

RELEASE NOTES

# PHAST™

Taking hazard and risk analysis one step further

Version: 8.0  
Date: October 2017





Reference to part of this report which may lead to misinterpretation is not permissible.



Date: October 2017

**Prepared by: DNV GL - Software**

© DNV GL AS. All rights reserved

This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV GL AS

---

## Table of contents

1	NEW FEATURES APPLICABLE TO ALL PHAST USERS .....	2
1.1	Improved dispersion calculations for short-duration releases.....	2
1.2	Modelling of crater formation for breaches in long pipelines.....	7
1.3	Improvements in modelling and results for time-varying discharge calculations.....	8
1.4	Time-varying modelling of fireball size and intensity .....	9
1.5	Improved options for modelling expansion at the beginning of the release .....	10
1.6	Parallel processing available for dispersion and toxic calculations .....	10
2	NEW FEATURES APPLICABLE TO USERS OF THE FULL PHAST PROGRAM.....	11
2.1	Warehouse Model for modelling toxic plumes from a warehouse fire.....	11
2.2	Run in Parallel option for running multiple Scenarios at the same time.....	13
3	OTHER DIFFERENCES AND BUG FIXES .....	15
3.1	More options for specifying process flow conditions for Short pipe Scenarios .....	15
3.2	New option for modelling of a Standalone BLEVE Blast explosion.....	16
3.3	New option for modelling jet flames that impinge on the ground.....	16
3.4	New options for modelling a bund .....	16
3.5	Materials can be exported and imported between workspaces .....	17
3.6	Calculations for dispersion, flammable and toxic effects now all use the same height of interest	17
3.7	Building wake modelling now selected by default .....	18
3.8	Baker-Strehlow-Tang explosions now involve entire cloud volume.....	18
3.9	Simplification of options for initial view time for Dispersion Graphs.....	18
3.10	Simplification of consequence reporting for explosion .....	18
3.11	Simplification of some options for explosions .....	19
3.12	Some Long pipeline inputs simplified and clarified .....	19
3.13	Bug fixes .....	21
4	ALERTS AND WORKAROUNDS.....	23

# 1 NEW FEATURES APPLICABLE TO ALL PHAST USERS

The following features are applicable to users of Phast and Phast Lite, and to users with and without the extensions for multi-component modelling and 3D explosion modelling.

## 1.1 Improved dispersion calculations for short-duration releases

Releases with short durations pose a particular challenge for modelling, because the modelling of continuous releases performed in the core dispersion calculations assumes that the duration is sufficient for the release to reach a steady state in which air is entrained only through the sides of the cloud, and in which entrainment in front of and behind the cloud can be neglected. For short releases which do not reach a steady state, this approach to the modelling underestimates the overall entrainment rate and the concentration calculated at a given downwind distance is likely to be too high. These challenges apply also to releases with time-varying release rates or pool vaporisation rates; the total release duration might be relatively long, but the length of time for which the release has a given rate is too short to allow the cloud to reach a steady state for that particular rate.

The methods that were available for modelling such releases in previous versions had serious limitations, taking a simplistic approach that introduced visible discontinuities in the dispersion results. A new method is now available which has none of these limitations, but is a rigorous method that produces smooth, consistent, time-dependent dispersion profiles that include all of the information required by the risk calculations. This new method is called the **along wind diffusion (AWD)** method. It is set as the default method, so it will be used for all new workspaces, and for all upgraded workspaces for which the Dispersion Parameters are set to use the default method.

The along wind diffusion method will give differences in the concentration results for many continuous releases and for releases with pool vaporisation. The technical documentation supplied with the program includes a document entitled **Results differences between Phast and Safeti versions** that describes the type of differences to expect.

Implementing the Along wind diffusion method has involved making changes in some of the concepts underlying the dispersion calculations, and you will see the effect of these changes in the Reports and Graphs, and also in the form of the input data for some types of Scenario. These concepts and the main effects are described briefly below; for a fuller description, enter "Dispersion modelling overview" in the Index tab of the online Help.

*The main changed concept: the core dispersion calculations are now performed for "release observers" instead of "release segments"*

The core dispersion calculations model the changing conditions for **observers** that are released over the course of the event to move downwind with the cloud. An observer can be imagined as a particle-sized sensor that is released at the centreline of the cloud at a particular time and is then carried along with it.

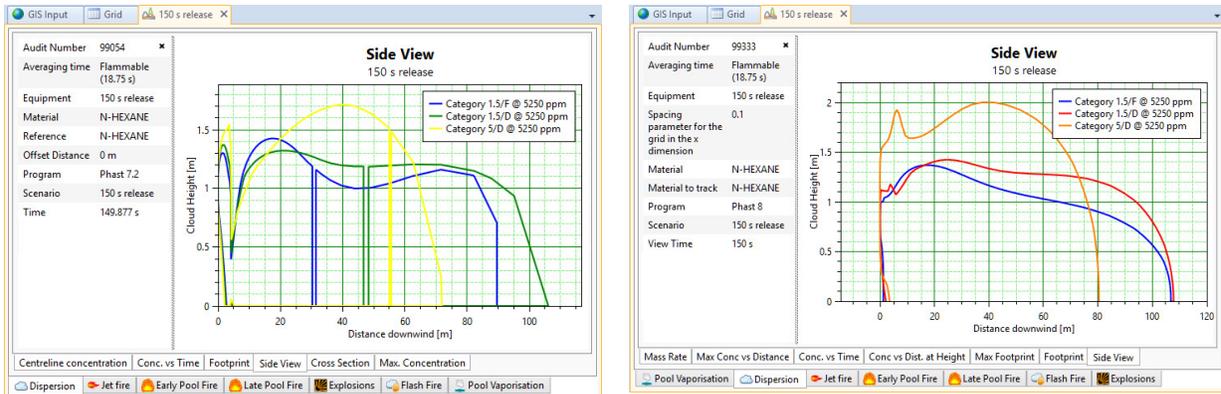
The modelling considers two types of observer: **instantaneous observers** that are used to model the start of an instantaneous release, and **continuous observers** that are used to model continuous releases and the vapour generated by an evaporating pool.

The observers used in the modelling of a particular Scenario and Weather are listed at the start of the Dispersion Report, as shown below for an instantaneous release with rainout and evaporation.

Observer Release Data and Observer Mass Data					Masses or mass rates			
Observer number	Release type	Start time [s]	Start downwind distance [m]	Unit	Release	Rainout	Pool vapour pick-up	Final
1	Instantaneous	0	0	kg	2000	593.801	339.059	1745.26
2	Continuous	13.9219	-7.097	kg/s	0	0	10.7407	10.7407
3	Continuous	16.4635	-7.097	kg/s	0	0	8.9126	8.91261
4	Continuous	23.4413	-7.097	kg/s	0	0	5.93604	5.93604
5	Continuous	35.4536	-7.097	kg/s	0	0	3.49846	3.49846
6	Continuous	59.6703	-7.097	kg/s	0	0	1.59758	1.59758
7	Continuous	152.268	-7.097	kg/s	0	0	0.226315	0.226316

The illustration above shows that each observer has a start time but no duration, and this is because each observer is tracking the cloud from the specific conditions associated with its start time, in a release that is recognised as being in a state of continuous change. This is different from the previous approach using release segments, in which each segment had an associated duration because it was used to represent the conditions over a particular period of the release, with the conditions assumed to be constant over the duration of the segment. The **Observer** method is designed to be able to interpolate between the results for the different observers to give smooth results for intermediate situations, whereas the previous approach was only able to treat the release as a series of step-changes.

This difference can be seen in the graphs below for a 150 s hexane release that gives rainout and pool vaporisation. Both graphs show the side view of the cloud at 150 s, and the v7.2 graph on the left shows the discontinuities between the different release segments and a simplistic, linear profile at the front of the cloud, whereas the v8 graph on the right has no discontinuities and has a rounded profile at the front of the cloud showing the effects of the along-wind diffusion and spreading.



The fact that an observer does not have an associated duration means that any event that includes continuous observers must always have at least two continuous observers: one released at the start of the discharge or pool vaporisation, and one released at the end. This is the case even with the simplest situation of a Scenario with a uniform discharge rate and no rainout: you will see in the Dispersion Report that this type of Scenario has been modelled with two identical continuous observers, the first with a start time of 0 s and the second with a start time equal to the duration. The final observer is necessary to mark an end to the release and to allow the program to interpolate properly for intermediate times before the end.

