

Cav_KO User Guide

Release Notes

Cav_KO is a 1-D Lagrangian Hydrocode written in Visual Basic (based on the Fortran code by John Borg) and imbedded as a macro in Excel. The programme is designed to be used to simulate plate impact experiments with up to 10 materials and 20 Lagrangian Markers (gauge simulators).

Contents

How do I?

[Set up a Simulation?](#)

[Analyse data from a Simulation?](#)

[Add a material to the Material Library?](#)

[Edit the details of a Material in the Library?](#)

[Main Simulation Platform](#)

[Edit Material Data Menu](#)

[Add New / Change Material Menu](#)

[Edit Simulation Parameters Menu](#)

[Plotting a graph of Data](#)

[Viewing the x-t diagram](#)

How do I...

[Set up a Simulation?](#)

[Analyse data from a Simulation?](#)

[Add a material to the Material Library?](#)

[Edit the details of a Material in the Library?](#)

Setting up a Simulation

1. Select the number of materials you wish to use and their details, using the ["Edit Material Data" menu](#). Open this menu using the button in the [Main Platform](#).
2. Select the Simulation details you want to use in your experiment using the ["Edit Simulation Details" menu](#). (Important input is experiment Runtime and the positions of Lagrangian markers at which data is to be recorded). Open this Menu using the button in the [Main Platform](#).
3. Once all details are set, click the button "Begin Simulation" in the [Main Platform](#). (Once started, the simulation can't be stopped within the GUI. If the simulation must be halted, use CTRL+BREAK to stop the Visual Basic compiler). The time elapsed in the simulation is displayed in the upper right hand corner.
4. Cav_KO will run the simulation in the desired View Mode, or display an error report if Simulation details are incorrect. The Simulation time elapsed is displayed in the top right hand corner. Once the Simulation is complete the message "Finished" will be displayed onscreen.

Analysing data from a Simulation

1. Raw data from the simulation can be viewed in the "Data" worksheet and shock data is plotted in the [x-t diagram](#) in the worksheet "x-t diagram". Data can be [plotted](#) using the "Plot Graph" button in the [Main Platform](#).
2. Data can be copied from the "Data" worksheet manually and used in a separate Excel worksheet, or other data analysis package.

Adding a Material to the Material Library

1. Open the ["Edit Material Details" menu](#), using the button in the [Main Platform](#)
2. Click the ["Add New Material"](#) button and use the menu to add new material details

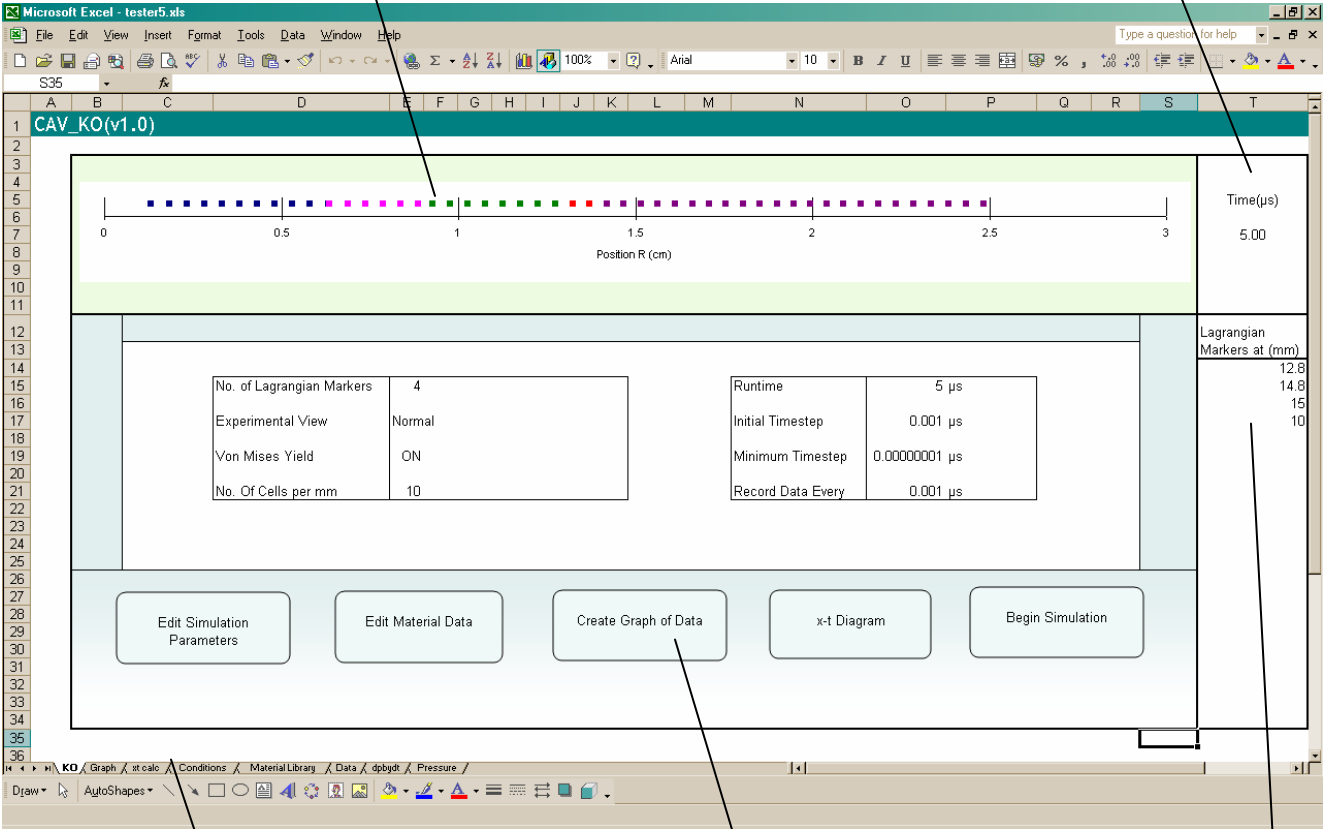
Editing the details of a Material in the Material Library

