Pressure)” for the warp variable.
- 9.) **Apply log scaling to each axis.** Select the check boxes for “Log Scale” on the X-Axis, Y-Axis and Warp Surface.
- 10.) For now skip over the checkboxes for the curves (400 series tables, just below the “Warp Surface” section) and the contours, below the “Curve Tables”.
- 11.) **Apply unit conversions.** Scroll down to the “Conversions” section of the properties panel. Click on the check box for “SESAME To SI”. This converts the data in the SESAME file from SESAME units to SI units, which is necessary to match the SI units of the data in the SymmetricImpact2.exo file.
- 12.) **Select SI unit conversion** by checking the SESAME to SI box. This converts the sesame data to SI units, so that it matches the units of the simulation data in SymmetricImpact2.exo.
- 13.) **Associate Simulation variables and SESAME variables.** Click on the “Simulation Data” tab at the top of the “Properties” panel. There are three drop down boxes. Each box selects the variable in the simulation dataset to match with a variable in the SESAME dataset. Choose “Density1” for “Associate With X Axis”, “Temperature1” for “Associate With Y Axis”, and “Pressure1” for “Associate With Z Axis”.
- 14.) **Apply the Prism filter.** Press “Apply”. The SESAME data should appear as a surface and the simulation data as points. (At time=0, all the points appear as a single point, just off the surface.)

At this point you can play with all the controls – rotate the SESAME surface to see it from a different angle, color by a variable, etc. Press the “Play” button, which will animate forward in time; the simulation data will move across the surface as the animation proceeds. You can enable the curves, which show phase transitions as lines on the surface. You can show contours of another variable on the surface. Most things behave as standard ParaView objects, as you would expect. Linked selection should also work, as long as you realize that the simulation data displayed in the SESAME view is cell data – so use Select Cells On. Note that since this particular simulation is symmetric, each visible point is actually two collocated points. If you use Select Cells On you get the one “on top.”

The first example showed element (cell) data from full field numerical solutions as the simulation data. In the second example tracer data will be used. Follow the same steps as above, but replace steps 4 and 5 with:



- 4.) **Open the hscth file**, SymmetricImpact2.hscth. The default options are fine; click “Apply”. A spreadsheet view will replace the default 3D graphics view.
- 5.) **Apply the filter “Prism: Table To Points”**. This filter is only be visible in the “Alphabetical” list under the “Filters” menu, and only after the Prism plugin has been loaded. Set the “X Column”, “Y Column”, and “Z Column” drop down boxes to “XPOSITION”, “YPOSITION”, and “ZPOSITION”. Set the “Global Element Id Column” to “Element-ID”. Press “Apply”.

Note that the tracer data is also cell data, not point data. This may change in the future, but is required for now to enable linked selection.



## 2. NOTES ON FEATURES AND USAGE

The demonstration gave a quick survey of Prism's capabilities. More detailed notes are given below, as well as a list of known issues.

- 1.) General strategy for showing simulation data on SESAME surfaces: The Prism filter takes an input from the preceding element of the pipeline and interprets it as the simulation data. When running ParaView, then, apply filters to extract the simulation data you want to show on the surface *before* applying the Prism filter. This often involves a threshold or Clip (by scalar) filter to select the data for a single material of several in the simulation.
- 2.) Unit Conversions: Predefined conversions. The Conversions part of the property panel is used to convert the SESAME data to the units of the simulation data. If the "SESAME To SI" or "SESAME To cgs" box is checked, the "Conversions" part of the Properties panel displays a table with columns specifying the variable name, the conversion applied, and the factor that multiplies the value in SESAME units to obtain the value in SI or cgs units. What is not obvious is that the entries in the "Conversion" column can be double-clicked to expose a drop down menu of unit conversions for different variables.
- 3.) Unit Conversions: Custom Conversions. Default values for SI and cgs units are provided. If the "Custom" check box is selected, the numerical values for the conversions (in the "Factor" column) can be edited by double-clicking on the factor value.
- 4.) Unit Conversions: SESAMEConversions.xml file. For variables in the SESAME file, as well as additional variables (see below), conversions can be specified in an xml file. This provides a more convenient way to preserve commonly used conversions that are different than the defaults, so that the file can just be read rather than repeatedly entering factors as in note 2. Use the File Name dialog box, labeled "..." on the Conversions panel, and select the file. An example named "SESAMEConversions.xml" is included with the SymmetricImpact dataset. While SESAMEConversions.xml provides an example, any filename can be used.
- 5.) SESAME file format. Prism reads files in SESAME "Ship" format, and expects that a file will only contain tables for one material. Note that the EOS data from any model can be displayed in Prism, as long as it is in a SESAME file.
- 6.) SESAME file format: The format defined by LANL (see earlier footnote in the demonstration section) is the reference format, but G. I. Kerley defined an alternative SESAME format for (at the time) faster reading. Prism reads both the LANL format (ASCII "Ship" format) and the ASCII Kerley format (often called "Kerley SESAME" or "Sandia SESAME.") The proxium.asc file is written in Kerley SESAME format, while the proxium.1.301 is in LANL SESAME format.
- 7.) The SESAME file format specifies the order of variables in the file for each table, i.e., for the 301 table, the density and temperature values are specified as (essentially) one-dimensional arrays, and the other variables are two-dimensional arrays as functions of the density and temperature. The two-dimensional arrays are the pressure, then the internal energy, and finally, the Helmholtz free energy. All three two-d variables might not be present, but the order is fixed; if there is only one two-

