

KB1

**Field Precision**

Copyright 2000

PO Box 13595, Albuquerque, New Mexico 87192

Telephone: 505-296-6689      FAX: 505-294-0222

E Mail: [techinfo@fieldp.com](mailto:techinfo@fieldp.com)      Internet: [www.fieldp.com](http://www.fieldp.com)

## Section 1. Introduction

**KB1** performs hydrodynamic simulations of materials at high temperature and pressure. The code handles one-dimensional problems in planar, cylindrical and spherical geometries. **KB1** uses element-based methods based on difference equations derived from integral conservation relationships. The approach has several advantages.

# Elements have an unambiguous material identity.

# Zone refinement follows automatically from the compression and expansion of elements.

# There are no singularities associated with shock convergence on-axis in cylindrical and spherical problems.

# The method has good stability properties, even for abrupt collisions between objects.

The mathematical foundation of **KB1** is described in the manual *KB Methods and Tutorials* and the reference S. Humphries and C Ekdahl, *Finite-element simulation code for high-power magnetohydrodynamics*, *Laser and Particle Beams* **16** (1998), 405.

The **KB1** system consists of the following programs:

### **KB1**

Main simulation program controlled by an input script.

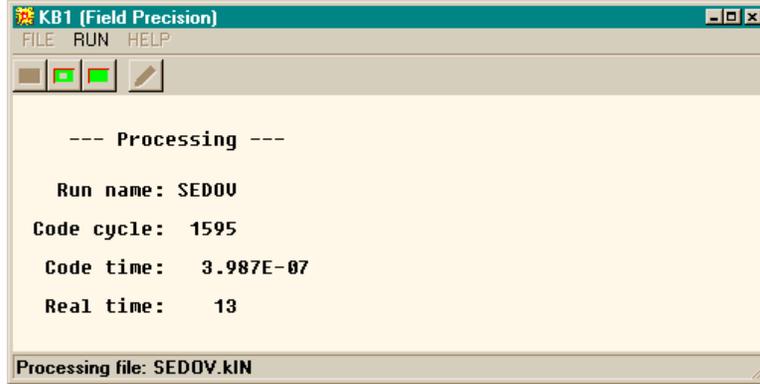
### **KB1V**

Interactive graphical post-processor for data files (spatial variations of quantities at specified times)

### **Probe**

Standard Field Precision plotting and oscilloscope utility, used to inspect and to plot **KB1** history data.

Table 1 summarizes physical units used in **KB1** and **KB1V**. In the remainder of this section we shall walk through an example to introduce you to the program.



**Figure 1. KB1**

<b>Table 1. Physical units for the KB1 programs</b>		
<b>Quantity</b>	<b>KB1: internal operations, data and history files</b>	<b>KB1V and PROBE1</b>
Position	m	cm
Pressure	Pa	GPa
Temperature	°K	°K
Internal energy	J/kg	MJ/kg
Velocity	m/s	km/s

To illustrate the sequence of operations in a **KB1** run, we shall follow the example SEDOV, described in *KB Methods and Tutorials*. The first step is to collect data that will be used by the program. Ensure that the files SEDOV.KIN and IGAS002.KBT are located in the working directory. In the example we assume the standard setup where programs are located in \KB and the working directory is \KB\BUFFER. Run KB.EXE, the **KB** program launcher. If necessary, set the program directory to \KB) and the data directory to \KB\BUFFER. Click the button next to **KB1** to run the program. You should see the display of Figure 1. Pick the *Start run* tool or click on the *Run/Start run* menu options. In the dialog, pick the file SEDOV.KIN. The program loads data and starts the calculation. It should take about 1-2 minutes to finish.

The program uses and creates different types of files. Go to the *File* menu in **KB1**, choose *Edit KIN file*, and then choose SEDOV.KIN in

the dialog. The program runs a full-featured Windows editor and loads SEDOV.KIN. The file (shown below) is an ASCII script that performs several functions

# Sets control parameters such as the time step or system symmetry.

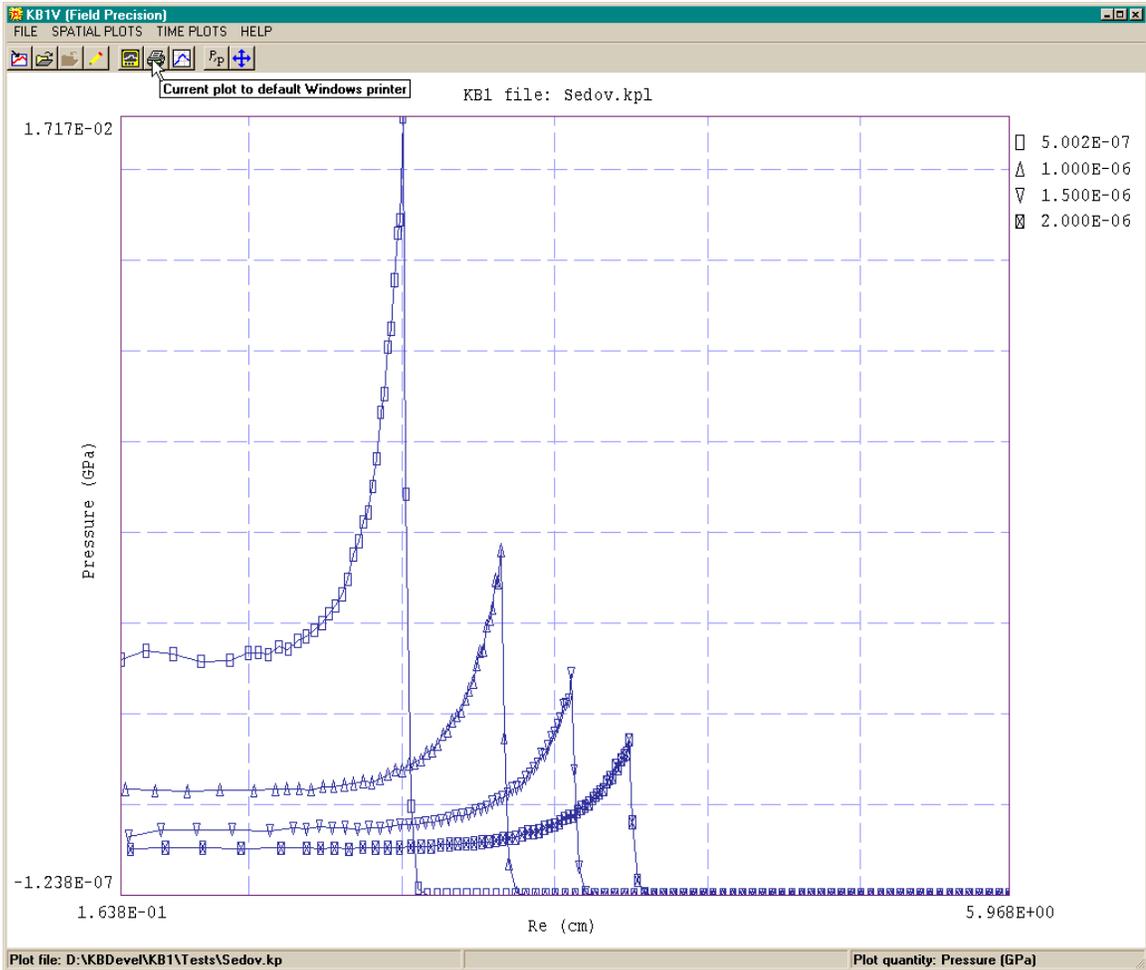
# Defines the geometry by dividing the solution space up into *regions* with common material characteristics

# Specifies material models to be associated with the regions

# Sets initial conditions and driving terms

# Controls diagnostic output

```
* --- CONTROL ---
TITLE: Sedov spherical blast wave
GEOMETRY: SPHERE
INTERPORDER: FIRST
DT: 0.25E-9
DTCHANGE 0.5E-6 2.5E-9
TMAX: 5.01E-6
* --- MATERIALS ---
MATERIAL 1 KBT IGAS0002
* MATERIAL 1 GAMMA 1.66667
* --- GEOMETRY ---
REGION 1 1 0.0000 0.002 5
REGION 2 1 0.0020 0.060 90
* --- REGIONS ---
HYDINIT 1 0.1000 2.135E7
* HYDINIT 1 0.1000 1.84E11
VISCOSITY 1 2.50
HYDINIT 2 0.1000 0.001
* HYDINIT 2 0.1000 0.000
VISCOSITY 2 2.50
* --- DIAGNOSTICS ---
DTIME: 0.25E-6
PROBESTEP: 25
SETPROBE: 2
SETPROBE: 70
*
ENDFILE
```



**Figure 3. KB1V program display**

Close the edit window and choose the command *Edit file* from the *File* menu. In the dialog, pick IGAS0002.KBT. This file provides the equation-of-state information required to solve the hydrodynamic equations. Note that the file is an ASCII format. You can incorporate the data into your own programs. The manual *KBT Tables* gives detailed information.

An inspection of the contents of \KB\BUFFER shows that the program has created four output files: SEDOV.KLS, SEDOV.P01, SEDOV.P02 and SEDOV.KPL. Open SEDOV.KLS using the *Edit KLS file* command. This listing file contains run information and extensive

quantitative data. The bulk of the file consists of *data dumps*, listings of hydrodynamic quantities at all points in space at times specified by the input script file. The file SEDOV.KPL contains the data dump information in binary format. This file is used by the **KB1V** program for interactive

analyses and plots. Finally, check SEDOV . 001 using the *Edit file* command. This file contains history information, a record of hydrodynamic quantities as a function of time in a given element. A **KB1** run may create several history files for different elements. You can use this information directly in your own analyses or use the **Probe** program to create plots and check values.

