SPARK3D

User Manual

1 Table of Contents

2 SPARK3D Online Help

The SPARK3D help system is organized into the following main topics:

Introduction	What is SPARK3D.
Tutorial	Guided tour of SPARK3D features. Recommended for new users.
Manual	Using SPARK3D - reference manual.

2.1 SPARK3D Introduction

The objective of this introduction is to explain the motivations behind SPARK3D development and the target of SPARK3D software, as well as the approach and basic concepts used by SPARK3D.
The introduction contains the following topics:

Objective	The objective of SPARK3D.
Features	Main Features in SPARK3D.
Limitations	Some limitations of SPARK3D

Objective

SPARK3D is a general software tool for Radio Frequency (RF) breakdown analysis. It is based on powerful and accurate numeric algorithms for predicting both Corona (arcing) and Multipactor breakdown onsets, which are two of the main high power effects that can severely damage a device. In this context, it is the final objective of this software tool to help the microwave components designing/manufacturing industries to decrease both the time to market and the development costs for the next generation of communication systems.

Features

SPARK3D is an efficient software tool for the accurate analysis of high power effects in RF structures. It imports the electromagnetic field computed with some of the most widespread electromagnetic simulation software tools like:

- FEST3D®
- ANSYS® HFSS™ v. 11 (or higher)
- CST® 2012 (or higher)
- FEKO suite 6.2 (or higher)

Besides, SPARK3D, is also able to incorporate arbitrary external DC fields to the simulation, either Electric or Magnetic, computed with ANSYS® MAXWELL™, or by importing rectangular CSV format mesh files.

SPARK3D offers a great versatility and is the first commercial software capable to compare high power results using different electromagnetic kernels.

Multipactor analysis

The Multipactor module is based on a full 3D electron tracker that employs a Leap-Frog algorithm for the path integration and the Vaughan model for SEY characterization of materials. This technique allows the analysis of Multipactor in complicated structures which involve arbitrary shapes in short computational times.

Corona analysis

Corona module is based on a numeric algorithm that uses an adapted FEM technique to solve the free electron density continuity equation. This technique allows the analysis of Corona in complicated structures which involve arbitrary shapes in short computational times.

Limitations

There are few limitations when importing fields:

- Only Continuous Wave (CW) single-carrier operation is supported.
- SPARK3D does not have information on the kind of material of the imported mesh points and will take everything (besides boundaries) as air/vacuum. Therefore the following considerations must be done:
  - **Importing from HFSS™:** The user must take care of exporting only the air/vacuum region of the device.
  - **Importing from CST®** (version 2012): CST® does not allow to export separately different volumes of the solution. Therefore, circuits with dielectrics should be avoided since they are not correctly interpreted by

SPARK3D.

- First order basis functions for HFSS™ and second order basis functions for CST® are supported.

2.2 SPARK3D Tutorial

The goal of the tutorials is to show you how to use the basic features of SPARK3D to analyze the high power characteristics and compute the threshold breakdown power of a RF device.

The first tutorial is a basic guide that illustrates how to export the EM fields from the electromagnetic simulation software tools compatible with SPARK3D to the format required by it. Tutorial 1 is provided to familiarize you with the SPARK3D graphical user interface and to show you how to run a basic simulation. Tutorial 2 shows how to define regions of analysis where corona and multipactor simulations are carried out. Tutorial 3 treats the analysis of results which are given both in tabular and graphic forms.

To learn the basic features of SPARK3D, you are recommended to work through tutorials in the order they are presented. It is also essential to play around with the list of examples provided to you during the installation in the folder "Examples".

- 0. EM field exportation gives some tips about how to export the required EM fields from the different electromagnetic kernels compatible with SPARK3D.
- 1. The First Run is a step by step guide to show you how to configure and execute a basic simulation of Corona and Multipactor.
- 2. Specifying Regions describes you how to define regions of interest where the analysis of Corona or Multipactor can be focused.
- 3. Analysis of Results shows you how to interpret and visualize the output data given both in tabular and graphic forms.

2.2.1 EM field exportation

In order to carry out the high power analysis of a device with SPARK3D, first of all the electromagnetic fields must be computed for the frequency under study and exported in the format required by SPARK3D from one of the compatible electromagnetic software tools:

- FEST3D ®
- ANSYS ® HFSS ™ v. 11 (or higher)
- CST ® 2012 (or higher)
- FEKO suite 6.2 (or higher)

Besides, SPARK3D is also capable to import arbitrary DC fields computed with external software in the following formats.

- ANSYS ® MAXWELL ™
- Structured rectangular CSV mesh

In this tutorial, you will learn how to export the EM fields once they have been computed for a single frequency from the different EM software tools.

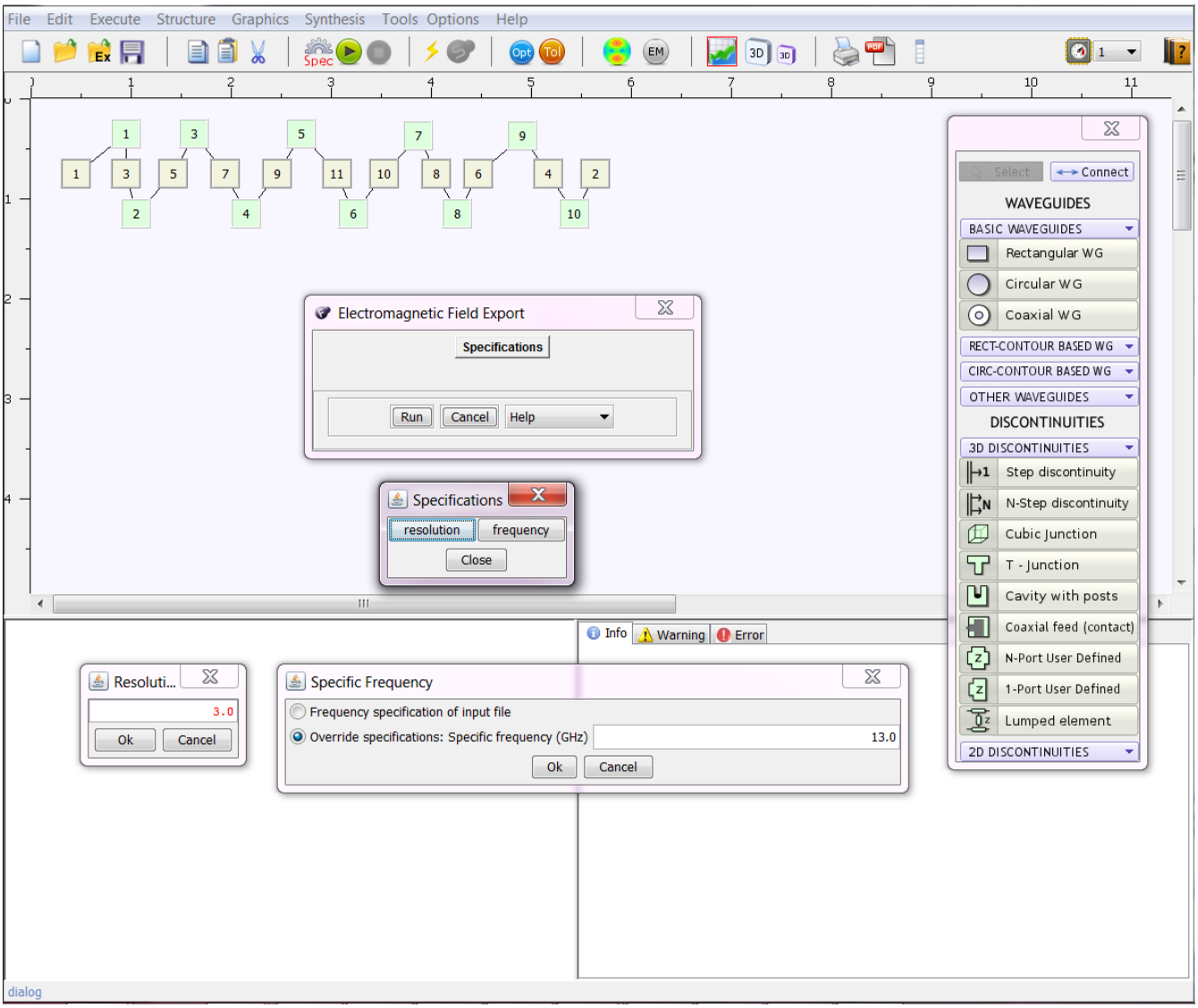
Please refer to the Introduction for more information on features and limitations.