d variable, it is the pressure, if two are present then the first is the pressure and the second is the internal energy.<sup>3</sup>

- 8.) SESAME file format: Additional Variables. For 301 tables, additional variables can be added beyond the Helmholtz free energy; in this case there is no guidance on the identity of the variable from the format. Prism will read and display these additional variables, and apply unit conversions as specified. For example, if the speed of sound is used as the fifth variable, and is written to the SESAME file 301 table in km/s (the SESAME units for speed), the units can be converted by selecting “Speed” in the drop down menu in the Conversions column for “Variable 6.” (The density and temperature are the first and second variables, then pressure, internal energy, and Helmholtz free energy.) As an example, the SymmetricImpact dataset includes speed of sound in the SymmetricImpact2.exo file as SoundSpeed1, and speed of sound is the sixth variable in the 301 table in the proxium.asc SESAME file. Prism’s ability to read additional variables is not restricted to 301 tables; they can be added to any table and Prism will read them.
- 9.) The SESAME reader (rainbow icon) is the same as the Prism filter, but does not accept simulation data as input. Use the reader to display SESAME files when you don’t have or want to show simulation data. All the SESAME Data controls on the Properties panel behave in the same way as the Prism filter.
- 10.) The SESAME Curve filter displays 400 series tables, which contain curves in thermodynamic space. The curves explicitly identify phase changes. Few SESAME files include 400 series tables, but if they are included Prism displays them with 300 series tables using the SESAME Curve filter from Prism.
- 11.) Prism Change View Scale: Prism open a special view (a special rendering window) that behaves somewhat differently than the traditional three-dimensional view used for simulation data. In the Prism view, an automatic rescaling of the different axes is applied. This rescaling accounts for the different types of variables on each axis; in the traditional three-d view, the variables are all of the same type and units, e.g., length in the X-, Y-, and Z-dimensions, but in the Prism view they are of different thermodynamic quantities that cannot be compared, even if they were in the same units. The Prism Change View Scale allows the user to control this autoscaling in the Prism view. By default, the bounds (min and max) of each variable are determined for all of the data presented in the view. If multiple SESAME files are visible in the same view the bounds are the superset of the bounds for each file. The axes are rescaled such that the ratio of the bounds are 1:1:1 for the X-, Y-, and Z-axis variables. When the threshold sliders on the Properties panel are used for the X- or Y-axes, the data displayed is restricted, but by default, the axes are still scaled from the bounds of the data. The user can select “threshold bounds” on the Prism Change View Scale window to instead rescale the axes based on the threshold values. Finally, the user can select Custom on the Prism Change View Scale to directly specify how the axes are rescaled; the values

---

<sup>3</sup> In some SESAME libraries, particularly those produced by G. I. Kerley, the second variable is entropy for some materials. These do not strictly comply with the SESAME format; and furthermore, there is no way to tell that the second variable is not internal energy from the file itself.

specified in the text boxes will define the axis range scaled to 0:1 with respect to the bounds of other axes. Note the Prism Change View Scale does not affect what data is displayed, only how it is displayed. Be aware that, particularly when log scaling is enabled, the view scale can significantly alter presentation of the data.