Next, run **Probe** from the KB program launcher. Click on *Load probe file* and pick SEDOV . P02 in the dialog. The message box shows information on the file. Click *OK* to continue. Under *Plot parameters*, click on *Pick plotted quantity* and choose *density*. You should see the plot of Figure 2. Under *Plot functions*, choose *Oscilloscope mode*. By moving the mouse, you can measure points on the plot. Click the right mouse button to return to the program. The **Probe** manual gives a detailed description of the program capabilities.

Finally, run **KB1V** from the **KB** program launcher. Under *File* click on *Load plot file*. In the dialog, choose SEDOV . KPL and click *OK*. Under *Spatial plots*, choose *Set included plots*. The listing in the dialog shows that dumps were recorded at intervals of 0.25 ?s. You can turn plots *ON* and *OFF* by clicking on them. Activate the plots for 0.50 ?s, 1.00 ?s, 1.50 ?s and 2.00 ?s and click on *OK*. Under *Spatial plots*, choose *Screen plot*. You should see the display of Figure 3. The **KB1V** manual describes other features of the program.

## Section 2. Running KB1

The program KB1 . EXE can run in two modes:

# Interactive Windows program

# Autonomous program run from the Command Prompt, usually under batch file control

We shall first discuss the interactive mode. There are several ways to start KB1 . EXE: click the entry on the **KB** program launcher, set up a shortcut on the Windows Desktop or Start menu, double-click on the program in Windows Explorer, or use the Run option on the Start menu. In all cases, the window shown in Figure 1 appears. Entries in the pulldown menus have the following functions.

### **Edit KIN file, Edit KLS file, Edit file**

The program has a built-in Windows editor that can be used to inspect input and output ASCII files. The difference between the

commands is the choice of files displayed in the opening dialog. Files with suffix `KIN` are **KB1** input files. Files with suffix `KLS` are output listing files which contain run information and data dumps. Changing directories in the dialog does not change the working directory of the program. Close the editor to continue **KB1** operation.

### **Start run**

This command initiates a **KB1** calculation. The dialog displays all files in the current working directory with the suffix `KIN`. Pick a file and click *OK*. The run proceeds if the command script is syntactically correct and the required input data files are present. The data window (Figure 1) shows the run identifier, the number of cycles (time steps) processed, the simulation time, and the actual time elapsed.

### **Pause run**

**KB1** is a number cruncher that can occupy the full resources of your computer. As a result other programs may run slowly. If you need to perform a critical task in the middle of a **KB1** run, you can pause the program. Click the *Resume* button to continue with no loss of data.

### **Stop run**

The *Stop run* command is useful if a run is taking too long or you want to check output from the first part of an extended run. This command prompts for confirmation, closes all files and stops the program. Previously recorded data dumps and probe information will be available.

### **KB1 manual**

Displays a text version of this manual. The file `KB1 .HTM` must be in the same directory as `KB1 .EXE`.

You can run **KB1** as an autonomous program from the Command Prompt by supplying the prefix of the input file as a command line parameter. For example, suppose that the program is in the directory `\KB` and the input file `SEDOV .KIN` is in the directory `\KB\BUFFER`.

Type

```
\KB\KB \KB\BUFFER\SEDOV <ENTER>
```

This capability is useful if you want to prepare a batch file for a set of extended runs. For example, the following file would perform five calculations with no operation intervention:

```
REM Cylindrical detonators
REM Variation of initiator radius
\KB\KB \KBDATA\CYLDET\CYLDET01
\KB\KB \KBDATA\CYLDET\CYLDET02
\KB\KB \KBDATA\CYLDET\CYLDET03
\KB\KB \KBDATA\CYLDET\CYLDET04
\KB\KB \KBDATA\CYLDET\CYLDET05
REM Job completed
```

### Section 3. Geometry and mesh conventions

The basis of the finite-element approach is the division of the solution volume into many small pieces or *elements*. In the limit where the element size is small compared to the scale length for significant variations, the continuous equations of hydrodynamics reduce to a set of coupled linear equations that can be solved easily on a computer. The division is carried out so that each element has a unique material identity. In other words, the boundaries between elements are chosen so that they lie along the boundaries between different types of materials in the solution space. Another important term is *vertex*. In the one-dimensional solutions of **KB1**, a vertex is the boundary between elements. (In the two-dimensional solutions of **KB2** vertices are the intersection points of element boundaries.) To solve the hydrodynamic equations it is convenient to associate some quantities with elements (usually material properties and hydrodynamic state) and others with vertices (Table 2).

Figure 4 shows the division of a volume into elements for a **KB1** solution. The program handles systems with planar, cylindrical and spherical symmetry. In the three modes elements represent slabs, cylindrical shells, or thin spherical shells. In planar geometry slabs move in  $x$  and have infinite lengths in  $y$  and  $z$ . In cylindrical geometry shells move in  $r$ , are symmetry in  $\theta$ , and have infinite length in  $z$ . In spherical geometry shells move in  $r$  and have symmetry in  $\theta$  and  $\phi$ . The thickness of elements need not be uniform in **KB1** solutions. Elements have indices in the range 1 to  $N_e$ , while vertices are labeled from 0 to

$N_e$ . With this convention Element  $n$  is bounded by vertices  $n-1$  and  $n$ . There are two other terms that apply to the organization of elements: *Material* and *Region*. The *material model* specifies the equation-of-state applied to an element. For given values of density and internal energy, the material model gives the corresponding pressure (and temperature for KBT materials). Sections 7 and 8 describe the available options. Up to 20 material models can be defined for a **KB1** run. The quantity *MatNo* (an integer in the range 1 to 20) specifies the material model associated with the element. Elements are grouped into *Regions* (layers) that share common properties. Up to 127 Regions can be defined in a **KB1** run. All elements in a Region have the same material number. The number is assigned to the Region, and then all included elements are marked accordingly. Two separated or adjacent regions may use the same material model but may have different initial properties (i.e., hydrodynamic state, velocity, element thickness, artificial viscosity coefficient,...). Table 2 lists the quantities used in **KB1** to characterize vertices, elements, materials and regions. The quantities are divided into three classes:

*Class 1.* Can be set directly in the input script file (**bold**)

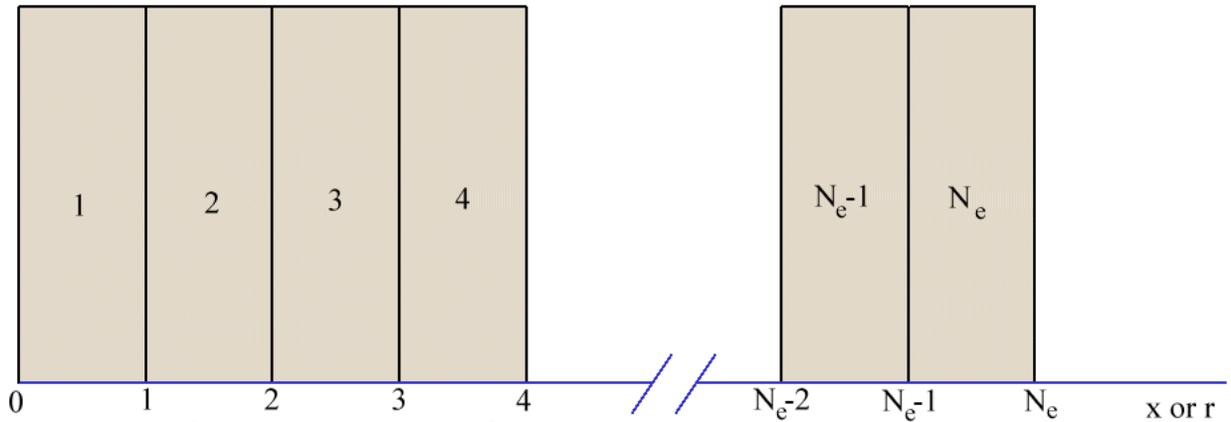
*Class 2.* Calculated by the program and recorded int the listing file (standard)

*Class 3.* Calculated by the program and used internatilly (italic)

<b>Table 2. Quantities used in KB1</b>	
<b>Vertex quantities</b>	
x[n]	Boundary position between Elements $n$ and $n+1$ (m)
v[n]	Velocity of Vertex $n$ (m/s)
<b>Element quantities</b>	
<b>MatNo</b> [n]	Material number
<b>RegNo</b> [n]	Region number
Rho[n]	Density (kg/m <sup>3</sup> )

<b>Table 2. Quantities used in KB1</b>	
<i>Press</i> [n]	Pressure (Pa)
<i>Temp</i> [n]	Temperature (°k), applies only to KBT materials
<i>Epsi</i> [n]	Internal energy of Element n (J/kg)
<i>Mass</i> [n]	Invariant mass (kg), internal program use
<i>WVis</i> [n]	Artificial viscosity (Pa), internal program use
<i>Squash</i> [n]	Compression flag, applies only to Void materials
<i>Detonated</i> [n]	Detonation state, applies only to Explosive materials
<b>Material quantities</b>	
<b>Type</b> [i]	KBT, Void, ShockEOS, Gamma, Explosive
<b>MatPrefix</b> [i]	KBT table name such as ALUM3715, (KBT)
<b>DBounce</b> [i]	Minimum element thickness (m), (Void)
<b>C0</b> [i], <b>S1</b> [i], <b>S2</b> [i]	Hugoniot EOS parameters (ShockEOS, Explosive)
<b>PInit</b> [i]	Ignition pressure (Pa), (Explosive)
<b>Q</b> [i]	Specific energy of detonation (J/kg), (Explosive)
<b>Initiated</b> [i]	Detonation status at t = 0, (Explosive)
<b>Region properties</b>	
<b>NMat</b>	Material number
<b>RI</b> [j]	Initial inner radius (m)
<b>RO</b> [j]	Initial outer radius (m)
<b>NElem</b> [j]	Number of elements
<b>VInit</b> [j]	Initial velocity (m/s)
<b>RhoInit</b> [j]	Initial density (kg/m <sup>3</sup> )
<b>PressInit</b> [j]	Initial pressure (Pa)
<b>EpsiInit</b> [j]	Initial internal energy (J/kg)
<b>TempInit</b> [j]	Initial temperature (°K), applies only for KBT materials