FEST3D ®

It is quite straightforward to export the EM field from FEST3D to SPARK3D input format.

In the menu bar of FEST3D, click on **Execute -> Export fields to SPARK3D** to open the exportation window. The specifications window will appear so that you must set:

- the frequency where the EM field will be computed and
- the spatial resolution that will be used to mesh the device (by default, it is expressed in mm).



Click on **Run** button and the EM field will be exported in the proper format in a directory, whose name is the same as the .fest3 file. The recently created file has the extension .mfe and also has the same name as the .fest3 file. It will be used as input data in SPARK3D.

ANSYS ® HFSS ™ v. 13

Requirements

In order to export the EM fields from HFSS™ to SPARK3D format you must take into account that:

- The EM field has to be saved for the frequency under study,
- The mesh used for the simulation has to be of first order (that is, in the menu bar of HFSS™ **Solution Setup->Options->Solution Options->Order of basis functions** must be set to First Order).

> ⚠ It is important to point out that the fields should be given in peak values and correspond to an average input power of 1 W. So, the sum of power in all excited ports must be equal to 1W.

Procedure

Once the electromagnetic response of the structure has been simulated and the EM fields have been computed for the frequency under study, you must follow the following steps:

1. In the menu bar of HFSS™, go to **Tools-> Run script** and select the script named **ExportToSpark3D.vbs**, which has been distributed together with SPARK3D software. You can find it in the folder of SPARK3D installation, typically:

    a. SPARK3D standalone: "C:\Progam Files(x86)\SPARK3D\dist\HFSSexportscript").
    b. SPARK3D with FEST3D: "C:\Program Files(x86)\FEST3Dx.y\bin\external

\spark3d\dist\HFSSexportscript".

- Through this script, the following variables will be created:
    a. Real_vector_E
    b. Imag_vector_E
    c. Real_vector_H
    d. Imag_vector_H

which will be used as input data in SPARK3D.

2. Select from the Solids of HFSS model the ones corresponding to vacuum. Right-click on them and select the option **Plot Fields->Named Expressions**. From the displayed options, you must plot all the previously created variables, described in step 1. It is mandatory to use the variables created through the script.
3. Select the proper solution corresponding to one where the EM fields have been saved for the frequency under study.
4. In the Project Manager, go to **Field Overlays**, right click on it and select the option **Save As.** Choose all the four Named Expressions presented before. They will be saved on a unique file of extension .dsp, that will be used as input data in SPARK3D.

Limitations

- SPARK3D does not have information on the kind of material of the imported mesh points and will take everything (besides boundaries) as air/vacuum. This is the reason why it is mandatory to export the EM fields corresponding to vacuum from the HFSS™ model.
- SPARK3D is not prepared to accept EM fields from HFSS projects considering symmetries. If you have an already existing project which was simulated taking into account symmetries, it is mandatory that you compute the EM fields again disregarding the symmetries. Make sure that in **Project Manager-> Excitations**, the parameter **Port Impedance Multiplier** is set to 1.
- When using a very dense mesh in the HFSS™ solution or when the problem is quite large, there could be some memory problems when importing the fields. In this case, it could be necessary to divide the geometry of the problem in several pieces/solids and work with the EM fields exported from each one.

CST ® 2012

The exportation of the EM fields from CST ® to SPARK3D is done as follows:

In the menu bar of CST ®, click on **Macros->Solver->F-solver->Export Fields to SPARK3D** to open the exportation window. You must specify the frequency under study and the directory where the created file will be saved. The extension of this file is .f3e and it will be used as input data of SPARK3D.

Limitations

- SPARK3D does not have information on the kind of material of the imported mesh points and will take everything (besides boundaries) as air/vacuum. CST ® does not allow to export separately different volumes of the solution. Therefore, circuits with dielectrics should be avoided since they are not correctly interpreted by SPARK3D.

FEKO ® 6.2

FEKO allows for automatically write simulation results to a SPARK3D format . fse file. This is done as follows (please refer to the FEKO manual for more detailed instructions) :

- Define your model in FEKO normally specifying the geometry input/ouput ports and excitations.
- A dielectric with vacuum properties must be defined within the "Media". Such a material must be assigned to at least one volume in the model. Other media can be defined and assigned to other volumes in the model but only those defined as vacuum will be imported as analysis regions in SPARK3D. Make sure that in such vacuum volumes the solution is set to FEM or MoM with Volume Equivalence Principle (VEP). Otherwise they will not be exported.
- Create a "standard configuration" and set the operation frequency to "single-frequency". Enter the desired frequency for the field calculation.
- Mesh the model. Use adaptive meshing or specify a characteristic length.
- Add a "Near fields" request. Set the "Definition" methods in the "Position" tab to "Tetrahedral mesh". In the "Advanced" tab select "Fields" type and check the "Electric fields", "Magnetic fields" and "Export fields to SPARK3D *. fse file" boxes.
- The mesh must be set to a second-order tetrahedral mesh. This is a default option in FEKO. However you can make sure by setting the "element order" to "second order" in the "FEM" tab of "Solver/run" -> "solver settings".
- Solve the structure. If correctly defined, the simulation should produce a . fse file in the output folder

Limitations

Export to near-field from FEKO to a SPARK3D file (i.e. near-field at points defined by a tetrahedral mesh) is only supported for dielectric volume regions with solution method set to FEM or VEP. Currently all dielectric volumes in a FEKO model must have the same solution methods setting (e.g. FEM regions are not supported together with a SEP or VEP regions in the same model).

Although FEKO writes boundary material information to the SPARK3D file, the current version of SPARK3D considers only vacuum regions for the analysis, and sets the same SEY curve for all boundaries (in multipactor simulations).

DC Fields

ANSYS® MAXWELL™

Once the magnetic or electric DC fields have been computed, you must follow the following steps:

1. Select from the Solids of MAXWELL model the ones corresponding to vacuum. Right-click on them and select the option **Fields->B->B_Vector**, for magnetic field, and **Fields->E->E_Vector**, for electric field.
2. Select the proper solution corresponding to one where the DC fields have been saved.
3. In the Project Manager, go to **Field Overlays**, right click on it and select the option **Save As.** Choose the B_Vector (or E_Vector) box. It will be saved on a unique file of extension .dsp, that will be used as input data in SPARK3D.

CSV format

CSV (comma-separated-values) format files are text files with . csv extension that consist on tabulated data. SPARK3D can import DC fields which are saved in CSV format files, whenever the mesh is rectangular, structured and based on regular hexahedra.

Next, we describe the specific format that the CSV data should have in order to be imported by SPARK3D. Columns should be separated by one of the following characters: space, tab, comma, semi-colon. Each column represents a magnitude. Rows are separated by newlines. The format follows a 6-column scheme: x y z F_{DCx} F_{DCy} F_{DCz} , being x, y, z the coordinates of each node in the mesh and F_{DC} the values of the DC field.