- 12.) Log Scaling: negative values. Log scaling can be applied to any of the variables by selecting the appropriate boxes on the Properties panel. While density and temperature are most often the X- and Y-axes and have positive values, the pressure and energy may be negative. When that is the case, the negative values are replaced by a threshold value equal to 1 (or, equivalently, the log of the negative value is replaced by 0.) The resulting surface usually has a broad, flat, “floor” region in which the pressure appears to be invariant with density and temperature. In cases where large regions of the data are positive but less than 1, the floor is actually in the middle of the pressure or energy range, which can be misleading. In future versions of Prism, the threshold value will likely be set to two or three orders smaller than the smallest positive data value.
- 13.) Additional filters in the Prism view. Users control ParaView by adding filters in sequences to produce a data processing pipeline. The Prism filter actually adds four filters and displays them in a special view (the Prism view). Sometimes all four filters are not required and only one or two of them actually process data, but the four are always added visible in the Prism section of the pipeline. While this suggests that these are the only available filters for the SESAME and simulation data, in fact additional filters can be added. All of the unit conversions and axis rescaling are applied to these additional filters. The Prism filter provides a contour filter, but as an example consider an additional contour filter applied to the SESAME Surface (not the SESAME Contour.) If the first contour filter displayed contours of density (isochores), and the second displayed contours of temperature (isotherms), a grid of pattern would appear on the surface if density and temperature were selected for the x- and y-axis variables. Note that the SESAME surface, SESAME Curve, and SESAME Contour filters output different selections of the SESAME data, while the simulation data is accessible from the Simulation Data filter.

#### Known Issues:

- 1.) Linked selection. When rubber-band selecting in the Prism view with “Select Cells Through”, simulation data is not selected. “Select Cells On” does not have this problem, and selecting in the view with the exodus data works with both “Select Cells On” and “Select Cells Through.”
- 2.) Cube Axes do not correctly display labels or tick marks for variables that have been log-scaled. The bounds are correct though.
- 3.) When reading 600 series tables, the Conversions part of the Properties panel does not correctly identify the third, warp-surface variable; it always lists “Mean Ion Charge”. The default units are also those for Mean Ion Charge. The user has to double click and choose the appropriate variable from the drop down list for the correct unit conversion.

These issues, and some others as well, are being addressed and should be corrected in the next version of ParaView.



### 3. PRISM FOR ALEGRA USERS

Writing SESAME files from Alegra:

- 1.) Alegra will write SESAME files for use in Prism using the “plot, sesame file” command block.
- 2.) “Plot, sesame file, ..., end” writes model data for single material, if the “material = *matnum*” line is included, or all materials otherwise. Multiple “plot, sesame file, ..., end” blocks can be used to write SESAME files for different materials, with the advantage that appropriate density and temperature bounds can be set for each material.
- 3.) “Plot, sesame file, ..., end” writes all applicable models for the material: EOS, and if available, conductivity data. EOS data is written to a file with “.301” appended to the name. Conductivity data is written to a file ending with “.600” if more than one 600 table is written, or with the table number for a single table; the number of tables depends on the particular material models. Known issue: For 600 series tables, “plot, sesame file, ..., end” labels all the tables as 601 tables. This has been corrected in the Alegra source code, but some versions may not write the correct table number. Users can edit the lines with the incorrect table number by hand as a workaround.
- 4.) At this time, Alegra does not write 2T EOS models to the SESAME file.
- 5.) Determining density and temperature ranges. Alegra writes the minimum and maximum density and temperature for each material to the hisplot file (.his file) as a function of simulation time. Alegra users can use shiv to look at these variables and determine appropriate values to enter in the “plot, sesame file, ..., end” block.
- 6.) The density and temperature points are specified by the “density bounds, ..., end” and temperature bounds, ..., end” blocks, but sometimes a particular density or temperature is desired in the SESAME file. Additional densities and temperatures points can be specified using the “density = *dval1*, *dval2*, ...” and “temperature = *tval1*, *tval2*, ...” commands in the “plot, sesame file” block.
- 7.) If the free energy is not available from the EOS model used by Alegra, dummy values of the Helmholtz free energy are written to the SESAME file; these should be ignored.
- 8.) Additional variables can be written to the SESAME file by including a line “extra variable = *material variable*” in the “plot, sesame file” block. The *material variable* is the case sensitive name of a variable the material model can provide as a plot variable. When an Alegra simulation produces an output file (*runid.out*) the names of the available plot variables for each material model are listed. See the Alegra Users Manual for more information.
- 9.) There are two interfaces to EOS data that Alegra uses, one in which density and temperature are the independent variables, and another using density and internal energy. When initializing the solution at the beginning of the simulation, the density-temperature interface is used. Also, Alegra uses the density-temperature interface to the EOS model to obtain the data written to the SESAME file. During the time integration of the solution, however, Alegra calls the density-energy interface. When EOS data is tabulated as a function of density and temperature, such as for