**Table 2. Quantities used in KB1**



**Figure 4.** Index conventions for **KB1**. Element (top) and vertex (bottom) indices.

**CVis[j]**

Artificial velocity coefficient

## Section 4. Structure of the input script file

**KB1** may use and create several types of files. Table 3 summarizes the function and format of different files. In this section and the following ones, we shall concentrate on the preparation of input files to control the run and to supply data. The input script is the fundamental control file required for all **KB1** runs. It is an ASCII text file that consists of a list of commands with parameters, similar to a BASIC interpreter program. The main difference is that the commands can appear in any order - the program sorts out the logic before proceeding to execution. The command file must have a name of the form `FPrefix.KIN`, where `FPrefix` is any valid DOS name from 1 to 8 characters in length. Long file names are not allowed to ensure that the programs will operate correctly on FAT disk partitions under batch file control. The script and other input files for equation-of-state data and pressure waveform information should be in the same directory. In the batch file mode the program is run with a command like

ProgDir\KB1 \DataDir\FPrefix <Enter>

where ProgDir represents the path to the executable files and DataDir is the path to the input files. In the interactive mode, you can use standard Windows file selection dialogs to change directories and pick a script.

<b>Table 3. KB1 input and output files</b>			
<b>Name form</b>	<b>I/O</b>	<b>Function</b>	<b>Format</b>
FPrefix.KIN	Input	Script controls operation of the program	ASCII
PressName	Input	Table of waveform values for an applied pressure	ASCII
KBTName.KBT	Input	Equation-of-state table for KBT material	ASCII
FPrefix.KLS	Output	Listing file with extensive run information and data	ASCII
FPrefix.KPL	Output	Plot file of data dumps for <b>KB1V</b> analyses	Binary
FPrefix.P01	Output	One or more probe files, temporal information for a given element	ASCII

Section 1 shows a sample input script. The file may contain command, comment and blank lines. A command line consists of a keyword and one or more parameters. Commands may appear in any order because **KB1** sorts and processes information after all commands are read. You can enter commands and parameters in upper or lower case. Comment lines must begin with '\* ' (asterisk-space). Commands and parameters can be indented. The program ignores spaces and other delimiters. The following are valid delimiters.

Space: ' '  
Comma: ','  
Colon: ':'  
Equal sign: '='  
Tab character  
Left parenthesis: '('

Right parenthesis: ‘)’

Command input terminates with the *EndFile* statement. The command signals that input information is complete and **KB1** should begin execution. You can enter descriptive text in any format after *EndFile*.

Commands fall into seven categories:

### **Controls**

Set general properties like the maximum run time.

### **Materials**

Define material models to be associated with regions and elements

### **Geometry**

Define the dimensions and divisions of regions

### **Regions**

Set region properties, such as the initial velocity or hydrodynamic state

### **Diagnostics**

Control how much information is written to the listing, plot and probe files

The following sections cover input script commands in detail. *Control* commands are reviewed in Section 5. Section 6 discusses the preparation of files to define applied pressure waveforms. Commands to define material properties are presented in Section 7, while Section 8 concentrates on explosive materials. *Geometry* commands are covered in Section 9 and *Region* commands in Section 10. Section 11 reviews *Diagnostic* commands. Finally, Section 12 is a useful reference chart of available commands.

## **Section 5. Control commands**

Control commands can appear anywhere in the script file, although it is good practice to group them at the beginning. This section shows each command in symbolic form and also as it might appear in the script.

The first two commands are required.

**GEOMETRY [RECT, CYLIN, SPHERE]  
GEOMETRY CYLIN**

**Comments:** Sets the symmetry of the one-dimensional calculation.

**Parameters:** A single parameter, either the string *Rect*, *Cylin* or *Sphere*.

**TMAX TMax  
TMAX 5.75E-6**

**Comments:** Sets the maximum run time.

**Parameters:** A single real-number quantity, the maximum run time in seconds.

**TITLE Run title string  
TITLE Ignition of TNT rod (small ignitor)**

**Comments:** The title string is useful for archiving a run and appears in plots created by **KB1V**.

**Parameters:** A string of up to 60 characters

**SAFETY Safety  
SAFETY 5.0**

**Comments:** **KB1** can pick a time step automatically for simple runs where there are no abrupt changes in quantities (i.e., collisions between regions). This command sets a safety factor for estimating a stable time step from the instantaneous element properties. Higher numbers are more conservative. Lower numbers reduce the run time at the risk of a numerical instability.

**Parameters:** A single real-number quantity for the *Safety* variable. The default value is 10.0.

**NCHECK NCheck  
NCHECK 50**

**Comments:** Sets the number of program steps between recomputations under automatic time step control.

**Parameters:** Single integer parameter, *NCheck*. The default value

is 5.

### **DT Dt**

#### **DT 1.0E-09**

**Comments:** This command sets the value of the initial time step and disables automatic time step selection. You may need to experiment to find a value of  $Dt$  that gives numerical stability with an acceptable run time.

**Parameter:** A single real-number parameter,  $Dt$  in seconds.

### **DTCHANGE TChange DtNew**

#### **DTCHANGE 5.0E-06 2.5E-09**

**Comments:** The time step  $Dt$  changes to the value  $DtNew$  at time **TChange**. This command allows you to fine-tune the program for difficult simulations. For example, if a slab accelerated smoothly and then abruptly collided with another slab, you could initially use a long time step and then shorten it just before the collision. Another application is to lower  $Dt$  to handle the convergence of a shock on the axis of a cylindrical or spherical system. The *DTChange* command over-rides values set by the  $Dt$  command or automatic step selection.

**Parameters:** Two real-number parameters, both in seconds. The first (*TChange*) is the approximate time at which the change should occur, and the second (*DtNew*) is the new value of the time step.

### **INTERPORDER [First, Second]**

#### **INTERPORDER First**

**Comments:** This command controls the type of interpolations performed for KBT materials. First order interpolations are generally sufficient for shock solutions at high temperature or pressure. Second order interpolations increase the run time significantly, but may be necessary to achieve accuracy in simulations where there are small changes of density or pressure. This command has no effect for Void, ShockEOS, Gamma and Explosive materials.

**Parameters:** A single string, either *First* or *Second*. The default is *First*.

## **ERRSTATUS [All,Partial,None]**

### **ERRSTATUS Partial**

**Comments:** **KB1** may perform millions of interpolations in runs with KBT materials. The operations are of the form  $T(?U)$  and  $P(?T)$ . This command controls how the code responds when it detects an interpolation error. Errors in  $T(?U)$  may occur if the material is near ambient temperature and if the KBT table has widely-spaced values at low temperature. In this case, there is little error introduced if we simply set the temperature equal to the initial value for the region if an error is detected. In the *Partial* option, the program ignores errors of the type  $T(?U)$ , but terminates following an error in  $P(?T)$ . In the *All* option, the program stops with an error message for any interpolation error. In the *None* option, the program ignores all interpolation errors. This option is useful mainly for debugging. This command has no effect for Void, ShockEOS, Gamma and Explosive materials.

**Parameters:** A single string, either *All*, *Partial* or *None*. The default is *Partial*.

## **FIXBOUND [Inside,Outside,Both,None]**

### **FIXBOUND Inside**

**Comments:** This command clamps the position of a vertex so that it does not move during the run. For example, we could apply the condition to a boundary in a planar simulation to model one half of a symmetric system. Another application is representation of an ideal containment shell for a detonating gas. The string parameter options have the following meanings. *Inside*: Clamp the solution boundary at  $x_{\min}$  or  $r_{\min}$ . *Outside*: Clamp the boundary at  $x_{\max}$  or  $r_{\max}$ . *Both*: Clamp both boundaries.

**Parameters:** A single string, either *Inside*, *Outside*, *Both* or *None*. The default is *None*.

## **Section 6. Applied pressure waveforms**

There are three ways to drive shock hydrodynamic solutions in **KB1**: setting initial velocities between colliding regions, detonating explosives, and applying a time-dependent drive pressure. This section explains how to define a time-dependent pressure waveform that acts at one or both of the solution boundaries. The controlling command has the following form:

**PRESSURE TabName [Inside,Outside]**  
**PRESSURE LinDrive.PRS Outside**

**Comments:** Two *Pressure* commands can appear in the input script file to define applied pressure waveforms at the inside and outside boundaries.

**Parameters:** There are two string parameters. The first gives the full name of a tabular function file in the current data directory. The file format is described below. The options for the second parameter have the following meanings. *Inside:* Apply the pressure at the solution boundary at  $x_{\min}$  or  $r_{\min}$ . *Outside:* Apply the pressure at the boundary at  $x_{\max}$  or  $r_{\max}$ .