*The modelling of along-wind effects is included in the core dispersion calculations for an instantaneous observer, but for continuous observers it is performed in a new post-processing stage*

The core dispersion calculations treat the modelling of along-wind effects in the same way as in previous versions, which means that these calculations include these effects for an instantaneous observer, but not for a continuous observer. For a continuous observer, the core dispersion calculations assume that the observer is in a steady-state cloud in which the along-wind gradients in concentration and density are small and not able to drive mixing of air in the along-wind direction, and so the mixing of air into the cloud takes place only in the cross-wind direction.

The modelling of along-wind diffusion and gravity spreading for continuous observers is performed instead through post-processing of the results of the core dispersion calculations, where the along-wind diffusion is accounted for by a process of Gaussian integration of the concentrations calculated for the observers. The type of post-processing that is performed depends on the settings in the **Time-varying and finite duration** tab of the Dispersion Parameters. A range of options are provided for different aspects of the post-processing, but the choice is provided mainly to allow comparison with earlier versions of the program, and the options that will give the most accurate modelling of along-wind behaviour are set as the defaults.

In some situations you may see the results for the core dispersion calculations referred to as the "pre-AWD" results, or as the results "before along-wind-diffusion effects".

*The Equipment Reports give concentration values from the core dispersion calculations, whereas the Summary Reports and the Graphs give the results after the post-processing*

In the group of Equipment Reports, the Dispersion, Commentary and Averaging Times Reports give the results of the core dispersion calculations, without any post-processing, whereas the Summary Reports and the Graphs give the results with post-processing.

For a Scenario that is modelled with only an instantaneous observer, the results in the three Equipment Reports will match the results in the Summary Report and the Graphs, because the core dispersion calculations for an instantaneous observer include the modelling of along-wind effects so no additional post-processing is needed in this situation. However, if the modelling for a Scenario includes any continuous observers, the different forms of results may show differences in the concentrations calculated for a given downwind distance at a given time.

The size of the differences will depend on the time-scale for the event-duration compared with the release duration that would be required for a steady-state, fully-developed cloud to become established. For example, if a release duration of 600 s would be required for a cloud with a given mass-rate to reach a steady state, then a Scenario that maintains that mass-rate for 600 s will show differences only at the beginning and end of the release, whereas a Scenario that maintains the mass-rate for only 10 s will show significant differences at all times and distances.

*There have been some changes in the Dispersion Graphs as part of the new method*

This can be seen in the illustrations of the Side View graphs on the previous page, where the list of graphs are different in v7.2 and v8. The graphs of cloud concentration in the Dispersion group are now as follows:

<b>Graph Name</b>	<b>Description</b>	<b>Equivalent in previous version</b>
Mass Rate	Shows the initial release rate, and if rainout occurs, the graph will also show the Rainout Rate and the Pool Vaporisation Rate. All rates are shown as a function of time.	None
Max Conc vs Distance	Shows the maximum concentration reached at a given height as a function of distance downwind, as calculated at a given averaging time.	None
Conc vs. Time	Shows the change in concentration during the course of the dispersion, measured at a location specified by a given downwind distance and height above the ground, and calculated using a given averaging time.	Concentration vs. Time
Conc vs Dist. at Height	Shows the concentration at a given height and time as a function of distance downwind, as calculated at a given averaging time.  Animation is available for this Graph.	None
Footprint	Shows the shape of the contours for up to four concentrations inside the cloud, measured at a given height and time, and calculated using a given averaging time. The graph also shows the maximum extent reached by the liquid <b>pool</b> (if one is formed).  Animation is available for this Graph.	Footprint
Max Footprint	Shows the shape of the contours of the maximum concentration reached during the dispersion, for up to four concentrations inside the cloud, measured at a given height and calculated using a given averaging time. The graph also shows the maximum extent reached by the liquid <b>pool</b> (if one is formed).	Maximum Concentration
Side View	Shows the shape of the contours for up to four concentrations inside the cloud, measured at a given time, and calculated using a given averaging time.  Animation is available for this Graph.	Side View

There is no equivalent in v8.0 of the **Centerline Concentration** and **Cross Section Graphs** that were available in previous versions. The **Conc vs. Time** and **Side View** graphs now always give the results along the downwind axis, i.e. with a crosswind offset of zero, and there is now no option to specify a different value for the crosswind offset.

In previous versions, a Dynamic Concentration Report was available for the Graphs that have animation showing the details of the cloud concentrations at the time currently displayed. This Report is no longer available.

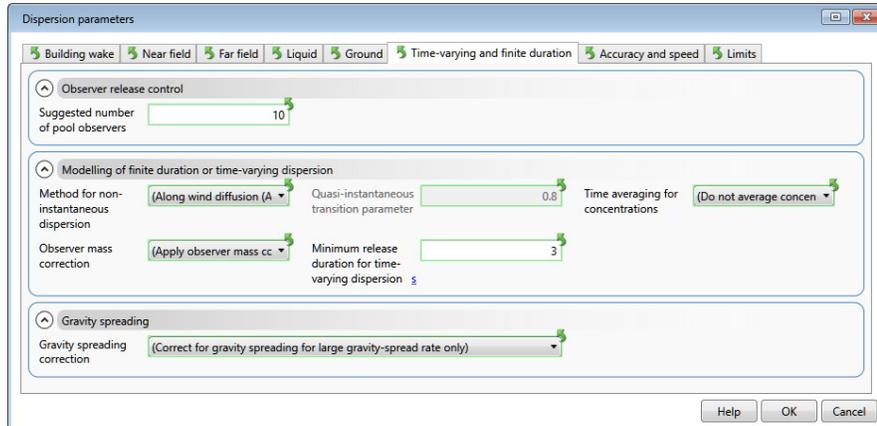
For further details of the Graphs, enter "Graphs" in the Index tab of the online Help.

*There are some changes to the Dispersion Parameters for the new method*

*Note for Phast Lite users:* The dialogs shown in this section are not visible in the program, and the default values that are shown in the illustrations are the values that will always be used in all calculations in Phast Lite.

The **Other** tab in the Dispersion Parameters dialog has been replaced by two new tabs: **Time-varying and finite duration** and **Accuracy and speed**. These new tabs contain fields that were previously on the **Other** tab and also new fields related to the new method, as shown in the illustrations below. You can click on the *Help* button in the dialog for details of the fields.

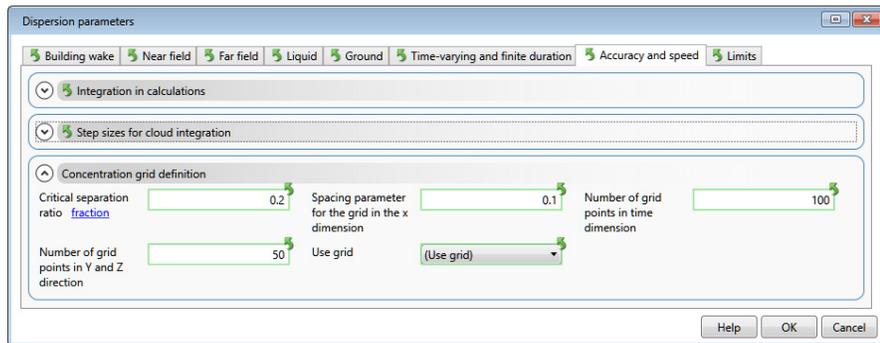
The **Time-varying and finite duration** tab allows you to choose the methods to use for different aspects of the calculations. The default settings are the recommended settings, but other options are provided to allow comparisons.



The list of options for the **Method for non-instantaneous dispersion** includes the methods *QI without duration adjustment*, and *Finite duration correction* are methods that were available in previous versions. However, the *QI with duration adjustment* method that was available in previous versions has been removed because it was the most simplistic of all of the previous methods (e.g. the adjustments were performed in a way that reduced the mass in the cloud).

In the **Accuracy and speed** tab, the new parameters are the parameters in the **Concentration grid definition** group.

For both instantaneous and continuous releases, the program uses the concentration grid in



calculating the size of effect zones for flammable and toxic effects and in preparing dispersion results for plotting on Graphs. These calculations obtain the concentrations at the points on a four-dimensional grid where the dimensions are x location, y location, z location and time. The same grid is used in the post-processing that is performed for continuous observers.