SESAME EOS *models*, the table must be “inverted” when the density-energy interface is used; this just means that iteration must be performed to obtain the thermodynamic variables at the specified internal energy. However, “inverting” the table may introduce slight errors in the variables returned because of the approximate nature of the iteration, particularly near phase transitions. Also, many of the “clips” and techniques applied for handling extrapolation beyond the bounds of the table are only implemented in the density-energy interface. Users should be aware that Alegra simulation data might not match the data in the SESAME file because of the differences in the two EOS interfaces.

- 10.) The “plot, sesame file” block must be within the physics block, i.e., inside the “hydrodynamics, ..., end” block, “mhd, ..., end” block, or similar. If it is placed outside the physics block, an error about an unexpected keyword will result during the parsing of the Alegra input file.
- 11.) An early method for writing material model data to SESAME files is described in the material model section of the Alegra-MHD Users Manual. This alternative, “write mat, ..., end” approach is less robust and will be deprecated.

#### Alegra simulation data: exodus plot files

- 1.) Alegra users should specify plot variables written to exodus files with Prism visualization in mind. In particular, material densities, temperatures, pressures, and energies should be written since they correspond to the EOS data in the SESAME file for each material. The material speeds of sound are also useful, if the user also includes “SOUND\_SPEED” as an additional variable in the “plot, sesame file” block. For conductivity models, the mean ionization ( $\bar{z}$ ) and the parallel components of the material electrical and thermal conductivities should be written. Note that the parallel components correspond to the data in the SESAME file.

#### Alegra simulation data: hisplot history files

- 1.) Alegra writes history files named *runid*.his in hisplot format, a binary, platform dependent format whose lack of portability is hidden by shiv. ParaView will not read hisplot format. However, a utility called his2hscth, distributed with Alegra, will convert Alegra produced hisplot files to files in hscth format, an ASCII format written by Spy.
- 2.) ParaView has an hscth reader that correctly interprets the rows of the file as different output times, and associates the “.N” and “+M” in the CTH variable names as tracer N and material M. The output type of the hscth reader is a VTK “table” (unrelated to a SESAME table) and is shown in a spreadsheet view.
- 3.) Prism provides a filter called “Prism:TableToPoints” that converts a VTK table to points with coordinates and other data variables. The filter appears only in the “Alphabetical” listing of all the filters. To use this filter, select hscth variables “xposition,” “yposition,” “zposition,” and “ElementID” for the “X Column,” “Y Column,” “Z Column,” and “Global Element ID Column,” respectively. If the Alegra simulation was not three-dimensional and doesn’t have a three coordinate positions, just pick an arbitrary variable for the corresponding column. The



positions are used when hscth tracers are displayed simultaneously with exodus data. The Global Element ID variable is used to implement linked selection.

- 4.) Note that even when Prism is not used, the Prism:TableToPoints filter is useful for showing tracers in hscth files on top of field data in ParaView. Because this filter comes with the Prism plugin, the plugin must be loaded to access the filter.
- 5.) Before applying the Prism filter, use other ParaView filters to select/extract the simulation data you want to show in thermodynamic space. This almost always includes a Threshold or Clip by Scalar filter applied to the material volume fraction.
- 6.) By convention, Alegra always writes hisplot files in SI units, even when the rest of the simulation output files are written in cgs units. The utility his2hscth does not convert units, so the data in the hscth files remains in SI. Users that intend to view simulation data from Alegra exodus and hisplot files simultaneously are advised to write their data to exodus files in SI units as well.

Alegra simulation data: tracers in exodus files

- 1.) Alegra writes tracer data to exodus files using the “plot, exotracer, ..., end” block. This data is not useful for displaying tracers in Prism because only the tracer position and velocity are written; no material variables are provided.



## 4. PRISM FOR CTH USERS

In general, Prism works for CTH simulation data in the same way as for data from other codes. This document describes some specific things CTH users should be aware of when working with Prism. There are no known Prism issues relating to CTH data specifically; however, Prism has not been extensively tested on CTH data.