Arbitrary time variations of pressure can specified with tabular functions that follow the same format in all Field Precision programs. Tabular function are ASCII files that contain up to 256 data lines. Each data line contains two real numbers separated by spaces or other valid delimiters (see Sect. 4). The first number in each line is the time (in s) and the second number is a corresponding value of the pressure (in Pa). The time interval need not be uniform. For example, you may want to cluster values near a time when there is a sharp transition of pressure. The *EndFile* command marks the end of data entry. Tables can also contain comment lines starting with '\*' (asterisk-space) and blank lines. The following illustrates a normalized pressure table:

```
* Normalized smooth step function
* Rise to 1.0 at time 1.0
* =====
      0.00000      0.00000
      0.05000      0.00616
      0.10000      0.02447
      0.15000      0.05450
      0.20000      0.09549
      0.25000      0.14645
      0.30000      0.20611
      0.35000      0.27300
      0.40000      0.34549
      0.45000      0.42178
      0.50000      0.50000
      0.55000      0.57822
      0.60000      0.65451
      0.65000      0.72700
      0.70000      0.79389
      0.75000      0.85355
      0.80000      0.90451
      0.85000      0.94550
      0.90000      0.97553
```

0.95000	0.99384
1.00000	1.00000
1.05000	1.00000
1.10000	1.00000
1.50000	1.00000
2.00000	1.00000
5.00000	1.00000
10.00000	1.00000
20.00000	1.00000
50.00000	1.00000
100.00000	1.00000

ENDFILE

**KB1** finds interpolated values using a cubic spline interpolation of the data. This method is very flexible, but may give unexpected results for noisy or inaccurate data. For this reason, the program writes a test series of 50 interpolations uniformly spaced over the time interval 0.0 to *TMax* in the listing file.

An alternate form of the *Pressure* command allows you to add multiplying factors to normalized tables.

**PRESSURE TabName [Inside,Outside] TMult PMult  
PRESSURE LinDrive.PRS Outside 1.0E-06 25.0E09**

**Comments:** With this command you can maintain a library of normalized waveforms, using multiplication factors to apply them to different simulations. Note that the test interpolation listing shows values as they will be used in the program.

**Parameters:** There are two string and two real-number parameters. The first string gives the full name of a tabular function file in the current data directory. The options for the second string parameter have the following meanings. *Inside*: Apply the pressure at the solution boundary at  $x_{\min}$  or  $r_{\min}$ . *Outside*: Apply the pressure at the boundary at  $x_{\max}$  or  $r_{\max}$ . The first real-number parameter is a multiplication factor for time values. The values used in the program are the table values multiplied by *TMult*. The second real-number parameter is a multiplication factor for the pressure values.

## Section 7. Standard material properties

The manual *Physics of the KB Codes* discusses the material models used in **KB1** and **KB2**. It is important to note that the programs are useful for shock phenomena at very high temperatures and pressures.

The current versions do include support for elastic or inelastic material strength and do not include effects of energy transport by thermal conduction or radiation. The programs use five models for the hydrodynamic properties of materials. They differ in how element pressure is inferred as a function of density and internal energy.

### **KBT Materials**

For these materials the programs use the KB tables to find both pressure and temperature of the material. The tables are based on the Sesame tables distributed by Los Alamos National Laboratory (see the manual *KB Tables* for a complete description). The *KBT* model has several advantages compared to the *Gamma* and *ShockEOS* models:

# It includes effects of phase changes in materials

# It gives information on temperature as well as density, pressure and specific energy

# It correctly represents both shock compression and subsequent expansion of materials

The disadvantage is that the interpolations require considerably more run time than the simple formula evaluation of the *ShockEOS* and *Gamma* models.

### **Shock EOS materials**

The shock equation-of-state model gives useful approximations for materials that are not covered by the KB tables. The model yields changes in pressure as a function of density and internal energy with the assumption that material states lie on the shock Hugoniot curve. The manual *Shock EOS Tables* gives detailed information. The model is applicable to materials undergoing shock compression, but it does not provide an accurate representation of subsequent expansion.

### **Gamma law materials**

Many gases follow the law  $p = (\gamma - 1) \rho_o U$  over a wide range of density and pressure. If applicable, the model has two advantages

over interpolations on the KB tables.

# There are fewer numerical operations, leading to shorter run times.

# There is less chance of an interpolation error under extreme conditions.

The gamma law model is useful to avoid numerical instabilities in treating materials with very high density compressions, such as a gas entrapped between colliding solid regions.

### **Explosive materials**

Explosive materials use both the *ShockEOS* and *Gamma* models. The shock equation-of-state applies to the solid or liquid materials before detonation. An element of explosive material detonates if it is subjected to a shock that raises the pressure above a critical value. At detonation, the density is unchanged while the internal energy increases by  $Q$  (the specific energy of detonation). For subsequent times the increased pressure is calculated from the gamma law model.

### **Void Materials**

A void is a single-element region that occupies the space between initially-separated material regions. Voids are included because the finite-element method requires continuity of the solution volume. They have a simple equation of state. The pressure is zero if the void element width is larger than a scale distance *DBounce*. If compression reduces void width to *DBounce*, the element becomes incompressible and assumes a pressure equal to the average of values in adjacent material elements.

The following commands set the properties of a numbered material model. You can define up to 20 material models. A model can be associated with several regions.

**MATERIAL MatNo KBT TabName**  
**MATERIAL 5 KBT ALUM3715**

**Comments:** Loads the KB table TabName .KBT and assigns in as

material model *MatNo*. The table must be in the current directory. The initial hydrodynamic state of the material can be set individually in different regions.

**Parameters:** The parameter *MatNo* is an integer in the range 1-20 to identify the material model. The string *KBT* designates that the equation-of-state will be determined from interpolations on a KB table. The string *TabName* (1-8 characters) is the prefix of the KB table.

### **MATERIAL MatNo SHOCK C0 S1 S2**

#### **MATERIAL 4 SHOCK 2.985E3 0.5148 2.814E-4**

**Comments:** Defines material *MatNo* as a *ShockEOS* material and sets the relationship between  $u_s$  (shock velocity) and  $u_p$  (particle velocity) as defined in the manual *ShockEOS Tables*,

$$u_s = C_0 + S_1 u_p + S_2 u_p^2.$$

The initial density can be set individually in different regions. The initial pressure equals zero.

**Parameters:** The parameter *MatNo* is an integer in the range 1-20 to identify the material model. The string *Shock* designates that the equation-of-state will be determined from the Shock EOS model. The three real-number parameters are  $C_0$  (m/s),  $S_1$  and  $S_2$  (s/m).

### **MATERIAL MatNo GAMMA Gamma**

#### **MATERIAL 3 GAMMA 2.53**

**Comments:** Defines material *MatNo* as an gamma-law material where pressure  $p$  is related to internal energy  $U$  by

$$p = (\gamma - 1)\rho_o U.$$

The initial density and internal energy of the material in a particular region can be set with *Region* commands of Sect. 10.

**Parameters:** The parameter *MatNo* is an integer in the range 1-20 to identify the material model. The string *Gamma* designates that the equation-of-state will be determined from the gamma=law model. The real number parameter is the  $\gamma$  value.

### **MATERIAL MatNo VOID [DBounce]**

#### **MATERIAL 6 VOID 0.005**

**Comments:** Void regions are placed between initially separated

material regions. They provide logically continuity of the finite-element mesh. A void region must consist of a single element. The void has no effect on the hydrodynamic problem when its width exceeds the quantity *DBounce*. If the width decreases to *DBounce*, the element becomes incompressible and assumes a pressure to provide force continuity between the adjacent material elements. **Parameters:** The optional real-number parameter *DBounce* (m) is associated with material number *MatNo*. If the value is not specified, the void material is assigned a default value equal to the width of the smallest element in the simulation space.

## Section 8. Explosive material properties

This section covers the form of the *Material* command that defines the **KB** parametric model for explosives (discussed in Sect. 4 of the manual *Physics of the KB Codes*.) A typical simulation starts with the creation of a shock wave in the explosive by an initiator. The shock could be produced by collisions between regions, a rapid local heating, or the influence of an adjacent explosive. During the ignition phase it is sufficient to treat the solid explosive with the shock EOS model (see Sect. 3 of *Physics of the KB Codes*). If the shock pressure in an element exceeds a minimum value, *PInit*, the element material is undergoes a rapid chemical change. The simplifying assumption in **KB** is that the internal energy of the element instantaneously increases by  $Q$  (the specific heat of explosion) and that the material changes to a gaseous state. The detonation products are described by an ideal  $\gamma$ -law equation-of-state. Although the model is relatively simple, it has two advantages: 1) the required material data are generally available and 2) the results are in good agreement with theoretical and experimental values for detonation front velocities.

Note that results are not sensitive to the exact choice of the shock EOS parameters  $C_0$ ,  $S_1$  and  $S_2$ . If you do not have values for a specific material or initial density, you can usually get satisfactory results with parameters of a material with similar properties. The quantity *PInit* is not an absolute property of an explosive, but rather a parameter that can be adjusted to model different types of behavior. Values just below the Chapman-Jouguet pressure give well-defined detonation fronts. Lower values correspond to more sensitive explosives, such as inhomogeneous materials with potential hot spots.

**MATERIAL MatNo EXPL C0 S1 S2 PInit Q Gamma [INIT]  
MATERIAL 5 EXPL 2461.0 3.517 -8.810E-04 2.0E10  
5.183E+06 2.63**

**Comments:** The initial density and internal energy of the material in a particular region can be set with *Region* commands of Sect. 10.

**Parameters:** The parameter *MatNo* is an integer in the range 1-20 to identify the material model. The string *Expl* designates that the equation-of-state will be determined from the explosives model. There are six real-number parameters. The quantities  $C_0$  (m/s),  $S_1$  and  $S_2$  (s/m) define the shock equation-of-state for the undetonated material. The quantity *PInit* (Pa) is a threshold pressure for detonation,  $Q$  (J/kg) is the specific energy released in the explosion, and  $\gamma$  defines the equation-of-state for the gaseous detonation products. If the string parameter *Init* is included, the material will detonate at a  $t = 0.0$ . This option is useful for checking the safety of explosive configurations. Self-ignited material should not cause a detonation wave.