You can use the parameters to change the resolution for different aspects of the grid, where increasing the resolution will give smoother, more-accurate results, but will also make the calculations run more slowly.

*There have been some changes in the Pool Vaporisation Report for Source Scenarios*

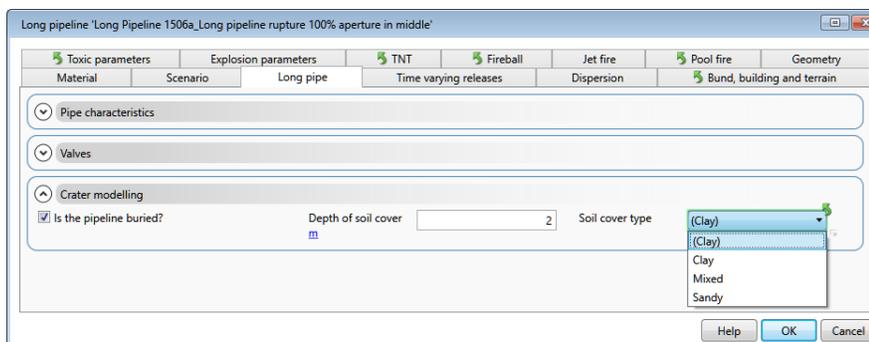
The change from release segments to release observers and the introduction of the concentration grid for calculating effect zones required changes to some of the reports for Source Scenarios. In the process of these changes, some aspects of the reporting were clarified and made more straightforward.

In previous versions the **Pool Vaporisation** report included details of the representative pool vaporisation segments modelled. These are no longer relevant and the Pool Vaporisation report now gives only a small amount of summary information about the pool vaporisation calculations.

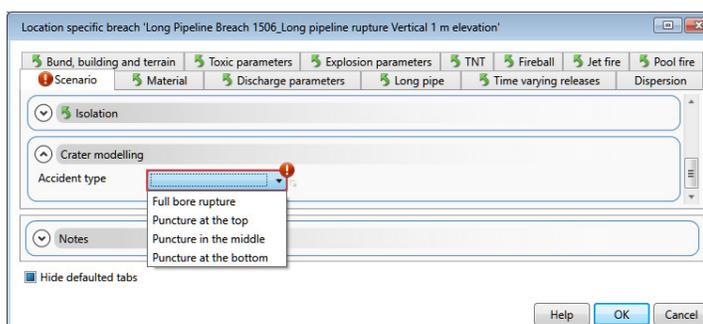
## 1.2 Modelling of crater formation for breaches in long pipelines

The **Long pipe** tab of the Long pipeline dialog now includes a Crater modelling group at the bottom of the dialog as shown.

If you check the box to say that the pipeline is buried, you must specify the **Depth of soil cover**, and choose the **Soil cover type** from the list shown.



The **Scenario** tab of the dialog for the Location specific breach Scenario now also includes a Crater modelling group at the bottom of the dialog. If the pipeline is set as being buried, then the **Accident type** field for the Scenario will be enabled and mandatory as shown, and you must specify the type of accident-geometry that you want to model.



The *Puncture* options involve simple holes, whereas the *Full bore rupture* is a breach that propagates along the pipeline for a distance set by the **Fracture length**, which is a new field in the Long pipe tab section for both types of Breach Scenario. The **Fracture length** field is supplied with a default value that is set in the Long pipe tab of the Discharge Parameters; the default value supplied with the program is 10 m, which is a typical distance between couplings in a long pipeline.

In the calculations for a buried pipeline, the program calculates the depth, width and length of the crater, and it also calculates the mass-flow of air that is mixed in with the release inside the crater and the velocity of the cloud as it leaves the crater, which sets the conditions for the start of the dispersion calculations.

The dimensions calculated for the crater are given in the Location Specific Breach Report, and the conditions for the start of the dispersion can be seen in the Dispersion Report.

The details of the calculations are given in the technical documentation supplied with the program.

## 1.3 Improvements in modelling and results for time-varying discharge calculations

There have been several improvements that affect the Time-varying Leak and Time-varying short pipe Scenarios.

*The value for the inventory is now taken as the total inventory rather than the liquid inventory*

The changes in the time-varying modelling did not involve any visible changes in the input fields, but the interpretation of the **inventory** value in the Material tab for the Equipment item has changed.

In previous versions, the value given for the inventory was taken as the inventory for the liquid side only, and the program would calculate the additional vapour mass needed to fill the vapour space. The value is now taken as the total inventory of both liquid and vapour.

This change means that v8 will typically give less conservative results as there is less liquid to release and the liquid level is more likely to be below the hole in the vessel.

*Improvements in modelling for greater consistency and stability*

The calculations have been improved in several areas:

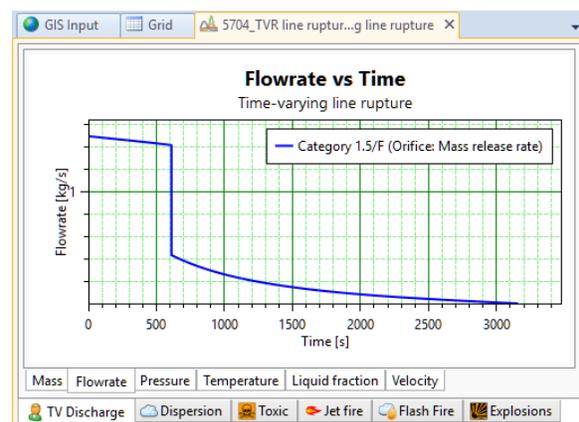
- In previous versions, the entire liquid inventory would typically be released, even in situations in which the liquid level fell below the height of the hole, and this has now been improved. For a pressurised liquid vessel, the release will now stop when the liquid level drops below the height of the hole, and for saturated conditions, the release will change from liquid to vapour.
- The modelling is more robust, and the calculations should no longer stop prematurely because of issues with numerical convergence.
- The modelling is less likely to fail with conditions near the critical point. Some simplifying assumptions have been made for conditions in the vicinity of the critical point, and these make an error in the calculations much less likely.
- The removal of the velocity cap for expansion to atmospheric pressure.

The details of the calculations are given in the technical documentation supplied with the program.

*Graphs of time-varying discharge results are now available*

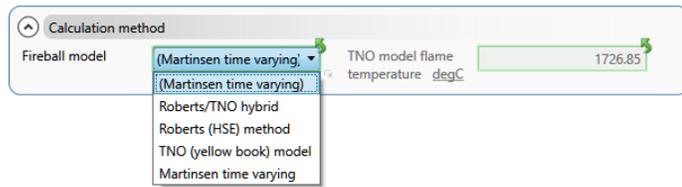
In previous versions, the detailed results of the time-varying discharge calculations were available only in the Time-Varying Discharge Report. The results are now also available in a new group of graphs: the **TV Discharge** Graphs, as shown.

The **Mass** graph shows the mass remaining in the vessel and the mass expelled, the **Flowrate** graphs shows the release rate in the orifice, and the other graphs show values in the vessel, in the orifice, and after expansion to final conditions.



## 1.4 Time-varying modelling of fireball size and intensity

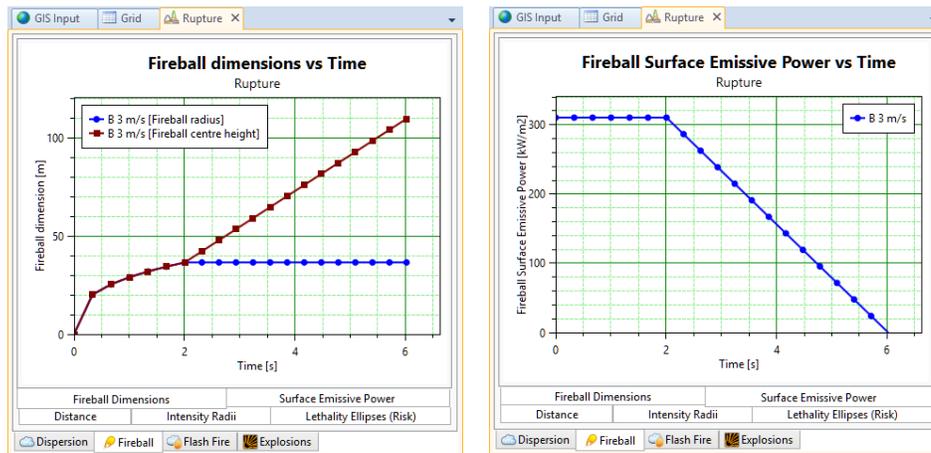
The **Fireball model** field in the dialog for a flammable Source Scenario and for a standalone Fireball Scenario allows you to choose between a range of correlations for calculating the size, duration and intensity of the fireball.