Writing SESAME files from Spy:

- 1.) Spy will write a SESAME file for each material using the following command.  
`WriteEosDataSesameShip(filename, dens_min, dens_max, num_dens, dens distribution, temp_min, temp_max, num_temp, temp distribution)`  
*filename* is a string (enclose it in quotes) that is the base of the SESAME file names to be written. Integers referring to the material number are appended to this base to distinguish the different material EOSs.  
*dens\_min*, *dens\_max*, *num dens*, and *dens distribution* describe the densities at which the EOS is evaluated. Dens distribution can be “lin” or “log” to specify a linear or logarithmic distribution of densities. The last four arguments describe the temperatures in the same way as the densities are described.
- 2.) Dummy values of the Helmholtz free energy are written to the file; these should be ignored.
- 3.) At this time, there is no capability to write additional variables to the SESAME files.
- 4.) The `WriteEosDataSesameShip` command is available in CTH versions released after September, 2011.
- 5.) At this time, `WriteEosDataSesameShip` writes a SESAME file for each material. A drawback is that the pressure and internal energy are evaluated at the same set of densities and temperatures for all the materials. A workaround is to call `WriteEosDataSesameShip` more than once, specifying different filenames, densities, and temperatures. Extra files will be produced, but SESAME files with the desired material-specific density and temperature ranges will be generated. Future versions of the `WriteEosDataSesameShip` will likely require the material to be specified explicitly as one of the arguments.

Writing CTH simulation data from Spy:

- 1.) As “derived” variables, Spy does not write material densities (DENS<sub>M</sub>) to spcth files. However, the CTH reader in ParaView will optionally compute the material densities for you if the variables they depend on are present. Generally “M,” “VOLM,” “TM,” “PM,” and “EM” should be written to the spcth file to use Prism.
- 2.) Spy will write material densities to hscth files for tracers, however, each material density must be specified explicitly. That is, “DENS<sub>M+1</sub>,” “DENS<sub>M+2</sub>,” etc., must be in the SaveHis arguments; in contrast to the behavior for other material variables, specifying “DENS<sub>M</sub>” will not write the material densities for all the materials to the hscth file. Generally “PM,” “TM,” “EM,” and each of the “DENS<sub>M+m</sub>” should be written to the hscth file to use Prism; also consider “TKM”.

Reading Spy files and using Prism

- 1.) ParaView has an hscth reader that correctly interprets the rows of the file as different output times, and associates the “.N” and “+M” in the CTH variable names

as tracer N and material M. The output type of the hscth reader is a VTK “table” (unrelated to a SESAME table) and is shown in a spreadsheet view.

- 2.) Prism provides a filter called “Prism:TableToPoints” that converts a VTK table to points with coordinates and other data variables. The filter appears only in the “Alphabetical” listing of all the filters. To use this filter, select hscth variables “xpos,” “ypos,” “zpos,” and “TracerID” for the “X Column,” “Y Column,” “Z Column,” and “Global Element ID Column,” respectively. If the CTH simulation was not three-dimensional and doesn’t have a three coordinate positions, just pick an arbitrary variable for the corresponding column. The positions are used when hscth tracers are displayed simultaneously with spcth data. The Global Element ID variable is used to implement linked selection.
- 3.) Note that even when Prism is not used, the Prism:TableToPoints filter is useful for showing tracers in hscth files on top of spcth data in ParaView. Because this filter comes with the Prism plugin, the plugin must be loaded to access the filter.
- 4.) Before applying the Prism filter, use other ParaView filters to select/extract the simulation data you want to show in thermodynamic space. This almost always includes a Threshold or Clip by Scalar filter applied to the material volume fraction.
- 5.) Unit Conversions. CTH uses cgs units, except that the temperature is written in electron volts (eV). First check the “SESAME To cgs” check box, then double-click the box containing “Temperature – K to K” conversion. In the drop down menu, you can select “Temperature eV – K to eV” so that the SESAME data is converted to the same units as CTH data. For hscth files, you could also output material temperatures in Kelvin by specifying “TKM” in the SaveHis argument list; in that case don’t apply the K to eV conversion.

## DISTRIBUTION

1 MS0899 RIM-Reports Management 9532 (electronic copy)





**Sandia National Laboratories**