1. Open the ["Edit Material Details" menu](#), using the button in the [Main Platform](#)
2. Click the ["Change Material Data"](#) button and use the menu to add new material details.

Main Simulation Platform

Material Position Display

Time Elapsed

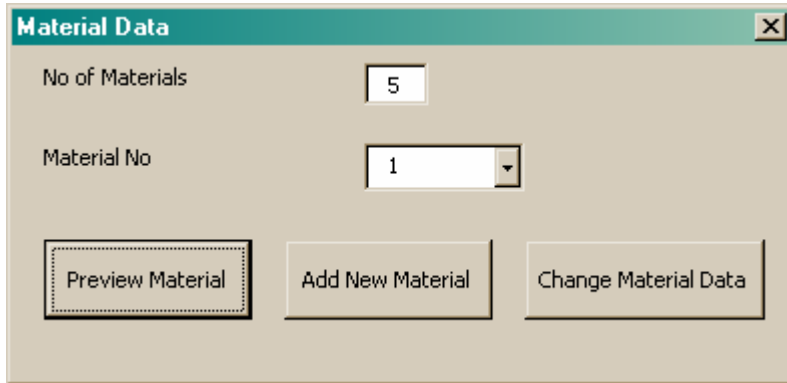


Use tabs to view worksheets of manually

Use Buttons to Navigate

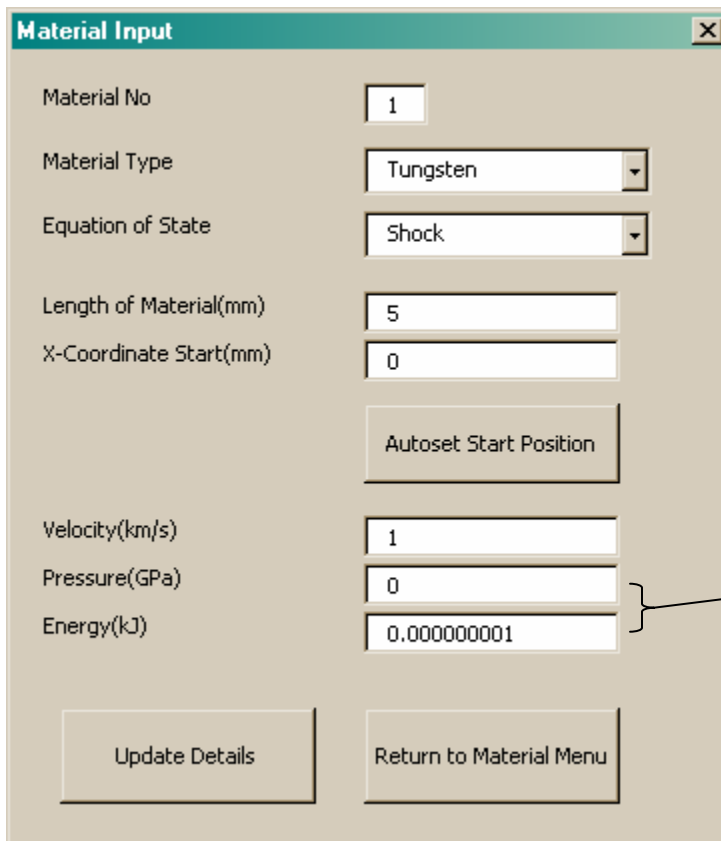
Shows the position Lagrangian Markers

Edit Material Data Menu



The "Material Data" dialog box has a title bar with a close button. It contains two input fields: "No of Materials" with the value 5, and "Material No" with a dropdown menu showing 1. Below these are three buttons: "Preview Material" (highlighted with a dashed border), "Add New Material", and "Change Material Data".