## Section 9. Defining the system geometry

Defining geometry in a one-dimensional simulation is easy. **KB1** requires only a single command. Depending on the setting of the *Geometry* command, the simulation volume can have three different forms:

**RECT:** A slab of material between the limits  $x_{\min}$  and  $x_{\max}$  with infinite extent in  $y$  and  $z$ .

**CYLIN:** A cylindrical shell of material between limits  $r_{\min}$  and  $r_{\max}$  with infinite extent in  $z$ . Note that **KB1** can handle solid cylinders with  $r_{\min} = 0$ .

**SPHERE:** A spherically symmetric shell of material between the limits  $r_{\min}$  and  $r_{\max}$ . Note that **KB1** can handle a solid sphere with  $r_{\min} = 0$ .

The solution volume is divided into *regions* where elements have like properties. For example, the elements in two adjacent regions may consist of different materials. Another possibility is that they consist of the same material but have different initial conditions (*i.e.*, directed

velocity, density, ...). You can also vary the element size by dividing a material volume into several regions. For example, to model a converging cylindrical shock it is often necessary to use smaller elements near the axis to preserve accuracy.

There are two rules for defining a geometry in **KB1**:

# Regions must be defined in order from left to right ( $x_{\min}$  to  $x_{\max}$ ) in planar simulations. In cylindrical or spherical systems, regions must be defined from inside to outside ( $r_{\min}$  to  $r_{\max}$ ).

# The regions must fill the space between  $x_{\min}$  and  $x_{\max}$  ( $r_{\min}$  and  $r_{\max}$ ). Any initially empty spaces must be filled with single-element voids.

The following command sets the properties of regions:

**REGION RegNo MatNo Ri Ro NElem**  
**REGION 5 1 0.000 0.002 50**

**Parameters:** The integer quantity *RegNo* is the region number, The maximum allowed value is 127. Regions must be numbered sequentially (starting at *RegNo* = 1) as they appear in the file. The integer parameter *MatNo* is the number of the material model associated with the region. The code issues an error message if a material number has not been defined. The two real number parameters *Ri* and *Ro* are the left and right (inner and outer) boundaries of the region. Enter the dimensions in meters. Finally the integer number *NElem* is the number of uniform-width elements in the region.

The following illustrates a valid geometry definition for a cylindrical shell:

```
REGION 1 1 0.005 0.010 20
REGION 2 1 0.010 0.050 20
REGION 3 2 0.050 0.150 50
```

Note that the region numbers are in order and that the spatial

dimensions proceed from left (inside) to right (outside). The start radius of a region equals the end radius of the previous region. Regions 1 and 2 consist of the same material, but the elements in Region 1 are smaller. Region 3 consists of a different material. **KB1** uses dynamic memory allocation, so the total number of elements is limited only by the installed memory of the computer.

## Section 10. Setting region properties

You can set initial properties of individual regions with the following commands.

### **VELINIT RegNo Vinit [WEIGHTED]**

#### **VELINIT 5 10.0E3**

**Comments:** The default initial region velocity is zero. The optional string *Weighted* applies only to cylindrical and spherical geometries. If it does not appear, all elements in the region have the same initial velocity. In this case, the relative size of elements in a cylindrical or spherical compression changes more rapidly on the inside of a layer and the density becomes non-uniform. When the *Weighted* option appears, the element velocity is assigned as  $v_e(\mathbf{r}) = v_{init} r_i / r$  for cylindrical systems or  $v_e(\mathbf{r}) = v_{init} r_i^2 / r^2$  in spherical geometries. The scaling ensures uniform density for converging or diverging shells. Note that the velocity *Vinit* occurs at the inside of the region.

**Parameters:** The integer parameter *RegNo* is the region number. The real-number parameter *Vinit* is the initial velocity in m/s. The string parameter *Weighted* is optional.

### **HYDINIT RegNo Rho Temperature**

#### **HYDINIT 4 1753.0 298.0**

**Comments:** This form of the *HydInit* command applies to KBT materials. You can check the validity of the initial internal energy and pressure interpolations by checking the list in the KLS file. The *HydInit* command has no effect on a void material.

**Parameters:** The integer parameter *RegNo* is the region number. The two real-number parameters are the initial density (in kg/m<sup>3</sup>) and temperature (in °K).

**HYDINIT RegNo Rho U**  
**HYDINIT 6 860.0 0.0**

**Comments:** This form of the *HydInit* command applies to Gamma, ShockEOS and Explosive materials. The *HydInit* command has no effect on a void material.

**Parameters:** The integer parameter *RegNo* is the region number. The two real-number parameters are the initial density (in kg/m<sup>3</sup>) and internal energy (in J/kg). Normally the internal energy equals zero.

**VISCOSITY RegNo Viscosity**  
**VISCOSITY 5 10.0**

**Comments:** In hydrodynamic shock simulations, it is essential to include artificial viscosity forces to damp short-wavelength disturbances. You can assign relative viscosity terms individually to each region. If the value is too low spurious oscillations will occur near the shock front. If the value is too high the shock will spread in space. Ideally, the change in properties across the shock should extend over a few elements.

**Parameters:** The integer parameter *RegNo* is the region number. The parameter *Viscosity* is a dimensionless real number. The default value is *Viscosity* = 5.0.

## Section 11. Diagnostics

The commands of this section control the data files created by **KB1**. There are two types of information available from an initial-value simulations: 1) records of quantities over the full solution volume at specified times (spatial data dumps) and 2) records of quantities at all time steps at specified positions (probes). Three commands control the creation of spatial dumps.

**DIAGDT DTime**  
**DIAGDT 5.0E-6**

**Comments:** Write a data dump at approximately uniform time intervals. Because the time step may vary, the record may not occur at an exactly uniform interval. The convention is that **KB1** writes the spatial information at or immediately after the specified time.

**Parameters:** The real-number parameter *DTime* is the time interval in seconds.

**DIAGTIME TDiag**  
**DIAGTIME 7.333E-8**

**Comments:** You can set up to 50 specific times to make data dumps. For example, you may want to create a detailed set of views at the collision time of two regions. This command can work in conjunction with the *DiagDt* command. The data dump occurs at or immediately after the specified time. The *DiagTime* commands must appear in the input script in chronological order (*i.e.*, ordered from the earliest to the latest data dump).

**Parameters:** Enter the time *TDiag* in seconds.

**DIAGSTEP NDiag**  
**DIAGSTEP 200**

**Comments:** When this command is issued, **KB1** makes a data at intervals separated by a given number of time steps. This command works in conjunction with the *DiagDt* and *DiagTime* commands. Be careful entering dump information, or you may fill a hard disk with data.

**Parameters:** The integer parameter *NDiag* is the number of time steps between data dumps.

The final two commands control the placement of probes in the solution volume. **KB1** can create from 1 to 12 probe files with names of the form *RunName.P01*, *RunName.P02*, .... Information in the files can be plotted or analyzed with the **Probe** program.

**SETPROBE NPElem**

## SETPROBE 58

**Comments:** Probes are located at the center of an element. The output files contain the corresponding element quantities (density, pressure, ...). Vertex quantities (position, velocity) are calculated at the element center-of-mass.

**Parameters:** The integer parameter *NElem* is number of the element where the probe should be located.

## PROBESTEP NStepProbe

### PROBESTEP 5

**Comments:** Simulations with short time step may run for thousands of time steps. In this case, there is no need to record probe information at each step because only 200-300 points are sufficient for a good plot. This command instructs **KB1** to make entries in probe files at intervals. For example, if *NStepProbe* = 5 the code makes a record every fifth time step. The purpose of the *ProbeStep* command is to keep probe files at a manageable length. You can make long files if you want. The **Probe** program has an automatic interval feature so that the number of loaded points does not exceed the dimensions of the plot array.

**Parameters:** The integer parameter *NStepProbe* is the number of code time steps per probe file record. The default is *NStepProbe* = 1.

## Section 12. Command summary

General program control
GEOMETRY [Rect,Cylin,Sphere]
TITLE Run title string
TMAX TMax
DT Dt
SAFETY Safety
NCHECK NCheck
DTCHANGE Time DtNew
PRESSURE TabName [Inside,Outside]
FIXBOUND [Inside,None,Outside,Both]
INTERPORDER [First, Second]
ERRSTATUS [All,Partial,None]

<b>Material properties</b>
MATERIAL MatNo KBT TabName MATERIAL MatNo VOID [DBounce] MATERIAL MatNo EXPL C0 S1 S2 PInit Q Gamma [INIT] MATERIAL MatNo SHOCK C0 S1 S2 MATERIAL MatNo GAMMA Gamma
<b>Geometry</b>
REGION RegNo MatNo Ri Ro NElem
<b>Region properties</b>
VELINIT RegNo Vinit [WEIGHTED] HYDINIT RegNo Rho Temperature (KBT materials) HYDINIT RegNo Rho U (Gamma, Shock and Expl materials) VISCOSITY RegNo Viscosity
<b>Diagnostics</b>
DIAGDT DTime DIAGTIME TDiag DIAGSTEP NDiag SETPROBE PIndex(NProbe) PROBESTEP NStepProbe

# Tutorial 01. One-dimensional shock resulting from the collision of slabs

The **KB1** simulation models a planar shock resulting from the impact of a moving slab striking a stationary target. An aluminum slab of thickness 0.5 cm moving at  $v_o = 10$  km/s strikes a Lucite target of thickness 1.0 cm. We can predict the properties of shocks moving into the materials from the contact point using the shock conservation equations (Sect. 3). A shock wave propagates into the Lucite with pressure  $p_2$  and particle velocity  $u_{p2}$ . Similarly, a shock propagates backward in the aluminum with pressure  $p_1$  and particle velocity  $-u_{p1}$  relative to the moving medium. Two conditions must be satisfied at the contact point between the materials:

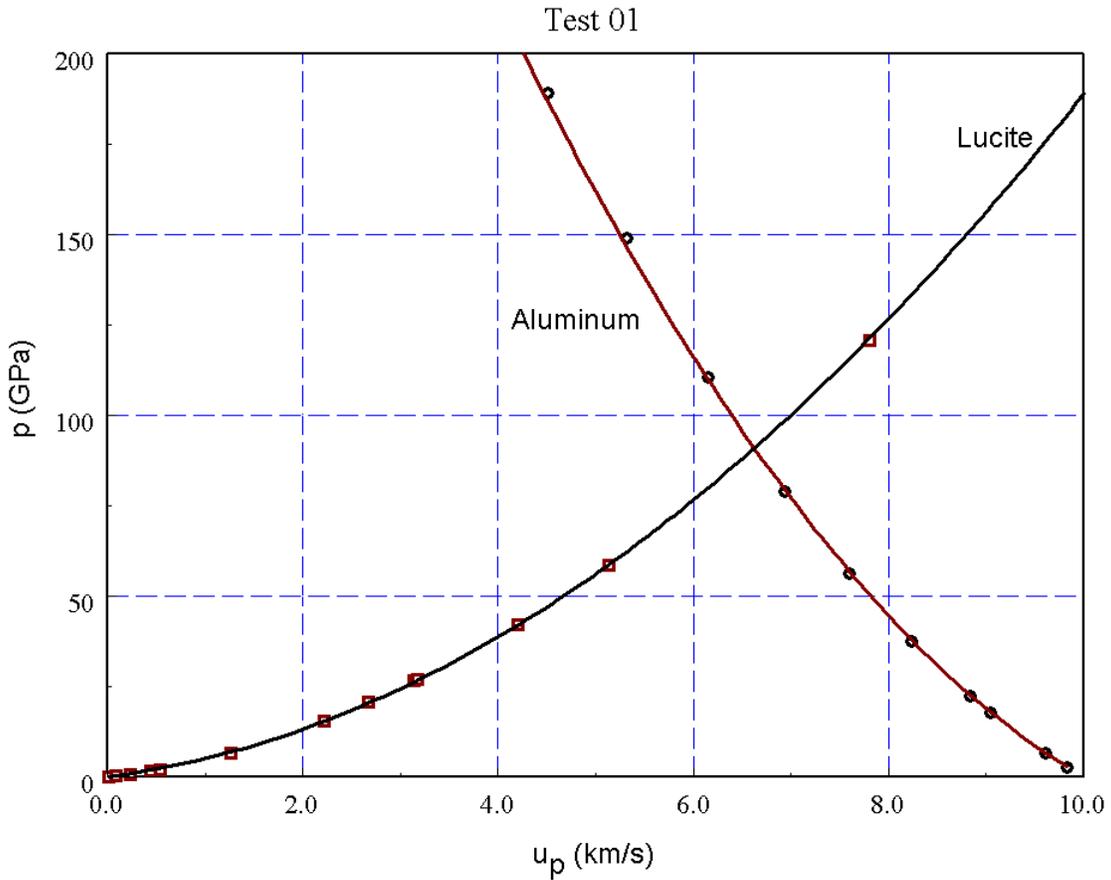
# The pressures are equal,  $p_1 = p_2$ .

# The particle velocities in the stationary frame are equal,  $u_{p2} = v_o - u_{p1}$ .

These conditions, combined with the Hugoniot relationships, define unique shock states in both materials.

One way to find the state is to plot the Hugoniot curve  $p_1(u_{p1})$  and the shifted curve  $p_2(v_o - u_{p2})$  and identify the intersection. Using **KBTView** to generate Hugoniot curves for the materials ALUM3715 and LUCT7750 leads to the plot of Fig. 1. The common point corresponds to the following shock parameters:  $p_1 = p_2 = 91$  GPa,  $u_{p2} = 6.62$  km/s and  $u_{p1} = 3.38$  km/s. The predicted shock velocity in Lucite with a density of  $\rho_2 = 1186$  kg/m<sup>3</sup> is  $u_s = p_2 / \rho_2 u_{p2} = 11.6$  km/s. The shock velocity in the rest frame of the aluminum ( $\rho_1 = 2700$  kg/m<sup>3</sup>) is  $u_s = p_1 / \rho_1 u_{p1} = 9.97$  km/s, so the shock front velocity in the stationary frame is 0.03 km/s.

The following files are required for the simulation: SLAB01.KIN, ALUM3715.KBT and LUCT7750.KBT. The **KB1** control file SLAB01.KIN is listed at the end of this section. The contact point is initially at position  $x = 0.5$  cm. Figure 2 shows the calculated pressure profile 350 ns after contact. Filled squares indicate the positions of aluminum elements and filled circles represent Lucite. The predicted positions of the shock fronts are 0.906 cm in the Lucite and 0.501 cm in the aluminum. The code values for pressure and shock positions agree well with the predictions.



**Figure 1.** Calculation of shock parameters from the Hugoniot curves.

```

* File SLAB01.KIN
* ----- Run control -----
GEOMETRY: RECT
DT: 0.25E-9
TMAX: 500.0E-9
* ----- Material properties -----
MATERIAL 1 KBT ALUM3715
MATERIAL 2 KBT LUCT7750
* ----- Geometry -----
* Aluminum flyer plate
  REGION 1 1 0.000 0.005 40
* Lucite target
  REGION 2 2 0.005 0.015 80
* ----- Region properties -----
VISCOSITY 1 5.0
VELINIT 1 1.0E4

```

```
HYDINIT 1 2.7000E+03 298.0
VISCOSITY 2 5.0
VELINIT 2 0.0
HYDINIT 2 1.1860E+03 298.0
* ----- Diagnostics -----
DTIME 50.0E-9
ENDFILE
```

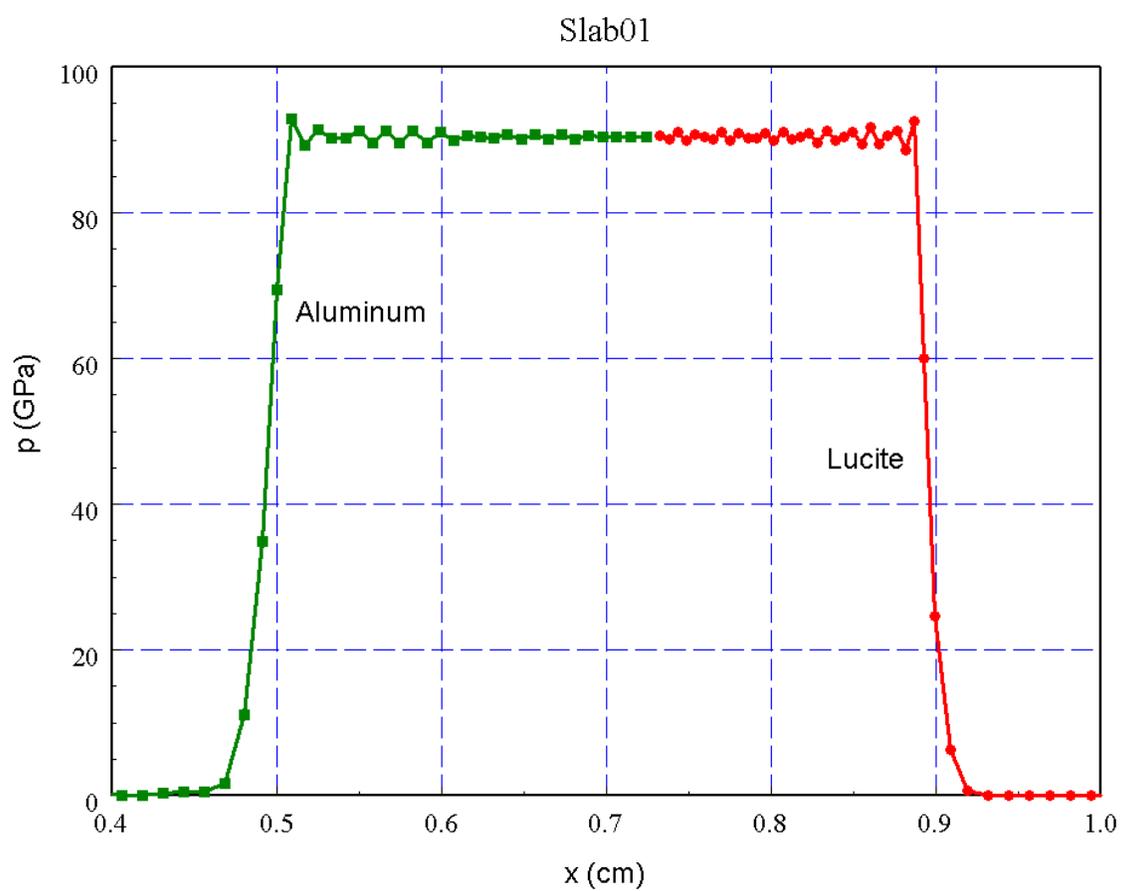


Figure 2. Pressure profile at 350 ns.