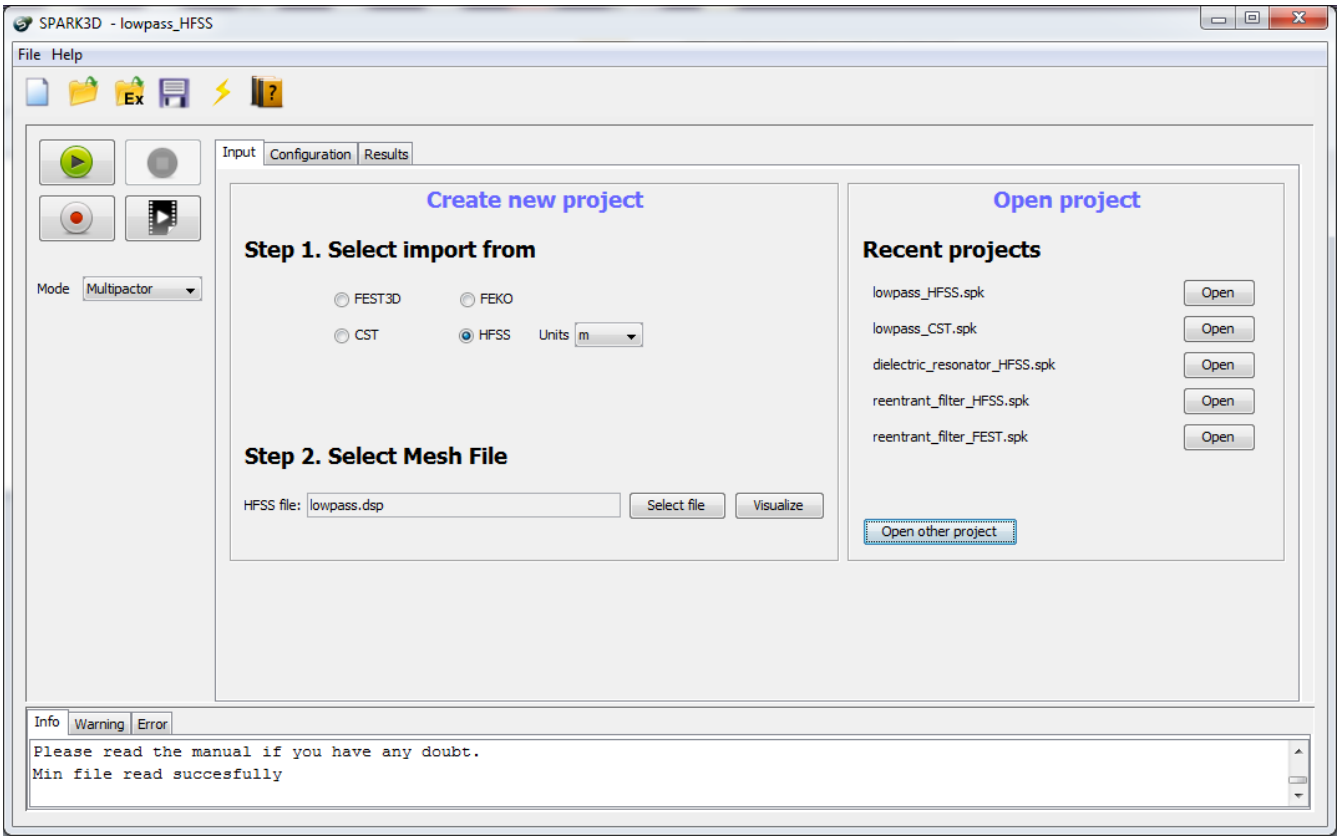
2.2.2 First Run

In this tutorial you will learn how to run your first simulation with SPARK3D. It presents a guided example where the whole high power analysis process is explained step-by-step. It is divided in 3 parts:

1. **Preliminaries.** We open an example and discuss how to import and visualize the EM field from a file.
2. **Running Corona mode.** The main parameters are set for Corona analysis and the simulation is launched. An overview of the Corona output is given.
3. **Running Multipactor mode.** The main parameters are set for Multipactor analysis and the simulation is launched. An overview of the Multipactor output is given.

Preliminaries

First, the EM field data of the device under study must be loaded. From the input tab, we can either create a new project or open an existing one, where the EM field is automatically loaded.



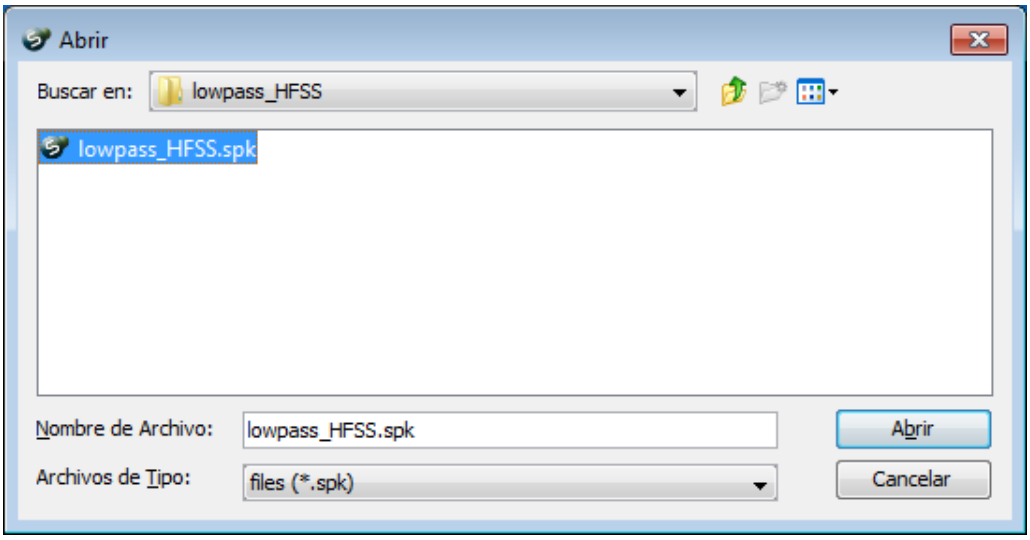
To create a new project, we follow the steps indicated by the GUI.

Step 1: Depending on the electromagnetic kernel previously used to compute the electromagnetic field distribution inside the component (see EM field exportation tutorial), the user imports the field using one of the different formats supported by SPARK3D, which includes:

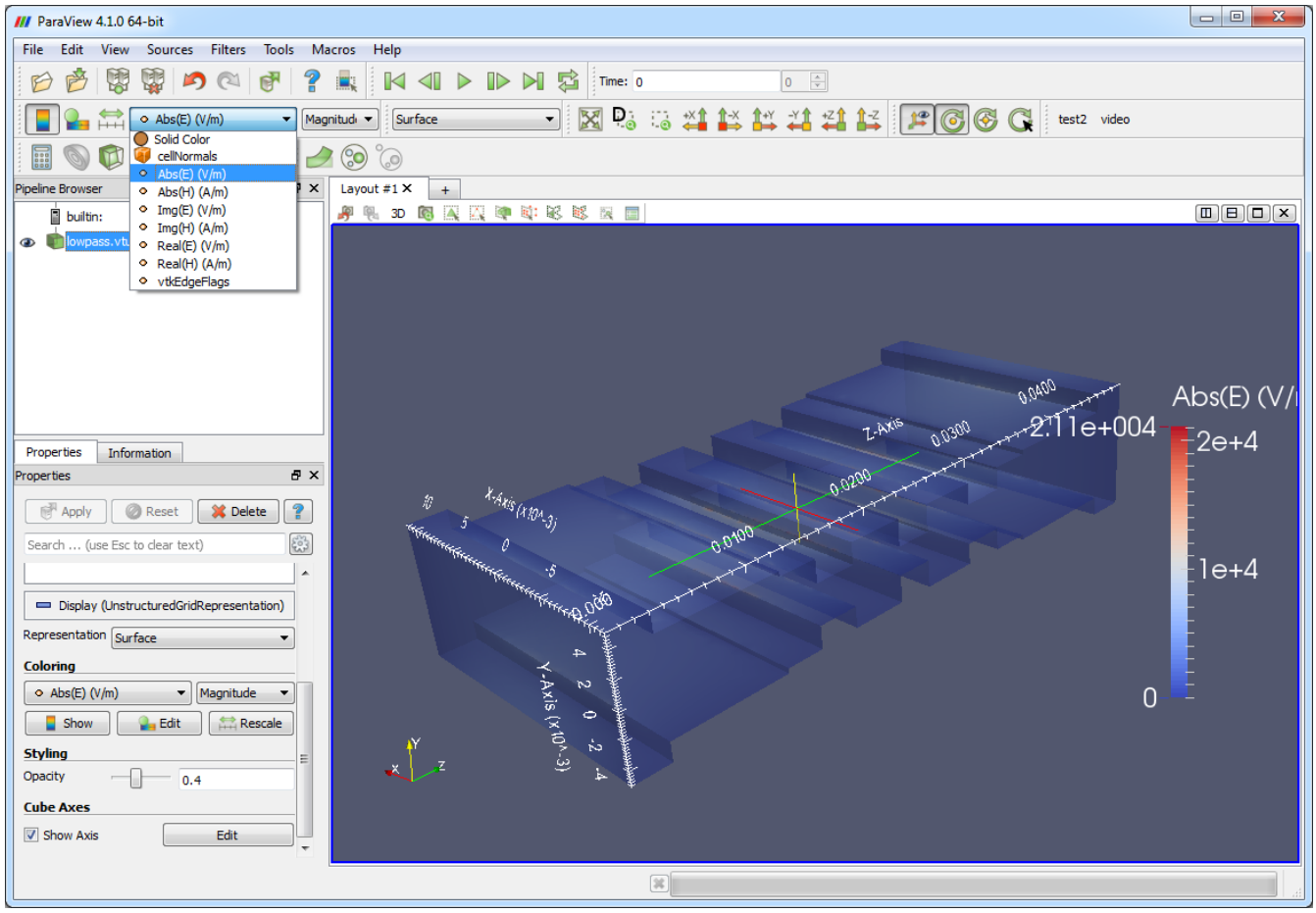
- FEST3D,
- CST,
- HFSS,
- FEKO.

Step 2: We select a mesh file containing the EM field of the device. With the **Select file** button, you can browse to access the appropriate file, whose extension will change depending on the type of EM kernel used to create it.

In this tutorial, we will load an existing example. Click on the examples folder and select Lowpass directory, where you choose **lowpass_HFSS.spk**.

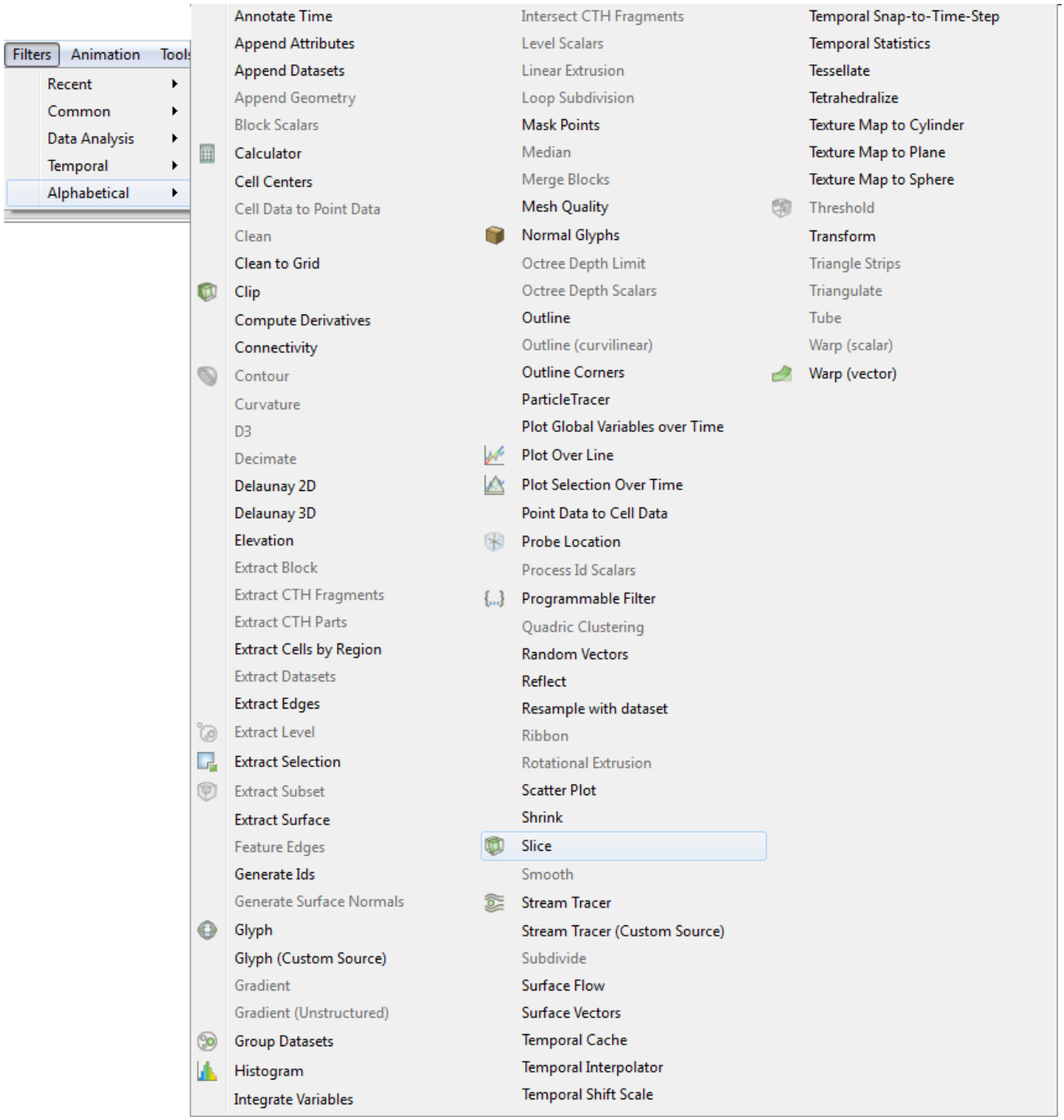


Once the mesh file has been loaded, it is advisable to visualize the EM field through the 3D CAD viewer included with SPARK3D distribution, Paraview. Click on the **Visualize** button and the main window of Paraview will open with the EM field previously computed inside the device, which looks like:

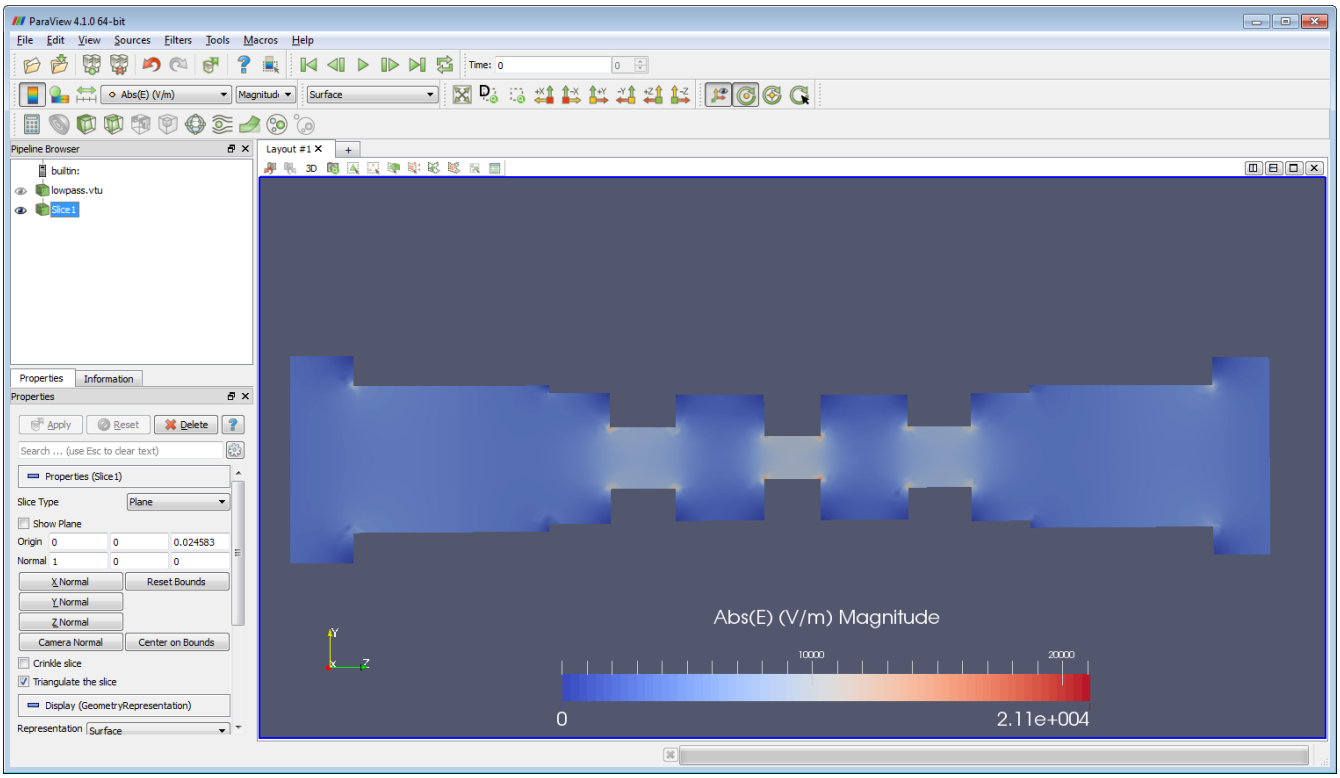


With the left, right and center button of your mouse you can rotate, zoom and translate the camera view. In the menu bar there is a display list where the different fields (magnitude, real and imaginary parts of electric and magnetic fields) can be selected.

2D cuts allow you to visualize the fields inside the structure, so that you can detect the potential areas of the structure where the breakdown onset is more likely to occur, that is, where the electric field is maximum. You can create a 2D cut with the slice button, that you can access from the menu bar like this:



In the figure below we see that the waveguides located in the center of the device are the main candidates for breakdown onset. This information will be helpful when defining regions of study (see Tutorial Specifying Regions).



Computing voltage with Paraview

With Paraview it is also possible to compute the voltage as the integration of the electrical field between two points in the mesh. In SPARK3D, the fields are defined for an input power of 1W, therefore the computed voltage is also at 1W, called V1W. This can be useful for multipactor to translate from breakdown power to breakdown voltage and compare results with theoretical parallel-plate predictions. The expression to convert from power to voltage is the following:

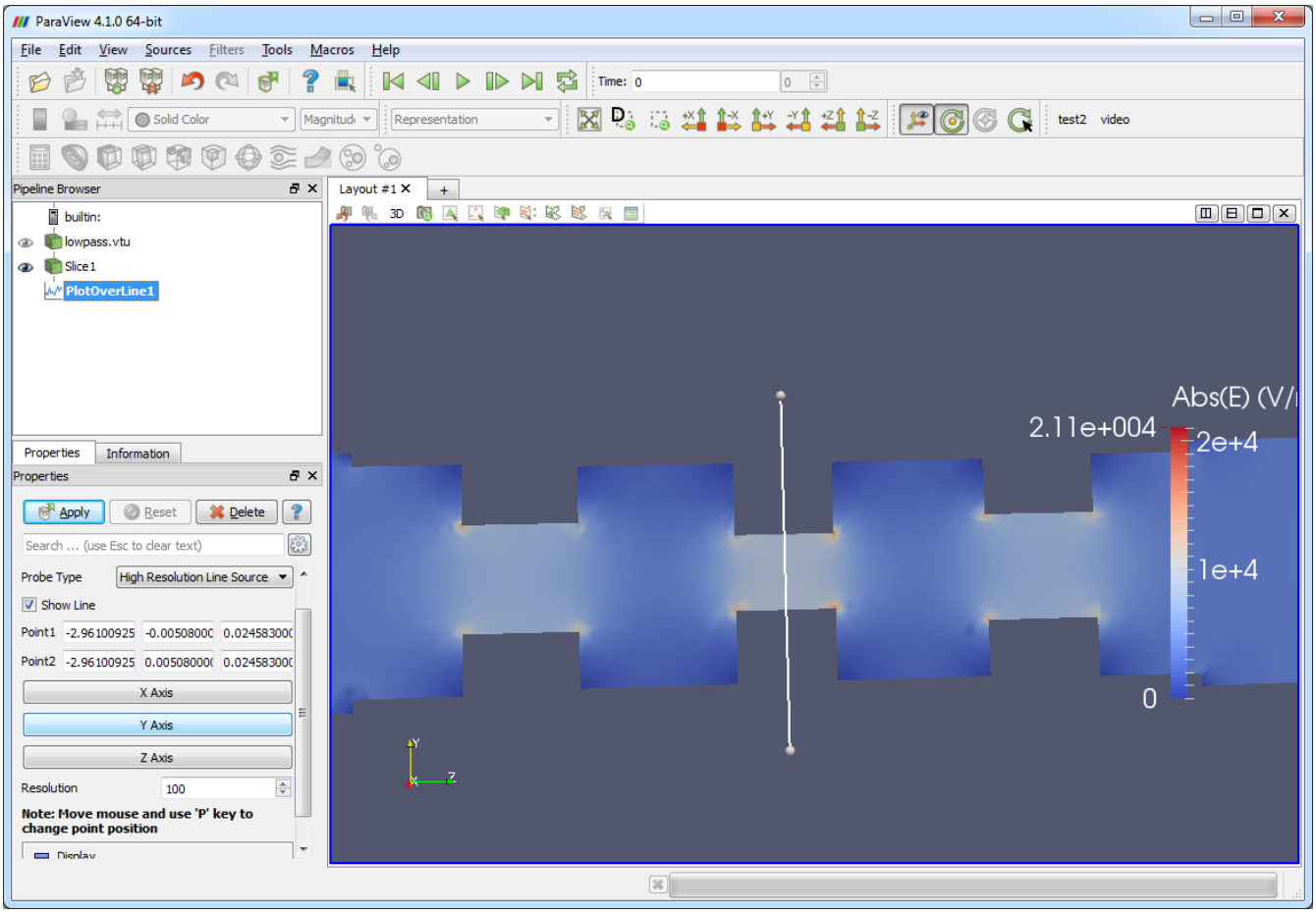
$$V = V1W\sqrt{P}$$

Be careful because the voltage computed this way depends on the selected path in the mesh. In order to have meaningful results, the device geometry and fields, should be similar to a parallel-plate case.

The process is as follows:

1. Apply paraview filter "plot over line"
2. Specify the coordinates of the line
3. Apply paraview filter "Integrate variables".