To add materials to the simulation, use the "Edit Material Data" button on the [Main Platform](#), to bring up the materials menu. Here you can select how many materials you wish to use in the simulation. To add details for each material in the simulation, select the material number in the drop down menu and go to "Preview Material" to bring up the "Material Input" menu and edit the material details.



The "Material Input" dialog box has a title bar with a close button. It contains several input fields: "Material No" (1), "Material Type" (Tungsten), "Equation of State" (Shock), "Length of Material(mm)" (5), and "X-Coordinate Start(mm)" (0). There is an "Autoset Start Position" button. Below these are three more input fields: "Velocity(km/s)" (1), "Pressure(GPa)" (0), and "Energy(kJ)" (0.000000001). At the bottom are two buttons: "Update Details" and "Return to Material Menu".

Pressure and Energy settings can be ignored for plate impact experiments

Select the material type you want in the drop down menu.

Then select the EOS you want to use for that material – current options are Polynomial, Shock and [Tri-Linear](#). If [Tri-linear](#) is selected, then upon selecting Update Material Details, [a new menu will appear](#) requesting the tri-linear data input. Values for C₀ and S in each region of the tri-linear fit and the compression of the material in which the regions are divided must be inputted.

Then input the length of the material in mm and the starting position of the material. It is important that the materials are adjacent to each other – i.e. the starting co-ordinate of the material is the same as the end co-ordinate of the previous material. (the GUI can do this automatically by pressing “Autoset Start Position”) as an initial gap between materials is not supported.

Input the velocity of the material (for a stationary target this will be 0km/s).

Once the correct material details are inputted, use the Update Details button to refresh this data.

Input Trilinear Details

2nd Slope

Relative Volume Start Interpolation	0	C ₀ (km/s)	0
Relative Volume End Interpolation	0	S	0

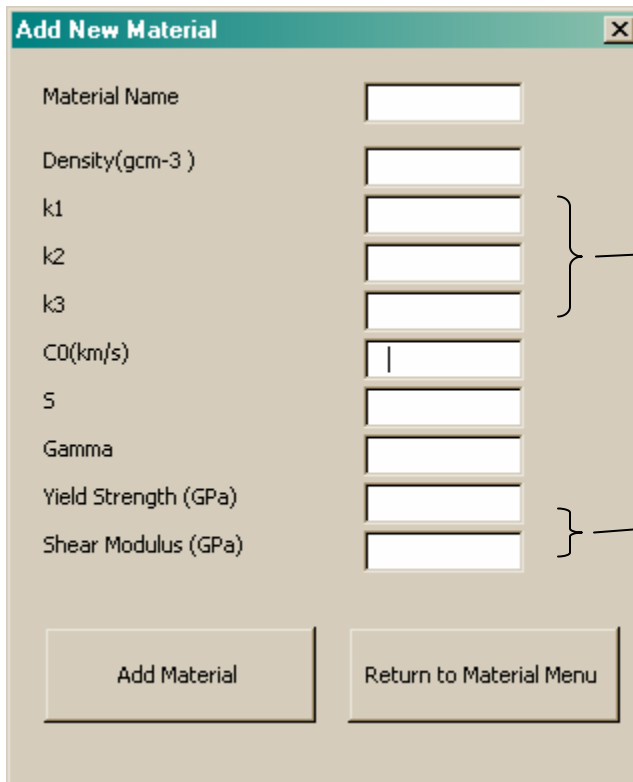
3rd Slope

Relative Volume Start Interpolation	0	C ₀ (km/s)	0
Relative Volume End Interpolation	0	S	0