The list for this field now includes the *Martinsen time varying* correlation, as shown.

The *Martinsen time varying* option has been set as the default option for v8, which is a change from the previous default, which was the *Roberts/TNO hybrid* option referred to as the “Recommended” option in the previous version. If you open an analysis from a previous version that contains Scenarios that are set to use the default for the **Fireball model** field, these scenarios will now use the *Martinsen time varying* correlation, and the results for fireball effects will change.

As shown in the graphs below, the Martinsen time varying correlation models the fireball radius as increasing until the point when the fireball lifts off from the ground. The surface emissive power is constant to the moment of lift-off, and then decreases.



The results for the modelling are reported in a new Report called the **Dynamic Fireball** Report. The report that was called the **Fireball** Report in previous versions is now called the **Static Fireball** Report, and this gives the results for the three other fireball correlations.

The details of the calculations are given in the technical documentation supplied with the program.

## 1.5 Improved options for modelling expansion at the beginning of the release

In the Discharge parameters tab for a continuous storage Scenario, the **Atmospheric expansion method** field gives a range of options as shown for the modelling of expansion to atmospheric pressure.

The option *DNV GL recommended* is a new option that has been set as the default for v8. If this option is selected, the calculations will use the *Conservation of momentum* method for situations in which rainout will not occur, and the *Closest to initial conditions* method for other situations. Rainout will not occur if the material is carbon dioxide, or if multi-component modelling is selected for the Equipment item, or if the final liquid fraction predicted by the *Conservation of momentum* method is zero.

In the Near field tab of the Dispersion parameters, the **Modelling of instantaneous expansion** field gives a range of options as shown for the modelling of the expansion of a pressurised instantaneous release from the storage pressure down to atmospheric pressure.

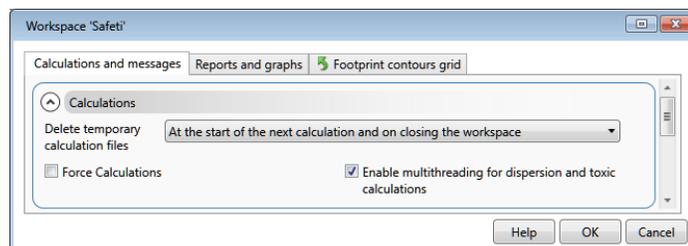
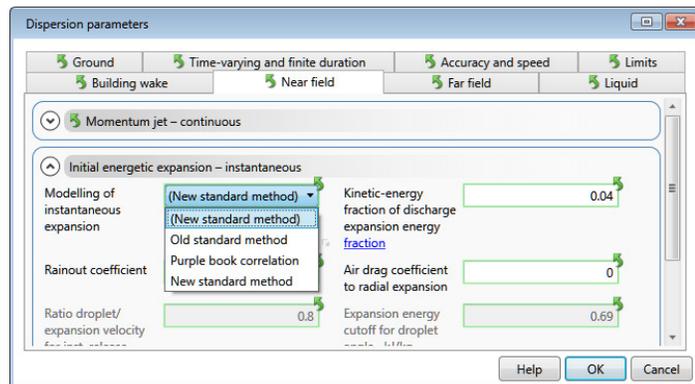
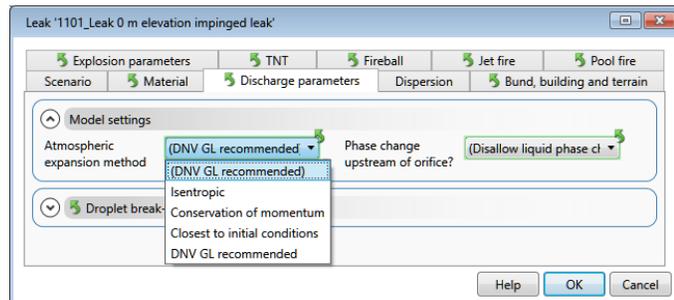
The option *New standard method* is a new option that has been set as the default for v8. In previous versions, the modelling of instantaneous expansion often underestimated rainout significantly, and the new option improves the modelling of two-phase and liquid releases considerably, incorporating radial expansion of liquid droplets within the release, and accounting for time-varying rainout.

The details of the calculations are given in the technical documentation supplied with the program.

## 1.6 Parallel processing available for dispersion and toxic calculations

The Calculations and messages tab of the Workspace dialog now includes the option **Enable multithreading for dispersion and toxic calculations** as shown.

If this option is checked, then these aspects of the consequence calculations for a given Scenario will be run simultaneously on different CPU cores, which has the potential to allow the calculations to run more quickly. If the option is not checked, the program will run all of the consequence calculations for a given Scenario on a single core, with no parallel processing for any aspects of the consequence calculations.



## 2 NEW FEATURES APPLICABLE TO USERS OF THE FULL PHAST PROGRAM

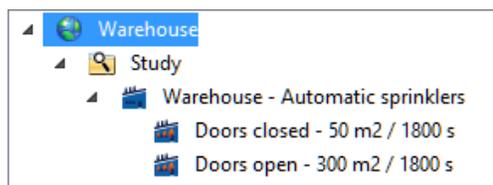
These features are not available in Phast Lite.

### 2.1 Warehouse Model for modelling toxic plumes from a warehouse fire

The Warehouse Model considers a fire in a warehouse, and models the effects of the fire as a toxic plume which contains a mixture of hydrogen chloride, nitrogen dioxide and sulfur dioxide. This Model is an updated version of the Warehouse Model that was present in versions of the program before v7.

The size and duration of the plume will depend on the amount and type of materials stored, on the ventilation for the warehouse, and on the scale of the fire, i.e. on the way in which the fire develops, and the effectiveness of the fire-fighting system.

To model a fire with a particular size and duration, you generate or define a Fire Scenario which has an associated probability or frequency; you can generate the PGS-15 Scenarios associated with the fire-fighting system for the warehouse, or you can define any number of user-defined Fire Scenarios for a particular warehouse. The illustration shows a Warehouse Model that has two Fire Scenario nodes defined.



The input data for the Warehouse Model is more complex than for most of the Equipment items and Scenarios in the program, since there are several stages in completing the input data, as described below.

#### *Stage 1: Defining Warehouse Materials*

When you define a Warehouse, you specify which materials are stored in the warehouse, and you do this by selecting them from a list of the Warehouse Materials that are defined in the program as part of the Materials data. This means that you should make sure that all of the Warehouse Materials for the warehouse are defined in the Materials tab before you complete the input data for the warehouse.

The input data for a Warehouse Material consists of the number of atoms of particular elements in the average composition of the Warehouse Material, and of the molecular weight of the material.

#### *Stage 2: Defining a Building type and a Building for the Warehouse*

You must insert a Building type icon under the Buildings folder in the Map tab of the Study Tree, and define the ventilation data for the warehouse.

You must then insert a Building under the Building type node, and define the location and dimensions of the Warehouse Building by placing it on the Map.

### *Stage 3: Defining the Warehouse Model*

There are four tab sections of input data in the dialog for a Warehouse Model:

- The **Warehouse materials** tab: You must set up a list of the Warehouse materials that are stored in the Warehouse, giving the mass stored and the active mass fraction for each material.
- The **Building** tab: you must select the Building that contains the Warehouse. The Warehouse might only occupy part of this Building, and you can specify whether this is the case, and give the area and height of the Warehouse within the Building.
- The **Fire scenarios** tab: A given Fire scenario represents one possible development for a fire in the warehouse. A fire may develop in different ways, with different consequences, depending on where and how the fire started, and on the effectiveness of the fire-fighting response. You use the Fire scenarios tab to supply the input data for the scenarios for the Warehouse Model.
- The **Dispersion** tab: This is the Dispersion tab that is common to all Source Scenarios, for specifying concentrations and distances of interest.