In this particular case we will compute the voltage in the center of the centre iris, where the maximum field is located. In order to do so, one has to select the "**Plot Over Line**" filter in Filters->Alphabetical->Plot Over Line menu.

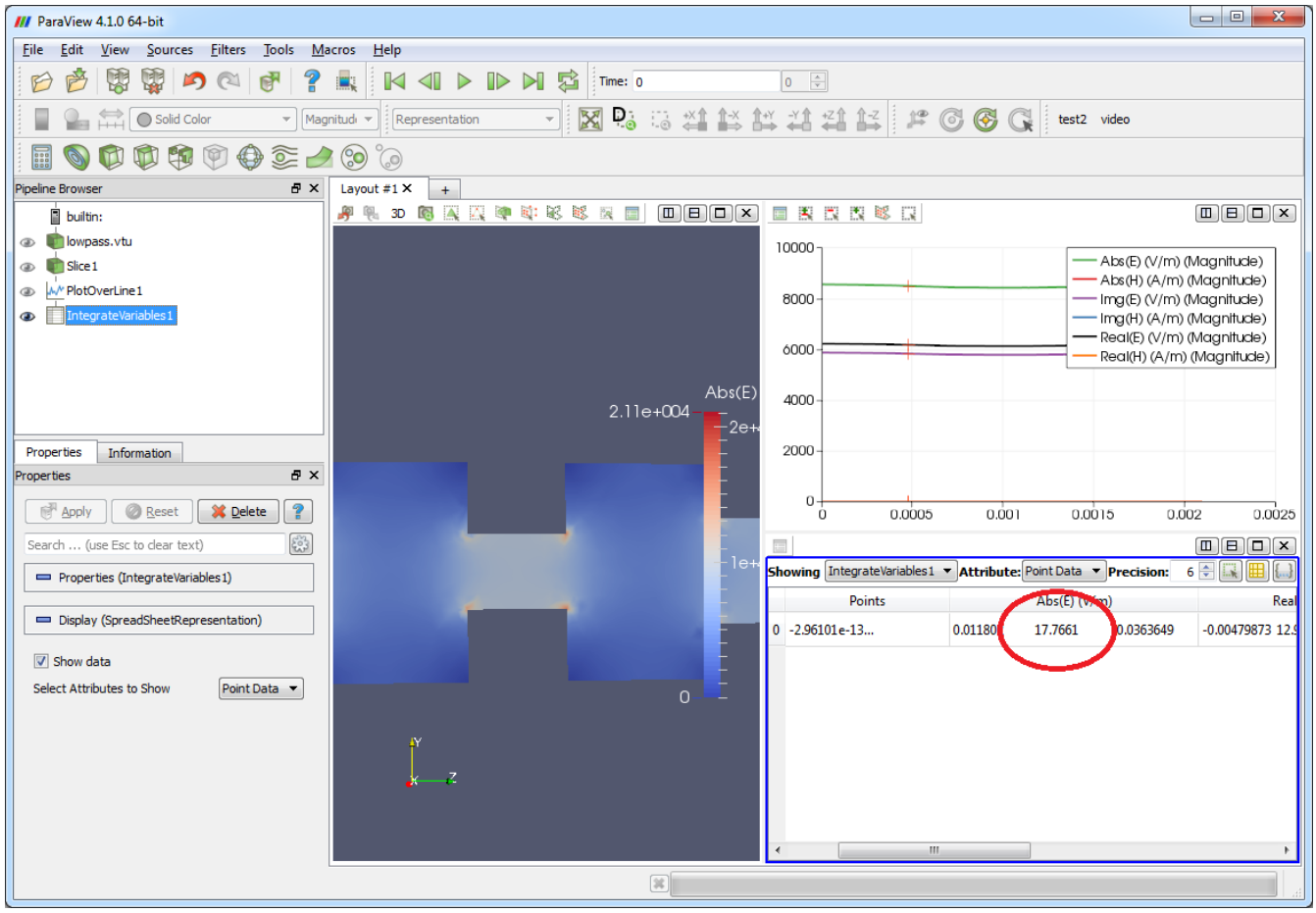


Select the line for displaying the data by either moving the start and end points with the mouse, or by inserting coordinates manually. In this case, just press "y axis" button to automatically orient the line properly. Then press "Apply" button.

A 2D plot with the fields displayed along the selected line appears. Now, apply another filter called "**Integrate Variables**" in Filters->Alphabetical->Integrate Variables. This filter will integrate all quantities displayed in the 2D plot.

In this case, **we obtain a voltage at 1W of V1W= 17.8 V** as shown below.

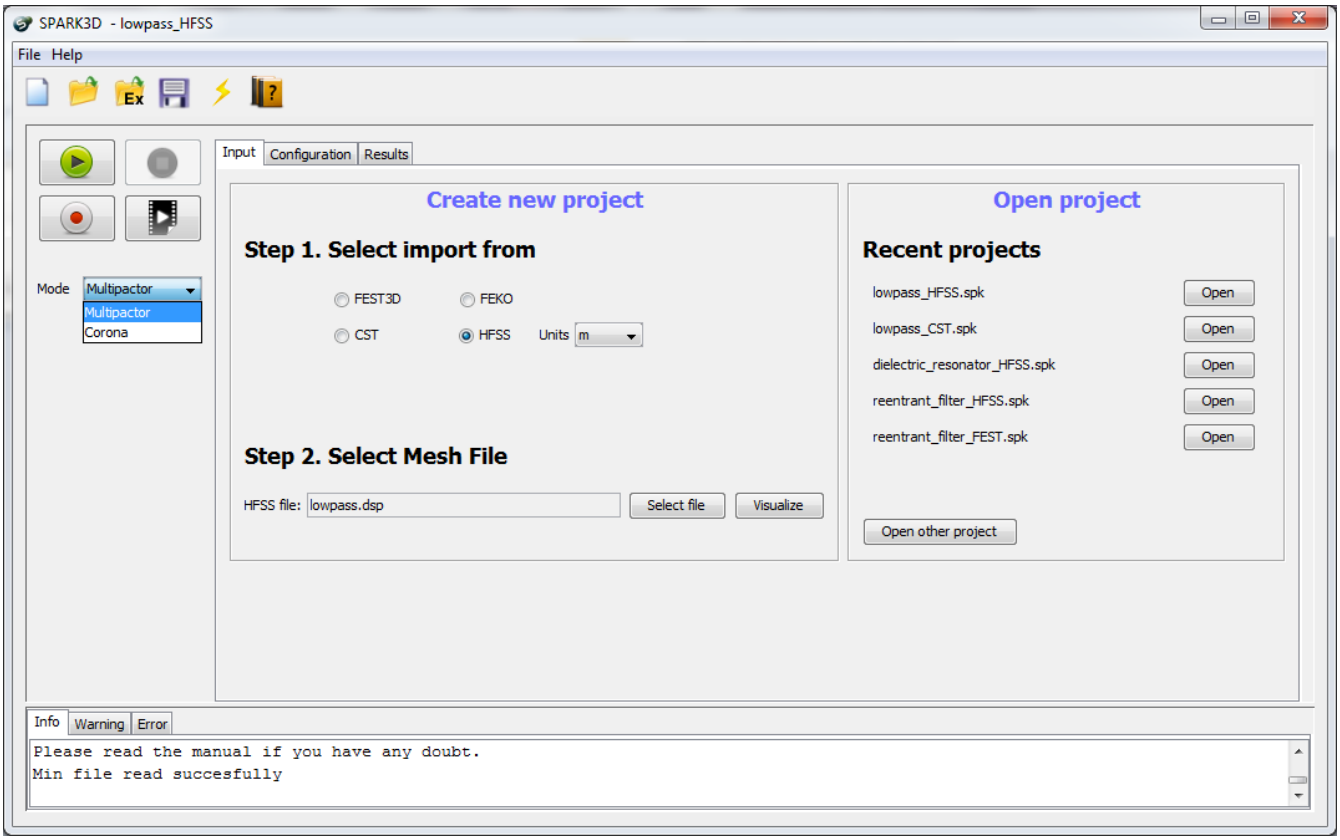
Note: Line start and end points must be adjusted to be inside a valid data region. If any of the line nodes lies outside, NaN integration values may appear.