Note – Relative volume is V/V_0 , so at the start of the simulation it is 1 for every cell. Thus, a value of 0.7 would be a volume 0.7 times the original cell volume. For tri-linear, the Material Library C₀ and S are used for the first region – the “Relative Volume Start Interpolation” sets the compression at which these first values stop being used and the “Relative Volume End Interpolation” set when the 2nd region values start being used. This relative volume must be calculated from the hugoniot

using
$$\frac{V}{V_0} = 1 - \frac{U_s - C_0}{S \cdot U_s}$$

Add New / Change Material Menu



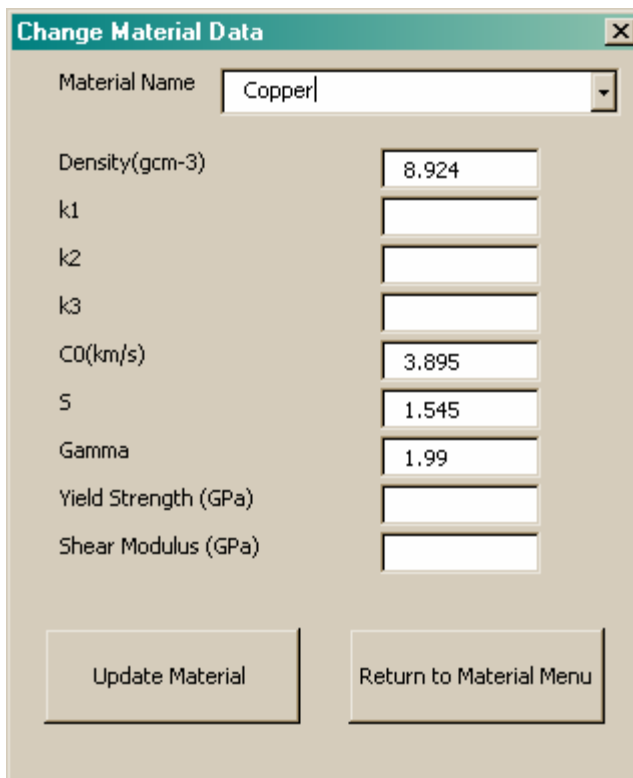
The 'Add New Material' dialog box contains the following fields and controls:

Field Name	Field Type
Material Name	Text Input
Density(gcm-3)	Text Input
k1	Text Input
k2	Text Input
k3	Text Input
C0(km/s)	Text Input
S	Text Input
Gamma	Text Input
Yield Strength (GPa)	Text Input
Shear Modulus (GPa)	Text Input

At the bottom, there are two buttons: 'Add Material' and 'Return to Material Menu'.

k1, k2 and k3 are only used in Polynomial EOS

Yield strength and shear modulus only used with Von Mises Yield



The 'Change Material Data' dialog box contains the following fields and controls:

Field Name	Field Type	Value
Material Name	Dropdown	Copper
Density(gcm-3)	Text Input	8.924
k1	Text Input	
k2	Text Input	
k3	Text Input	
C0(km/s)	Text Input	3.895
S	Text Input	1.545
Gamma	Text Input	1.99
Yield Strength (GPa)	Text Input	
Shear Modulus (GPa)	Text Input	

At the bottom, there are two buttons: 'Update Material' and 'Return to Material Menu'.

Edit Simulation Parameters Menu

Edit Simulation Parameters			
No of Lagrangian Markers	4		
Set Marker Positions	Marker No.	Position(mm)	Runtime
	1	12.8	5
Von Mises Yield	☒	Initial Time Step	0.001
Experimental View	Normal	Minimum Time step	0.00000001
No of Cells Per mm	10	Record Data Every	0.001
Update Simulation Details			

Once material data is inputted you can use the Simulation Details menu to select your experimental conditions.

First set the runtime in Microseconds.

The initial timestep is the timestep that will be used to calculate the first iterative programme step. Subsequently the timestep will be calculated from the sound speed through the material.

The minimum timestep is the lowest timestep that can be used by the programme in calculations. At points of extreme distortion in the material, the timestep may become infinitesimally small, causing the programme to run indefinitely. The minimum timestep thus sets a cut-off, so the programme will end if the timestep drops too low.

To set the frequency with which results are taken adjust the “Record Data Every” setting.

Data will be recorded for each of the markers that are set up in the material. First set the number of markers that are to be used in the simulation. Then set the position of the marker by selecting the marker from the drop down menu and entering a position in mm for the marker. This position is measured using the start of the flyer as 0mm. Data for Pressure, Stress, Particle Velocity and Energy are recorded for each marker.

Von Mises yield can be turned off using the tickbox, although this will have no effect for a material with no data for shear modulus and yield strength. If yield strength data is entered and Von Mises is switched on, then an elastic-perfectly plastic model will be used.