# Tutorial 02. Planar detonation wave in Composition B

In this **KB1** simulation the impact of a thin aluminum on a slab of the explosive Composition B initiates a planar detonation wave. We can compare results with the predictions of Section 8. The input files are `PLANEDET.KIN` and `ALUM3715.KBT`. The input script file has the contents

```
GEOMETRY: RECT
DT: 1.0E-9
TMAX: 10.1E-6
MATERIAL 1 KBT ALUM3715
MATERIAL 2 EXPL 1.56E+03 2.46E+03 3.52E+00 -8.810E-04 2.0E10 5.183E+06 2.63
REGION 1 1 -0.001 0.000 2
REGION 2 2 0.000 0.100 200
VISCOSITY 1 5.0
VISCOSITY 2 5.0
VELINIT 1 8.0E3
DTIME 2.00E-6
SETPROBE 3
PROBESTEP 1
```

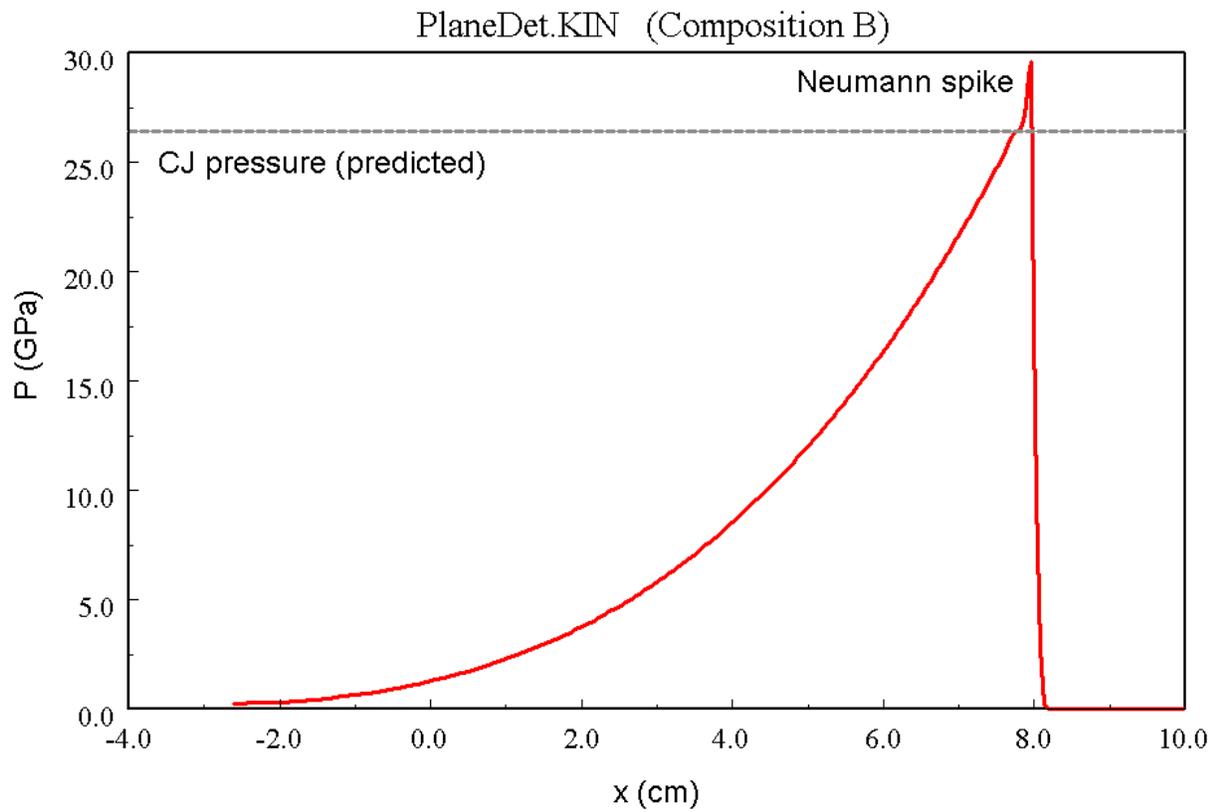
Line 5 defines properties of Composition B. The explosive has  $\rho_o = 1561 \text{ kg/m}^3$ ,  $Q = 5.186 \text{ MJ/kg}$  and  $\gamma = 2.63$ . Equation 8.9 predicts a Chapman-Jouguet pressure of  $p_{CJ} = 26.4 \text{ GPa}$ . The initiation pressure is set at 20 GPa, a value somewhat less than  $p_{CJ}$ . The equations of Sect. 8 give the following values for other material parameters at the CJ point:  $u_d = 7.83 \text{ km/s}$ ,  $u_p = 2.16 \text{ km/s}$ ,  $e = 7.51 \text{ MJ/kg}$  and  $\rho = 2154 \text{ kg/m}^3$ .

The explosive has a thickness of 10 cm. During the 10  $\mu\text{s}$  run the detonation front should move 7.83 cm. A probe is placed in the first element to check conditions for detonation. The aluminum plate (Region 1) is 1 mm thick. No detonation wave is observed for plate velocities less than 4 km/s. At the critical velocity of 4 km/s, the first element of the Composition B reaches 20 GPa at 85 ns after contact. At this time, the particle velocity up in the element is 1.9 km/s, close to the CJ value. A incident velocity of 8 km/s was used in the production run for fast initiation. In this case, the first element detonated 15 ns after impact.

The spatial variation of pressure in Figure 1 (10  $\mu\text{s}$  after initiation) follows the expected behavior of an ideal detonation wave [see, for instance, M.A. Meyers, *Dynamic Behavior of Materials* (Wiley, New York, 1994), 246]. The code gives the following parameters at the CJ point, in excellent agreement with predictions:

Pressure: 26.4 GPa  
Position at 10  $\mu\text{s}$ : 7.78 cm

Inferred detonation velocity,  $u_d$ : 7.78 km/s  
Density,  $\rho$ : 2158 kg/m<sup>3</sup>  
Internal energy,  $e$ : 7.51 MJ/kg  
Particle velocity,  $u_p$ : 2.17 km/s



**Figure 8.** Spatial variation of pressure at 10  $\mu$ s

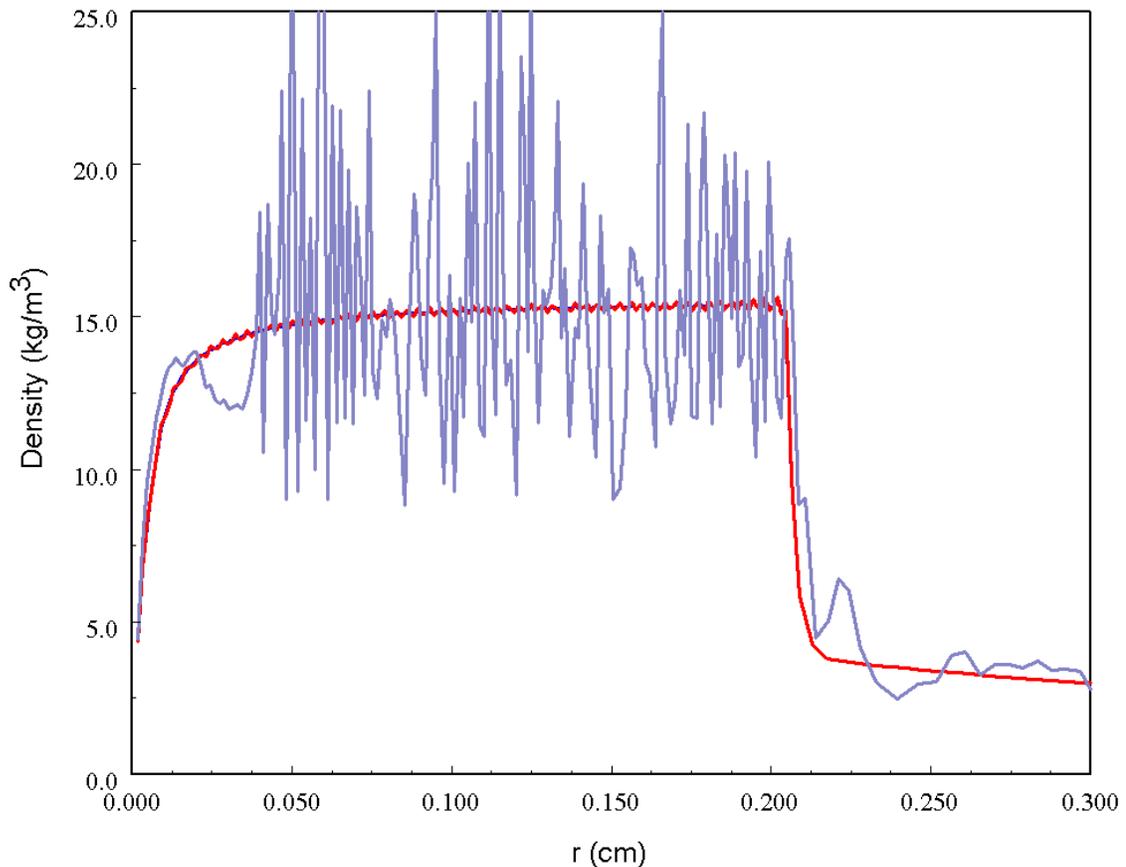
# Tutorial 03. Compression shock in an ideal gas

The **KB1** code models a standard hydrodynamic test, initiation of a shock by a uniform-velocity compression of an ideal gas. The simulation illustrates comparisons between the ideal gas and the KB Table material models as well as the relative accuracy of KB interpolations. In the model an ideal gas with  $\gamma = 5/3$  has an initial uniform density  $\rho_0$ . At all positions the gas has an initial velocity  $v_0$  directed toward a cylindrical axis. For the cylindrical bases, hydrodynamic theory predicts that a shock wave with density  $16\rho_0$  propagates outward from the axis with a velocity  $u_s = v_0/3$ .