### *Optional Stage 4: Generating the Fire Scenarios*

After you have completed the input data for the Warehouse Model, you can generate the Fire scenario nodes by right-clicking on the Warehouse Model and selecting *Generate warehouse scenarios*; a node for each Scenario will be added underneath the Warehouse node as shown. You use the nodes to view the Graphs and Reports for a Fire scenario, and you can also use them to run the calculations for an individual Fire scenario.

### *Running the calculations and viewing the results*

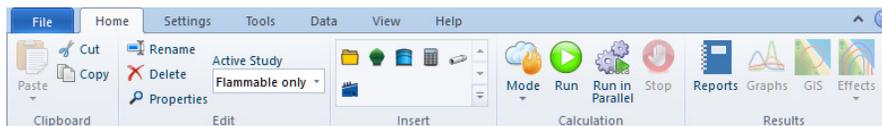
You do not have to generate the Fire scenarios yourself. If you have not already generated the nodes using the option in the right-click menu, the program will generate the nodes automatically when you run the calculations for the Warehouse Model, and it will then run the calculations for all of the Scenarios.

There is a Report called the **Warehouse Overview** Report that is specific to the Warehouse Model itself, giving details of the input data and of warehouse-level results. The Fire Scenarios have all of the Reports and Graphs applicable to any toxic Source Scenario, and also a **Warehouse Results** Report that gives input data for the Warehouse as a whole and for the Fire scenario, and details of the modelling of the generation of the toxic plume.

For further details of the input data and the results, enter "Warehouse" in the Index tab of the online Help. The details of the calculations are given in the technical documentation supplied with the program.

## 2.2 Run in Parallel option for running multiple Scenarios at the same time

The Home tab of the Ribbon Bar includes a *Run in Parallel* option as shown that uses parallel

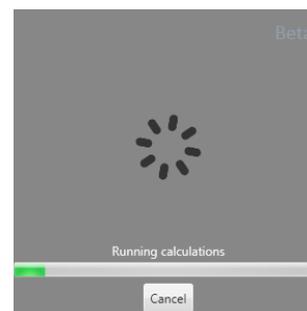


processing to run the calculations for multiple Scenarios at the same time

The *Run in Parallel* option will run the calculations for multiple Scenarios at the same time using parallel processing on multiple CPU cores. This parallel processing is performed by a special tool called the Console that allocates the processing between the different cores and combines the results when the calculations are complete. The parallel processing has the potential to run the calculations more quickly than the *Run* option, although in a small analysis the calculations might in fact run more slowly because of the additional time required for the Console to manage the processing between the different cores.

The option will run the calculations **for the entire workspace**, not just for the nodes that are currently selected in the Models tab of the Study Tree. It will run the calculations for all Scenarios, except for those that are set as excluded from the calculations and are greyed out in the Study Tree.

When you select the option, a Console Runner window as shown will appear in the centre of the program window. You will not be able to perform any actions in the program window while the Console Runner window is open, and you must either wait for the calculations to finish, or click on *Cancel* to stop the processing and close the Runner window.



The Console Runner window will initially say that it is **Preparing work** and will then move on to **Running calculations**, showing the status of the calculations with a progress bar. When the calculations are complete, the Console Runner will report that it is **Combining results**, and then it will close and you will be able to use the program window again.

Apart from these simple messages about the stage of the processing, the Console does not interact with the program window during the course of the calculations; there will be no messages about the calculations for individual Scenarios written to the Output View, and the display of the calculation-status of the Scenario nodes is not updated during the calculations. However, as soon as the Console Runner window closes, the Study Tree display will be updated, and you can view the full list of calculation messages for the Scenarios by right-clicking on the icon for any Equipment item or Scenario and selecting *Open parallel calculation log*, which will open a file of calculation messages in a separate Notepad window.

*Note:* The *Run in Parallel* option will run all of the calculations for a given Scenario on a single core, even if the option to **Enable multithreading for dispersion and toxic calculations** option is selected in the Calculations and messages tab of the Workspace dialog. The parallel processing performed by the *Run in Parallel* option involves running different Scenarios on different cores. To run different aspects of the calculations for a given Scenario on different cores, you must use the *Run* option with **Enable multithreading for dispersion and toxic calculations** selected.



*Note:* The **Beta** text in the Runner window and on the *Run in Parallel* icon shows that the parallel processing functionality is still under development. It has been included in the program as a "beta" feature with limited functionality, in the hope that users will experiment with it and will give feedback about their experiences and about their priorities for the next stages of the development. The feature has already received extensive testing during the development and the results for the calculations should be identical to those with the *Run* option, but the feature might sometimes be unstable (e.g. the Console Runner might stop working at some point in the processing), so it is advisable to save your workspace before using the *Run in Parallel* option.

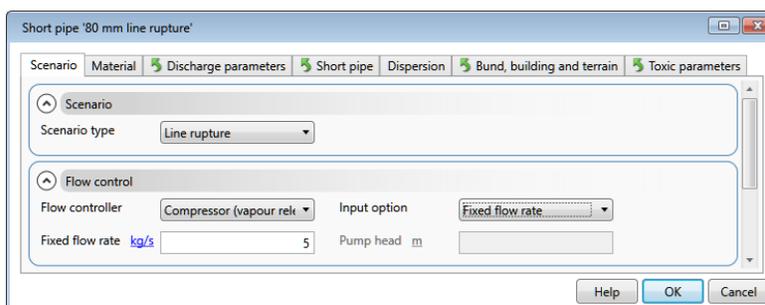
### 3 OTHER DIFFERENCES AND BUG FIXES

#### 3.1 More options for specifying process flow conditions for Short pipe Scenarios

In previous versions, you could give a value for the pump head for a liquid line rupture Scenario, but this was the only way of describing the process flow conditions.

There are now several options available the Short pipe Scenario, presented in the new **Flow control** group in the Scenario tab as shown.

The option to specify the **Flow controller** is available only if the **Scenario type** is set to *Line rupture*, and the details of the



options available depend on the storage conditions set for the Equipment and on the release phase set in the Material tab for the Scenario:

Storage Conditions	Phase to be released	Options available for Flow controller
Padded liquid or pressurised gas	Liquid	None Pump (liquid release) Control valve
	Vapour	None Compressor (vapour release) Control valve
Saturated liquid	Liquid	None Pump (liquid release)
	Vapour	None Compressor (vapour release)

If there is no flow through the pipe or you do not want to model the flow, you should set the **Flow controller** field to *None*.

If **Flow controller** is set to *Pump (liquid release)*, the **Input option** field will give a choice between *Fixed flow rate* and *Pump head*. If **Flow controller** is set to *Compressor (vapour release)* or *Control valve*, the only option for **Input option** will be *Fixed flow rate*.

When specifying the **Fixed flow rate**, you must give the flow rate through the pipe under operating conditions. If the flow controller is a pump or compressor, the program will calculate the upstream pressure that would give this value for the fixed flow rate, and the discharge calculations will use this pressure instead of the operating pressure that is set in the Equipment dialog. If the flow controller is a control valve, the program will use the operating pressure that is set in the Equipment dialog as the pressure upstream of the control valve, and calculate the pressure immediately downstream of the control valve that would give this value for the fixed flow rate, and the discharge calculations use this pressure. The Discharge Report gives the pressure that was used in the calculations.

For the **Pump head**, you must specify the equivalent head level for the energy added to the system by the pump. As in previous versions, the pressure used in the discharge calculations will be the sum of the operating pressure set in the Equipment item dialog, of the **Pump head**, and of the **Tank head**.

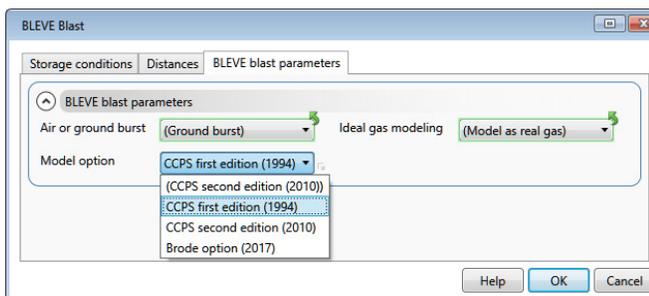
The details of the calculations are given in the technical documentation supplied with the program.