More information on Kitware's Paraview can be found in http://www.paraview.org/.

Launching the simulation

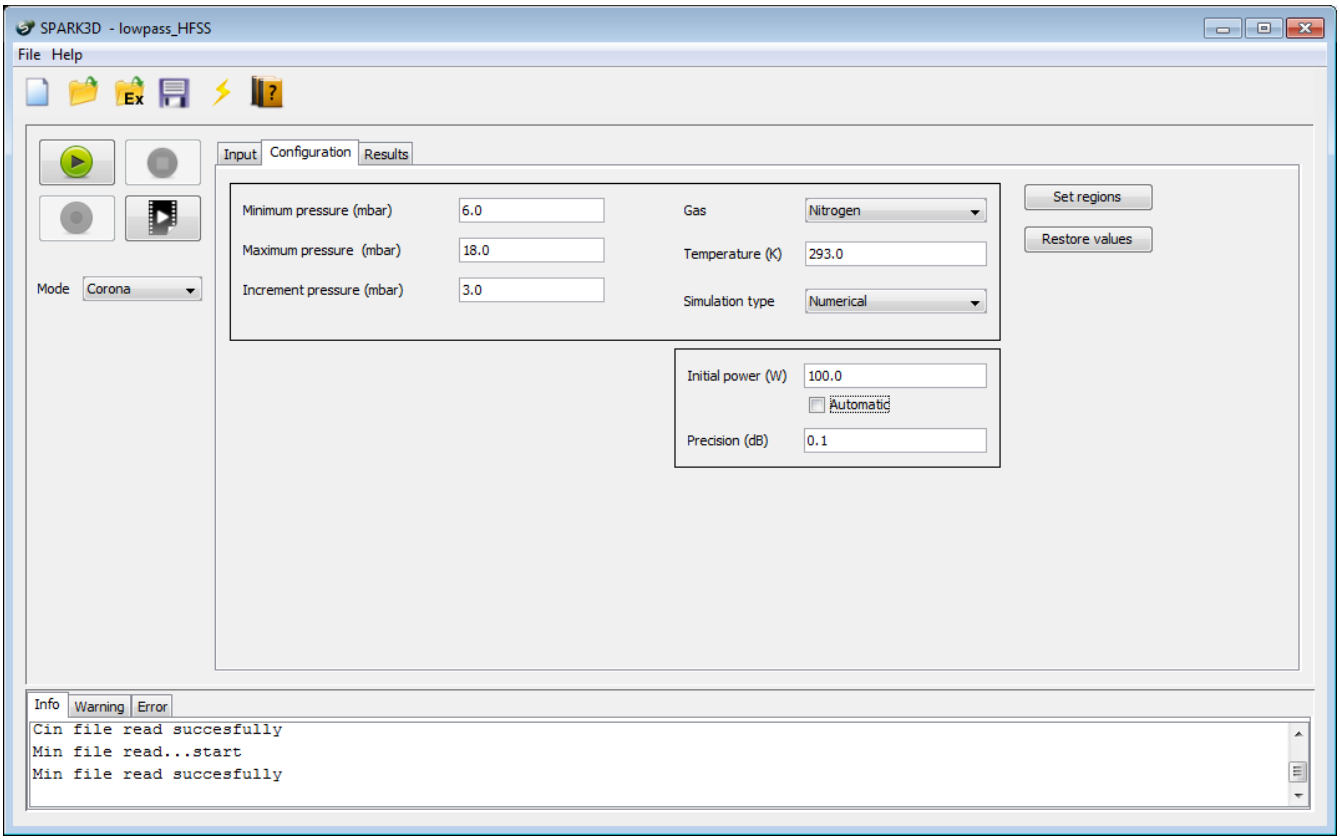
We must now select between Corona or Multipactor modes from the pull-down menu in the input tab to start setting up the configuration parameters.



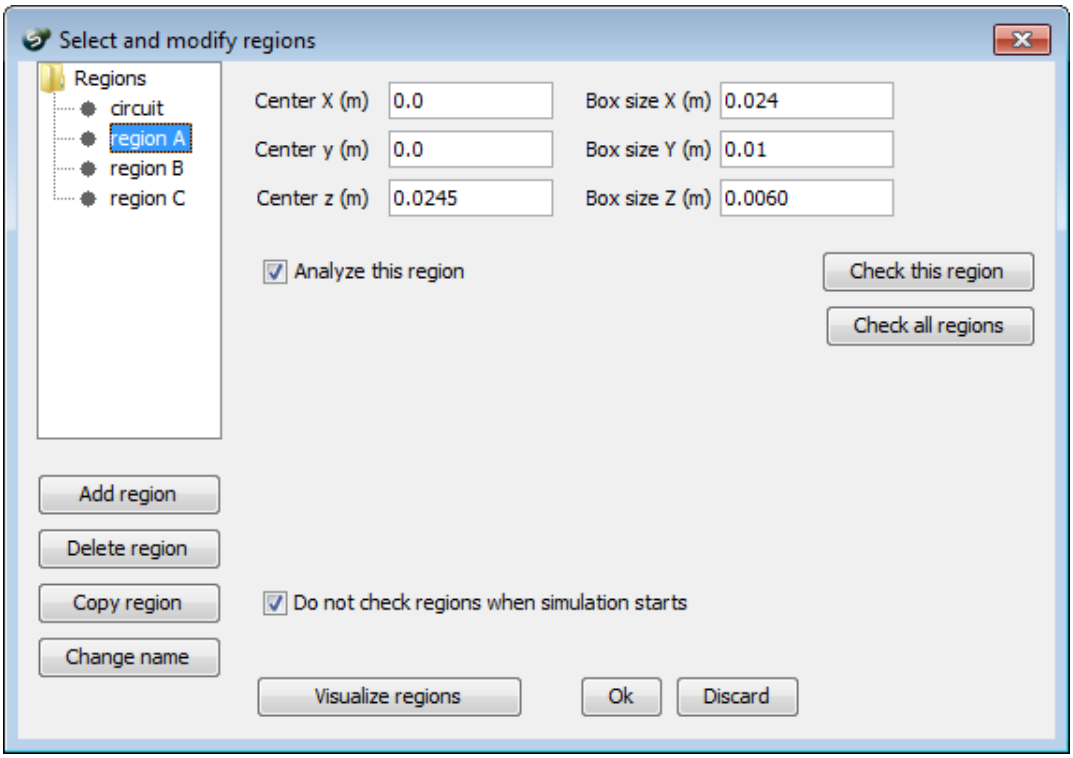
Running Corona mode

Once you have chosen Corona simulation mode, click on the **Configuration** tab to set up the configuration parameters. See Corona Analysis manual for a description of all the available options. In the configuration tab, fill in