Experimental view sets the display that will be shown once the simulation starts. "Normal" will display the main page and will show a realtime plot of the positions of each of the material in the display at the top of the screen. "dP/dt" will show a new screen when the simulation begins, with a graph displaying dP/dt for each cell in the simulation. In this manner, the shock fronts can be viewed travelling through each material in realtime. Similarly "Pressure" shows a separate screen during the simulation that displays Pressure at each position in the material.

Set No. of Cells per mm sets the number of Lagrangian cells that the programme uses for each mm of material. 10 per mm is standard – this need only be adjusted if distance scales less than 0.1mm are to be used in the experiment. If this is adjusted, then all the details for the number of cells for each of the materials will be updated automatically.

To ensure any changes are made (apart from No. of Lagrangian Markers which is updated automatically), use Update Simulation Details. (to cancel simply use the x to exit menu).

Plotting a graph of Data

Create a Graph [X]

Y-axis []

Versus Time(μ s)

Markers

Position(mm)	Select Markers	Position(mm)	Select Markers
12.8	☐		☐
14.8	☐		☐
15	☐		☐
10	☐		☐
	☐		☐
	☐		☐
	☐		☐
	☐		☐
	☐		☐
	☐		☐

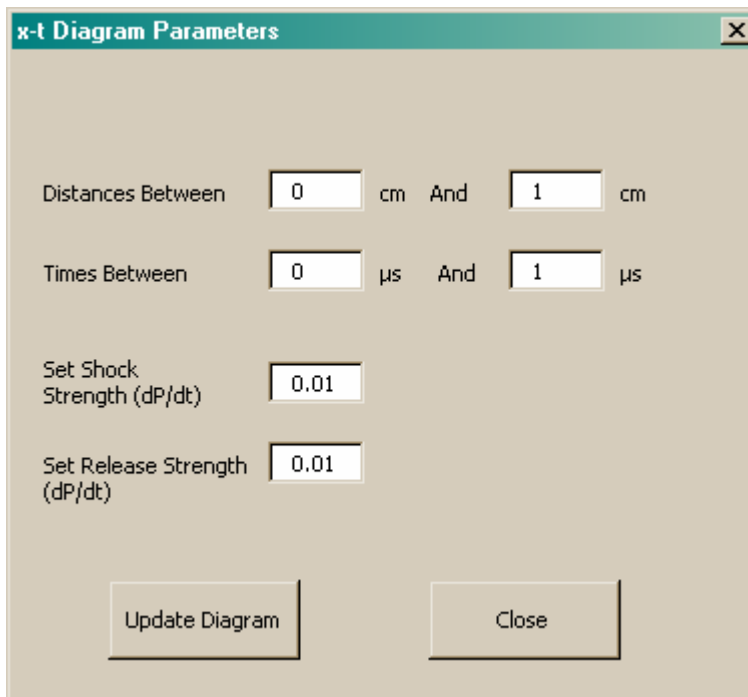
Plot Graph

In order to plot the results from one or more of the Lagrangian markers, go to the [main simulation platform](#). Click the button “Create Graph of Data”. In the Graph menu, select the parameter you wish to plot e.g. “Pressure” (which is default), then select the markers you wish to plot data for, using the checkboxes. Click the “Plot Graph” button to display the graph. To print the graph, press the “Print Graph” button in the “Graph” worksheet.

Viewing the x-t diagram

In order to view the x-t diagram for the simulation, click the button “x-t Diagram” in the [main simulation platform](#). This graph can be printed using the “Print Graph” button. Material boundaries are displayed in multicolour. The dark blue lines represent shock fronts in the materials and the red lines represent releases.

Shocks and releases can be picked out manually, using the “Edit Graph” button. In the Edit Diagram menu, regions on the graph can be selected by setting the start and end of the region in the x direction (distance) and the start and end of the region in the y direction (time). Shocks and releases in the selected region can then be picked out based on the strength of the front – “dP/dt”. Larger values of “dP/dt” represent stronger fronts, so raising this value can remove “noise” from the graph – the standard cut-off value for “dP/dt” is 0.01. Click the “Update Diagram” button to make these changes.



The image shows a software dialog box titled "x-t Diagram Parameters". It contains four input fields for setting parameters. The first two fields are for distance and time ranges, both set from 0 to 1. The next two fields are for shock and release strength thresholds, both set to 0.01. At the bottom, there are two buttons: "Update Diagram" and "Close".

Parameter	Value	Unit
Distances Between	0	cm
And	1	cm
Times Between	0	μs
And	1	μs
Set Shock Strength (dP/dt)	0.01	
Set Release Strength (dP/dt)	0.01	

Buttons: Update Diagram, Close