## 3.2 New option for modelling of a Standalone BLEVE Blast explosion

In the BLEVE blast parameters tab for a standalone BLEVE Blast Scenario, the **Model option** field now has a third option: *Brode option (2017)*. This option is not selected by default.

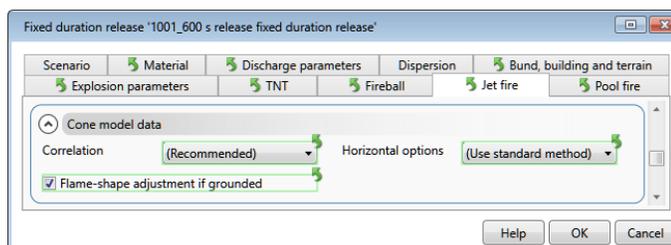
The Brode option uses the blast curves published in the CCPS second edition, but with scaled distances based on the Brode energy rather than on the isentropic energy.

The details of the calculations are given in the technical documentation supplied with the program.



## 3.3 New option for modelling jet flames that impinge on the ground

The **Cone model data** group is present in Jet fire tab for a flammable continuous source Scenario, as shown, and in the Jet fire parameters tab for a standalone Jet fire Scenario. This group now includes the **Flame-shape adjustment if grounded** option, which affects the calculations with the Cone jet fire model for flames that impinge on the ground.



If the option is checked, adjustments are made to the angle and/or elevation of the cone so that it does not impinge on the ground, and this adjusted flame is the flame used in radiation and effect calculations.

Despite the wording of the option, these adjustments do not in fact affect the flame shape, and the length and widths of the cone frustum are not changed.

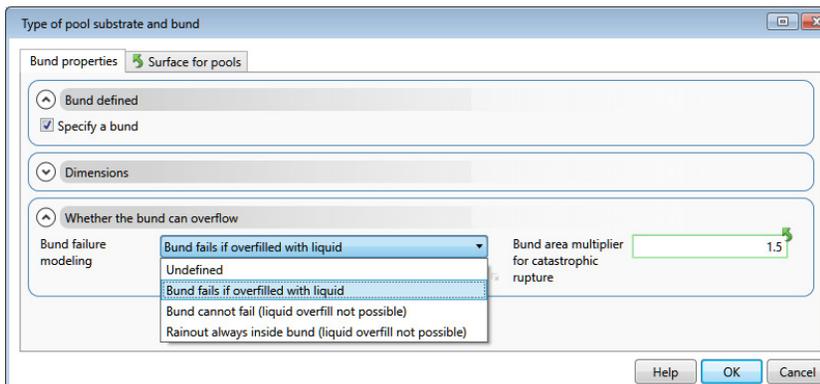
The details of the calculations are given in the technical documentation supplied with the program.

## 3.4 New options for modelling a bund

The node that was called the Bund type node in previous versions is now called the **Type of pool substrate and bund** node, reflecting the fact that it covers the definition of the surface on which a pool will form, whether or not a bund is present.

The Bund properties tab for this node has several new and changed fields as shown.

In previous versions, a bund would be modelled if you supplied non-zero values for the dimensions, but now you must check the **Specify a bund** field to state that the node covers the definition of a bund.

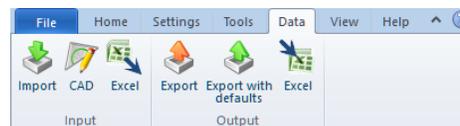


The list of options for **Bund failure modelling** has changed, with the addition of an option to force rainout to occur inside the bund, and with changes to the wording to make the effect clearer.

The **Bund area multiplier for catastrophic rupture** is a new field. For a Rupture Scenario under a Pressure Vessel, the calculations will apply this multiplier to the value for **Bund area (internal)** to obtain an effective bund area for use in the rainout and vaporisation calculations.

### 3.5 Materials can be exported and imported between workspaces

The Ribbon Bar now includes a Data tab as shown. This tab includes the *CAD* and *Excel* options that were previously on the Tools tab, and also some new *Import* and *Export* options that allow you to export Materials data to a separate file called a \*.psie file, and then to import the data from these files into another workspace. This gives an easy way of sharing Materials data between workspaces.



The *Import* option is enabled if the node selected in the Study Tree is the Physical Properties System node or the Materials node, and the *Export* options are enabled if the node selected is the Materials node or any node underneath the Materials node (e.g. a Component or a Materials Folder).

The *Export* option will export only the values for fields that have a non-default setting. This means that when you import the data, the fields that had default settings will take the default values for the current workspace, which may be different from those for the workspace from which you exported the data.

The *Export with defaults* option will export the values for all fields, including those that have default settings. This means that when you import the data, none of the fields will be left with their values unset, which means that none of the fields will be taking the default values for the current workspace.

### 3.6 Calculations for dispersion, flammable and toxic effects now all use the same height of interest

In previous versions, the Parameters had three different fields for specifying a height of interest for consequence results:

- **Toxics: height for calculation of effects** in the Toxic parameters, which was used for toxic effects.
- **Height for calculation of flammable effects** in the Flammable parameters, which was used for flash fire and radiation effects for source Scenarios.
- **Height for concentration output** in the General parameters, which was used in some dispersion graphs.

For simplicity and consistency, there is now a single field for specifying a height of interest for consequence results: the **Height of interest** field in the General parameters, which applies to toxic, flammable and concentration results.

When you open a file created in a previous version, the **Height of interest** will be set to the value for **Height for concentration output**. If the file you are upgrading contains a Parameter Set in which the toxic or flammable height is not set to the same value as the **Height of interest**, messages will be generated during the upgrade process saying that at least one Parameter Set will be affected by the fact that the toxic height or flammable height is no longer used, and that the change will give differences in results.

### 3.7 Building wake modelling now selected by default

In previous versions, the default value for the **Building wake effect** field in the Bund, building and terrain tab for a Source Equipment item was *None*, but this has been changed and the default is now *Roof/lee*. If you want to model in-building effects but not building wake effects, you must set the **Building wake effect** field to *None* after you have selected the **Release building**.

*Note:* if you upgrade a file from a previous version that includes Equipment items or Models with the option set to *None*, the **Building wake effect** field will be set to *None* on upgrade.

### 3.8 Baker-Strehlow-Tang explosions now involve entire cloud volume

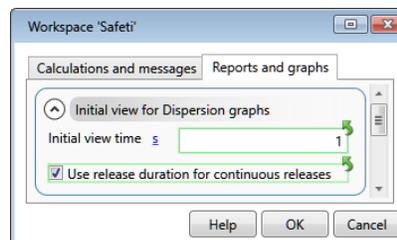
The Baker-Strehlow-Tang tab is present in the dialog for a flammable Source Scenario if the **Explosion method** is set to *Baker-Strehlow-Tang* in the Explosion parameters tab. It contains input values for use in the explosion calculations performed in the consequence calculations, which are separate from those performed in the risk calculations.

The tab contains a **Confined volume** field for defining the maximum volume of the confined region of the explosion. If the volume of the cloud is less than the value given for **Confined volume**, then the program will use the volume of the cloud in the calculations.

In previous versions, the **Confined volume** field had a default value of 1 m<sup>3</sup>, which was non-conservative. The default has been changed to zero, and with this value the entire volume of the cloud will be used in the calculations.

### 3.9 Simplification of options for initial view time for Dispersion Graphs

The Reports and graphs tab of the Workspace dialog contains the group **Initial view for Dispersion graphs**, as shown. These fields set the time used for the "initial view" of a cloud in the various Dispersion Graphs i.e. the default view that the program displays when it generates the Graph in the Graphs View.



In previous versions there were several different methods available for choosing the initial view time, but the options have now been simplified to the two fields shown.

For an instantaneous Scenario, the initial view will always be at the time set by the **Initial view time**.

For a continuous Scenario, the initial view depends on the setting for the option to **Use release duration for continuous releases**. If the option is selected, the initial view will be at the last moment of the release, but if the option is not selected, the initial view will be at the time set by the **Initial view time**.

### 3.10 Simplification of consequence reporting for explosion

In previous versions there were four reports for explosion results, with a separate report for early explosions for each explosion model, and a separate report for late explosions.