The input script file has the contents

```
* --- RUN CONTROLS ---
* (Test the geometry)
GEOMETRY: CYLIN
* GEOMETRY: PLANE
* FIXBOUND: IN
* GEOMETRY: SPHERE
DT: 1.0E-9
DTCHANGE: 0.5E-6 2.50E-9
TMAX: 6.01E-6
* (Test the interpolation accuracy)
* INTERPORDER FIRST
* INTERPORDER SECOND
* --- MATERIAL PROPERTIES ---
* (Test the material model)
MATERIAL 1 GAMMA 1.000 1.6667
* MATERIAL 1 KBT IGAS0002
* --- GEOMETRY ---
REGION 1 1 0.0000 0.0100 250
* --- REGION PROPERTIES ---
HYDINIT 1 1.000 0.001
VELINIT 1 -1000.0
VISCOSITY 1 2.00
* --- DIAGNOSTICS ---
PROBESTEP 10
SETPROBE 56
DIAGTIME 3.00E-6
DIAGTIME 6.00E-6
ENDFILE
```

Alternative run parameters are shown as comments.



**Figure 9.** Spatial variation of density at 6.0  $\mu$ s. Red: Results for a  $\gamma$ -law material model and for second-order interpolation on the KB Table IGAS0001.

Figure 1 shows simulation results. The red line is a superposition of results using a GAMMA material with  $\gamma = 5.3$  or the KBT material IGAS0001 with initial density  $1.0 \text{ kg/m}^3$ . The two models give almost identical results and are in good agreement with theory. The predicted position of the shock front at 6.0  $\mu$ s is 0.200 cm. The discrepancy from the ideal model near the axis is a feature that appears in all numerical hydrodynamic models. Because the initial pressure is zero, there is no mathematical path to convert the gas kinetic energy to internal energy in the absence of artificial viscosity. The introduction of artificial viscosity resolves the problem, but causes a dip in density and elevated temperature at the turning point. The pressure has the approximately uniform value  $5.3 \times 10^6 \text{ Pa}$  behind the shock. Note that the density in front of the shock is higher than the initial value. A uniform velocity (independent of  $r$ ) directed toward the axis puts the material into compression, even in the absence of the shock.

You can try alternate geometries. For the planar case, the vertex at  $x = 0.0$  is anchored (symmetry

boundary). Theory predicts that  $u_s$  should be the same for planar, cylindrical and spherical systems. Density values behind the shock are predicted to have the following values:  $4\rho_0$  for planar,  $16\rho_0$  for cylindrical and  $64\rho_0$  for spherical geometry.

The time for the run with the KB table is about 20 times that with the analytical model. The difference reflects the extra work involved in the second-order interpolations. This run severely tests the interpolation routines. The final element temperature of 60 °K is smaller than the first tabulated temperature of 100 °K in IGAS0001. For this reason, the first-order interpolation performs poorly (blue line in Fig. 1). This is an unusual case. It is important to note that first order interpolation usually give sufficient accuracy in more common runs with high shock temperatures.

# Tutorial 04. Sedov blast wave

The Sedov blast wave is an analytic solution for the propagation of a spherical shock in a uniform ideal gas. The shock is initiated by a delta-function distribution of internal energy at the origin. The theory is described in Ya. B. Zel'dovich and Yu. P. Raizer, **Physics of Shock Waves and High-temperature Hydrodynamic Phenomena** (Academic Press, New York, 1966), 93.

Consider a uniform ideal gas described by the parameter  $\gamma$  with initial density  $\rho_o$ . A total energy  $E_o$  is deposited near the origin. The radius of the spherical shock as a function of time is given by

$$r = \zeta_o \left( \frac{E_o}{\rho_o} \right)^{1/5} t^{2/5} .$$

For the choice  $\gamma = 5/3$ , the factor in the equation is  $\zeta_o = 1.15$ . The shock velocity is

$$u_s = \frac{2}{5} \zeta_o^{5/2} \left( \frac{E_o}{\rho_o} \right)^{1/2} r^{-3/2} .$$

The density and pressure immediately behind the shock front are

$$\rho_s = \rho_o \left( \frac{\gamma+1}{\gamma-1} \right), \quad p_s = \rho_o u_s^2 \left( \frac{2}{\gamma+1} \right).$$

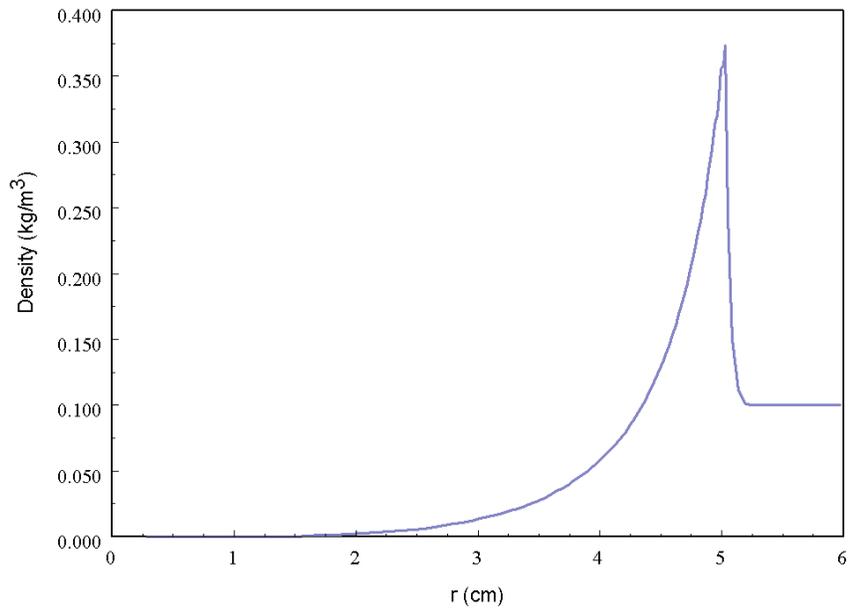
The **KB1** run `SEDOV.KIN`, illustrated below, uses either the `GAMMA` model or the `KBT` table `IGAS002`. The zero temperature gas with density  $\rho_o = 0.1 \text{ kg/m}^3$  extends to a radius of 6 cm. The point source of energy is represented by a region of radius 2 mm with a uniform internal energy. The choice of total energy  $E_o = 617 \text{ J}$  gives a shock radius of  $r = 5.0 \text{ cm}$  at  $t = 5.0 \text{ } \mu\text{s}$ . The mass of the central region is  $3.351 \times 10^{-9} \text{ kg}$ , giving an initial internal energy of  $U_o = 1.84 \times 10^{11} \text{ J/kg}$ . The theoretical values of density and pressure at the shock front are  $\rho_s = 0.4 \text{ kg/m}^3$  and  $p_s = 1.19 \times 10^6 \text{ Pa}$ .

```

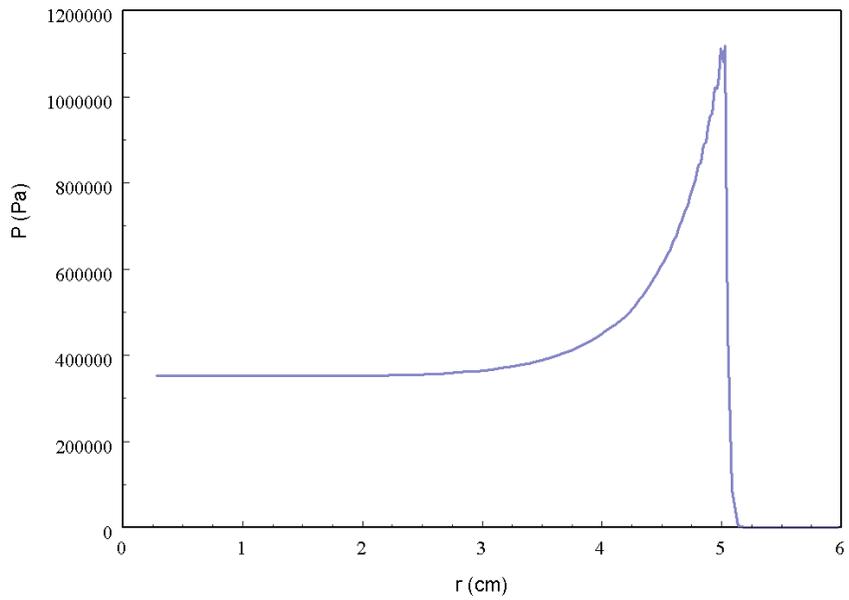
TITLE Sedov blast wave
*
* --- RUN CONTROL ---
GEOMETRY: SPHERE
* INTERPORDER: SECOND
DT: 0.25E-9
DTCHANGE 0.5E-6 2.5E-9
TMAX: 5.01E-6
* --- MATERIAL PROPERTIES ---
* MATERIAL 1 KBT IGAS0002
MATERIAL 1 GAMMA 1.66667
* --- GEOMETRY ---
REGION 1 1 0.0000 0.002 5
REGION 2 1 0.0020 0.060 90
* --- REGION PROPERTIES ---
* HYDINIT 1 0.1000 2.135E7
HYDINIT 1 0.1000 1.84E11
VISCOSITY 1 2.50
* HYDINIT 2 0.1000 0.001
HYDINIT 2 0.1000 0.000
VISCOSITY 2 2.50
* --- CONTROL ---
DIAGTIME: 5.0E-6
PROBESTEP: 25
SETPROBE: 70
ENDFILE

```

Figures 1 and 2 show the spatial variation of density and pressure at 5  $\mu$ s. The shock position and values of  $\rho_s$  and  $p_s$  are in excellent agreement with theory.



**Figure 10.** Variation of density at 5 μs.



**Figure 11.** Variation of pressure at 5 μs