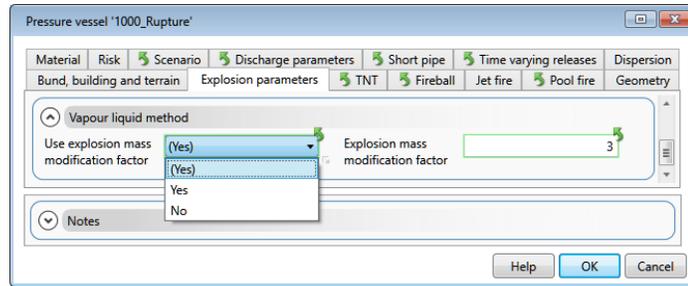
The distinction between early explosions and late explosions is no longer made in the program, and the results for the different explosion models are now all presented in the same form, which means that the reporting has been consolidated to give a single report called **Explosion** which covers all aspects of the explosion calculations performed in the consequence calculations.

### 3.11 Simplification of some options for explosions

The options available for two input fields have been simplified.

#### *Option for use of explosion mass modification factor*

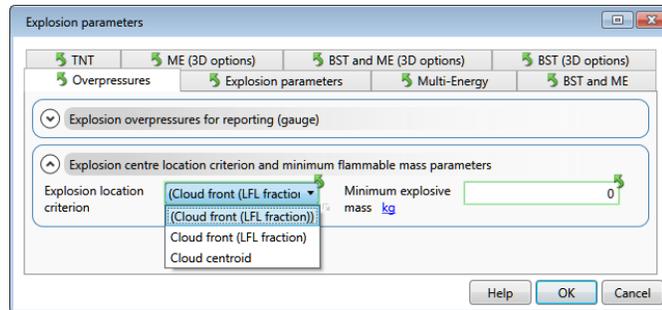
The Explosion parameters tab for a source Equipment item or Scenario includes the option **Use explosion mass modification factor** as shown. For a two-phase cloud, this factor is used in calculating the mass of the cloud that is involved in the explosion.



In previous versions this field gave the choice between using the mass modification factor for both early and late explosions and using it only for early explosions. The distinction between early and late explosions is no longer made in the program, and the option has been changed to a choice between Yes and No. If the option is set to No, then the explosion calculations will use the total flammable mass in the cloud, and if the option is set to Yes, then the explosion calculations will use a reduced explosive mass that depends on the vapour fraction at the time of the explosion.

#### *Option for location of explosion*

The Overpressures tab of the Explosion parameters dialog includes the **Explosion location criterion** field as shown.



In previous versions the list of options included *Cloud front (LFL)*, but this has been removed, leaving the two options shown.

### 3.12 Some Long pipeline inputs simplified and clarified

These changes have been made in two areas:

#### *Simplifying the Isolation option for the Location specific breach Scenario*

The Scenario tab for the Location specific breach Scenario includes an **Isolation** field. In previous versions this field was enabled if valves had been defined for the Long pipeline item, and you could choose between four isolation scenarios, depending on whether or not valves upstream and valves downstream operated successfully.

This field is now always disabled. If any valves are defined in the Long pipeline dialog, the field will set to *Full Isolation* and all valves will be modelled as operating successfully. If no valves are defined, the field will be set to *No Isolation*.

### Clarifying the definition of the size of a Breach Scenario

In the Scenario tab for the Location specific breach, the behaviour of the **Hole diameter** and **Relative breach aperture (area)** fields has been changed so that you can see more clearly the size of the outflow that will be modelled in the calculations.

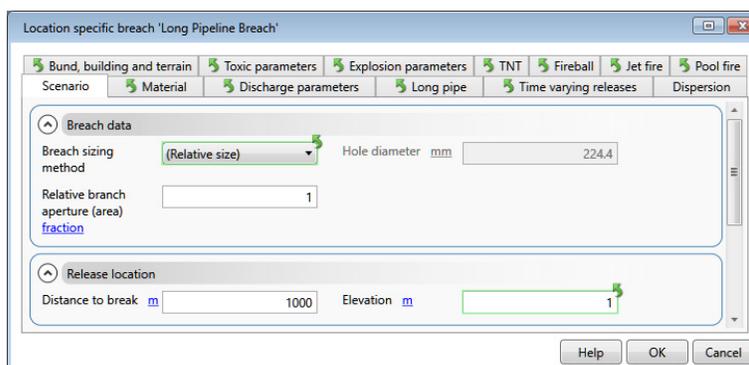
The size that you specify in the Scenario tab is the size of the outflow area that is to be modelled for the breach, and in deciding how to define the size, you must consider the **total potential outflow area** for the breach. For a given breach, the total potential outflow area will depend on the location of the breach along the length of the pipeline, as specified by the **Distance to break** field. If **Distance to break** is at the very beginning or the very end of the pipeline, the total potential outflow area will be equal to the cross-sectional area of the pipeline, whereas if the **Distance to break** is at any intermediate point along the length, the total potential outflow area will be twice the cross-sectional area of the pipeline, as outflow can occur from both the upstream and the downstream branches of the pipeline.

In the discharge calculations for a Breach Scenario, the program has always interpreted the outflow size in terms of the total potential outflow area, with its dependence of the location of the breach. This has not changed, but additional information is now given in the Scenario tab to show you how the outflow size that you have defined will be interpreted in the calculations. In the Breach data group in the Scenario tab, either the **Hole diameter** field or the **Relative breach aperture (area)** field will be disabled, depending on the setting for the Breach sizing method. In previous versions the field that was disabled would be blank, but the disabled field now displays a value that gives you a measure of the total potential outflow area for the breach Scenario.

If the **Hole diameter** field is disabled, it will display the hole diameter that corresponds with the current setting for the **Relative branch aperture (area)**. The Scenario shown is a 100% breach at an intermediate position in a pipeline with a diameter of 158.7 mm. For this situation the total potential outflow area is twice the cross-sectional area of the pipeline,

and the **Hole diameter** field is displaying a value of 224.4 mm because a circular hole with this diameter has an area twice that of the pipeline cross-section. If the breach were at the end of the pipeline, the **Hole diameter** field would show a value of 158.7 mm.

If the **Relative branch aperture (area)** field is disabled, it will display the relative aperture that corresponds with the current setting for the **Hole diameter**. For a pipeline size of 158.7 mm and an intermediate breach location, if the **Hole diameter** were enabled and set to 158.7 mm, the **Relative branch aperture** would be showing a value of 0.5, i.e. half the total potential outflow area.



### 3.13 Bug fixes

The following bugs have been fixed in v8:

**1** *B-13586* *Issues with reporting of Material to Track*

Description Some reports of toxic and flammable effects included the value set for the material to track, which could be taken as implying that the results in the reports were based on the concentrations for that material. This is incorrect, as these results are always based on the concentration for the whole mixture.

The value for material to track reported was the value set at the Equipment level, not the value set at the Scenario level.

**2** *D-10757* *Pool fire not modelled for a source Scenario when jet axis impinges on ground*

Description If the axis of the jet flame impinges on the ground, the jet fire calculations will fail and not produce results, and in this situation the program used to omit the modelling of pool fires, even though this modelling should not have been affected by the jet flame impingement.

*Note:* you can prevent the jet fire calculations from failing because of ground impingement if you make sure that the new **Flame-shape adjustment if grounded** option is checked. With this option, the position of the flame will be adjusted so that the grounding does not occur, and the jet fire calculations will produce results.

**3** *D-11050* *Negative times in vapour discharge calculations for long pipeline*

Description In some situations, the discharge calculations for a long vapour pipeline with valves would produce results with negative times and give an error.

**4** *D-11066* *Improved warning messages for Toxic Dose footprint graphs*

Description When you view the graphs for a toxic Scenario, the initial levels used for the toxic footprint graphs are those set in the Toxic parameters tab for the Scenario. For probit and lethality, the default values for the levels cover the likely range of results of interest, but for dose, the default values may be much lower than the levels calculated for the Scenario because the toxic lethality for the Scenario had dropped below the minimum level of interest when the dose level was still above the maximum default level. In this situation the dose footprint graphs will initially be blank, and a warning message about the lack of results will be written to the Output View.

The warning message used to suggest reducing the minimum probability of death in order to see dose footprint results. This would not be an efficient approach, as it would be much easier to use the Edit Settings dialog in the Consequence tab of the Ribbon Bar to increase the values for the dose levels that you want to plot for the current Graph View. The warning message has been changed to be more helpful, and now suggests increasing the target dose value.

**5** *D-11734*      *Negative concentrations in dispersion results*

Description      For a short-duration time-varying Scenario modelled with multiple rates, the Dispersion Report could sometimes show negative values for concentration.

## 4 ALERTS AND WORKAROUNDS

There are some known bugs that have not been fixed in v8, and some aspects of the program that may give behaviour that you do not expect, e.g. because of minor enhancements or minor changes in the modelling that may produce slight differences in the results.

For the most up-to-date list of known bugs and other issues, you can access the *Customer Portal* using the link in the Help tab of the Ribbon Bar. To log into the portal you must enter a valid user name and password, which you can obtain from your local DNV GL Software office.

<b>1</b>	<b>D-10289</b>	<b><i>Ruptures with rainout can leave gaps in the cloud</i></b>
Description	An instantaneous release with rainout and pool vaporization can sometimes produce dispersion results with a large gap between the cloud from the initial release and the cloud from the pool vaporization, e.g. the Footprint graph may show the instantaneous cloud disappearing at, say, 200 s, and the cloud from the pool vaporization not starting until 400 s.  For an instantaneous release, the dispersion calculations do not release observers for pool vaporization until the cloud from the initial instantaneous release is no longer over the pool. In some situations, the criteria that the calculations use to define the edge of the instantaneous cloud are lower than the concentrations displayed in the dispersion graphs, so the calculations regard the cloud as being over the pool even though the graphs suggest that it has moved far from the pool.	

<b>2</b>	<b>D-11192</b>	<b><i>Scalloping in Max Footprint graph, depending on grid spacing</i></b>
Description	In some situations, the Max Footprint Dispersion Graph may show “scalloping”, i.e. a “rippling” effect along the shape of the contours. The appearance of this effect depends on the settings for the Concentration grid definition fields in the Dispersion Parameters, and is most likely to happen when grid resolution is high.	

<b>3</b>	<b>D-11248</b>	<b><i>Bund modelling not conserving mass</i></b>
Description	In some situations the mass vaporized from the pool can be greater than the total mass released.	

<b>4</b>	<i>D-11540</i>	<i>Upgraded standalone pool vaporization Models from v6 not included in any Model Selection</i>
	Description	<p>If you upgrade a v6.54 or v6.7 file that includes a standalone Pool Vaporization Model, the Model will be upgraded as a Spill Scenario under an AST Equipment item, but it will be removed from the Model Selection which means that it will not be included when you run the calculations, no matter which Run Row is selected.</p> <p>The upgrade is handled in this way because the standalone Pool Vaporization Model can not be used in the risk calculations, but the Spill Scenario can. The Spill Scenario will be incomplete on upgrade because there is no event frequency specified, and it is removed from the Model Selection so that it does not cause problems with the risk calculations.</p> <p>You should decide whether or not you want to include the Scenario in the risk calculations. If you decide not to include it but you do want to be able to run the consequence calculations for it, you should create a separate Model Selection and Run Row for it, and only run the consequences calculations for that Run Row.</p>

<b>5</b>	<i>D-11858</i>	<i>Problem on changing Material when Material to Track is set</i>
	Description	<p>If a mixture was selected for an Equipment item, and one of the components of the mixture was selected as the material to track for the Equipment or a Scenario, this component will remain selected if you change the selection of Material to something that does not include the selected component. The Equipment and Scenario will not be shown as having an error in the input data, but the Scenario will give errors in the dispersion calculations about the chosen component not being present in the mixture.</p>

<b>6</b>	<i>D-12104</i>	<i>Dispersion Graphs are always present for time-varying Scenarios even if only the discharge calculations have been run</i>
	Description	<p>If you set the <b>Mode</b> to <i>Discharge</i> in the Home tab of the Ribbon Bar, run the calculations for a time-varying Scenario or a Long pipeline breach Scenario and then view the Graphs, you will find that the full set of Dispersion Graphs are present, though all of the graphs are blank except for the <b>Mass Rate</b> Graph.</p>

<b>7</b>	<i>D-12221</i>	<i>Reports flagged out of date when you change the averaging time for the graph</i>
	Description	<p>If you have both Reports and Graphs open for a Scenario, and you use the Edit Settings dialog for the Graphs to change the selection of averaging time, you will find that the Reports are shown as being out of date, even though the change to the averaging time is only relevant to the Graphs.</p>

**8** *D-12286* *Some functions are disabled for a Warehouse Model created by copying and pasting*

- Description
- If you create a Warehouse Model by copying and pasting another Warehouse Model, some functions will not work for the second Warehouse Model:
- If the Warehouse Model is in a folder and you right-click on the folder and select *Generate warehouse scenarios*, Fire Scenarios will not be generated for the second Warehouse Model. However, if you right-click on the second Warehouse Model itself and select *Generate warehouse scenarios*, the Fire Scenarios will be generated.
  - The *Run* option will be disabled for the second Warehouse Model, although it will be enabled for the individual Fire Scenarios under the Model.

**9** *D-12362* *Jet Fire Contour graph for a standalone Jet Fire Scenario shown as out of date after checking option to Show Flame Shape*

- Description
- If the option to plot side view contours is selected for a standalone Jet Fire Scenario, the option to **Show Flame Shape** will be enabled in the Consequence tab of the Ribbon Bar when the Jet Fire Contour Graph is selected. If you check this option, the Graphs will be shown incorrectly as being out of date.

**10** *D-12390* *Time-varying Scenarios not shown in the Study Tree as run successfully when the Calculation Mode is set to Discharge*

- Description
- If the Calculation **Mode** is set to *Discharge* in the Home tab of the Ribbon Bar and you run the calculations for a set of source Scenarios, the Scenarios whose discharge calculations run successfully should be shown with green ticks in the Study Tree. These green ticks are shown for most types of source Scenario, but not for Time-varying leak Scenarios or Time-varying short pipe Scenarios.

**15** *D-12431*      *A User-defined Source Scenario generated from a Scenario with multi-rate time-varying discharge results will give Jet Fire results that are different from those for the original Scenario*

Description      The right-click menu for a source Scenario that has results present will include the option to generate a User-defined Source Scenario. The discharge results for the selected Scenario will be used as the definition of the release observers for the User-defined Source Scenario.

If the original Scenario is a Time-varying leak Scenario, a Time-varying short pipe Scenario or a Location specific breach Scenario, and the **Method for calculating average rate** is set to *Up to 10 rates* in the Time varying releases tab section, then if you compare the results between the original Scenario and the generated User-defined Source Scenario, you will find that the results for dispersion, toxic results and delayed flammable effects are identical, but that the jet fire results are different. For the original Scenario, the jet fire calculations use discharge conditions based on the full, detailed discharge results, whereas for the generated User-defined Source Scenario, the calculations use discharge conditions based on the list of release observers in the Scenario tab section. These discharge conditions can be quite different between the two Scenarios, giving different jet fire results, especially if the release rate changes rapidly at the start of the release. Typically, the calculations for the User-defined Source will use a higher discharge rate and give a larger jet fire with greater effect distances.

When you generate a User-defined Source Scenario from a Scenario that has **Method for calculating average rate** set to *Up to 10 rates*, a warning message about the differences in jet fire results will be written to the Output View.

**16** *D-12478*      *Results for a User-defined Source Scenario are not cleared on changing the input data*

Description      If you run the calculations for a User-defined Source Scenario and then change the input data for the Scenario, the results will not be cleared and the Scenario will be shown with a green tick in the Study Tree, showing that it has results that the program regards as up to date. To be able to rerun the calculations for the Scenario with the changed input data, you must right-click on the Scenario and select *Clear results*.



## **ABOUT DNV GL**

Driven by our purpose of safeguarding life, property and the environment, DNV GL enables organizations to advance the safety and sustainability of their business. We provide classification and technical assurance along with software and independent expert advisory services to the maritime, oil and gas, and energy industries. We also provide certification services to customers across a wide range of industries. Operating in more than 100 countries, our 16,000 professionals are dedicated to helping our customers make the world safer, smarter and greener.

## **SOFTWARE**

DNV GL is the world-leading provider of software for a safer, smarter and greener future in the energy, process and maritime industries. Our solutions support a variety of business critical activities including design and engineering, risk assessment, asset integrity and optimization, QHSE, and ship management. Our worldwide presence facilitates a strong customer focus and efficient sharing of industry best practice and standards.