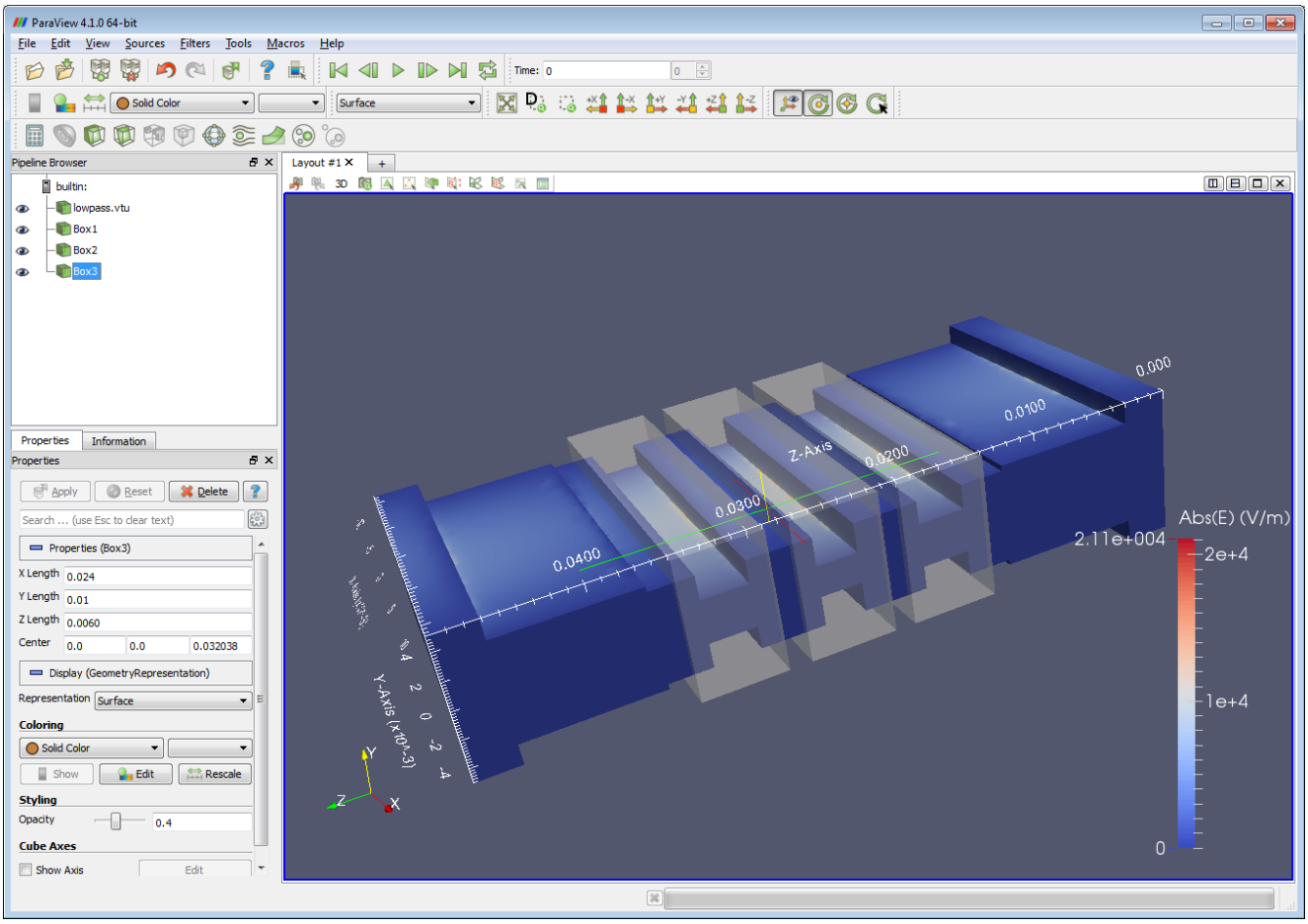
1. Minimum pressure (mBar): 6
2. Maximum pressure (mBar): 12
3. Increment pressure (mBar): 3
4. Simulation type: numerical
5. Temperature (K): 293
6. Precision (dB): 0.1
7. Gas: nitrogen



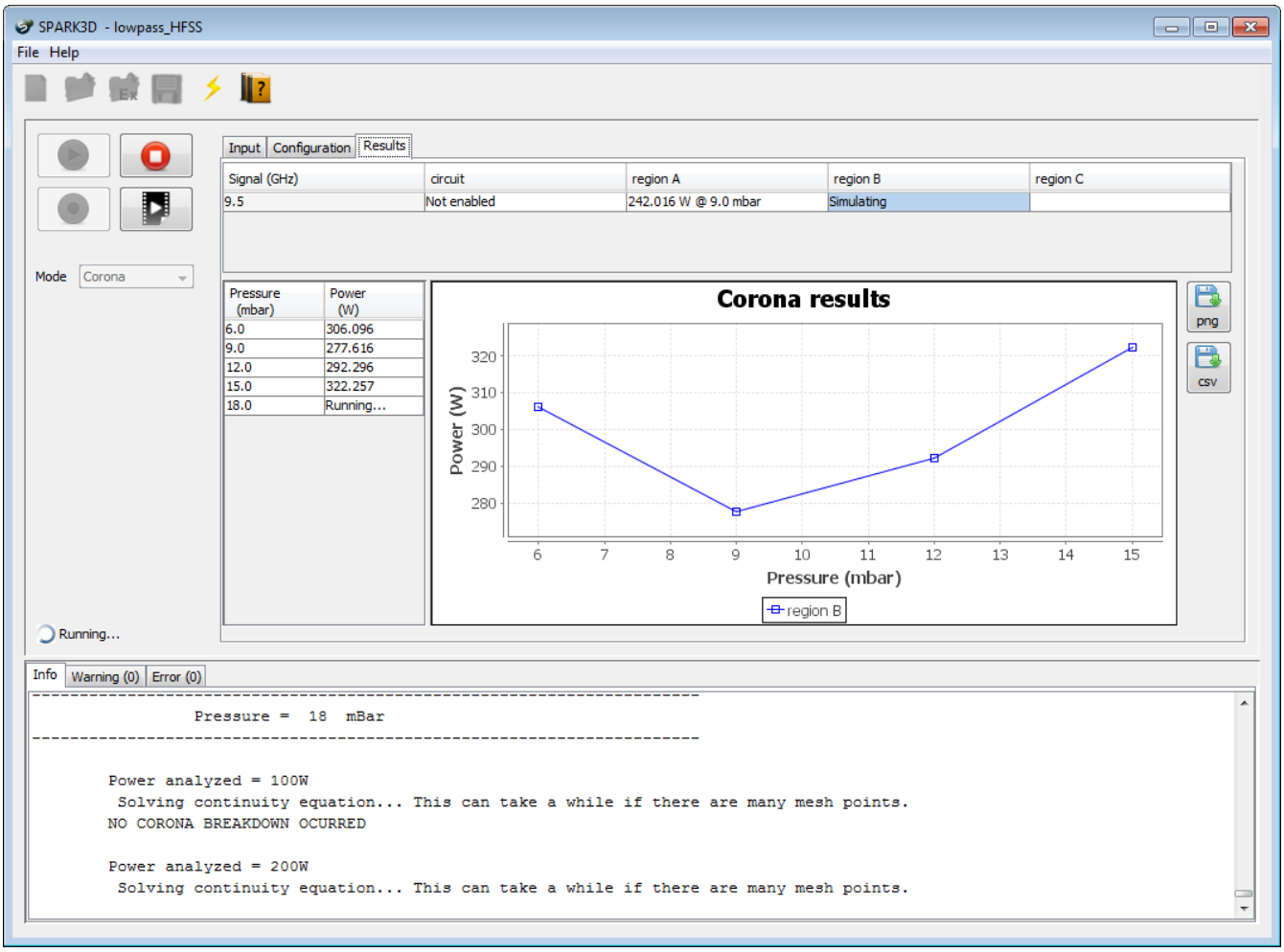
Press the **Set Regions** button and a new window will open, where you can configure the regions of analysis.



You can see the defined regions of study for this example by clicking on the **Visualize regions** button. The 3D CAD viewer Paraview will open and we will see the EM field of the device together with the defined regions, represented by boxes. For further information on how to work with regions see Specifying Regions tutorial.



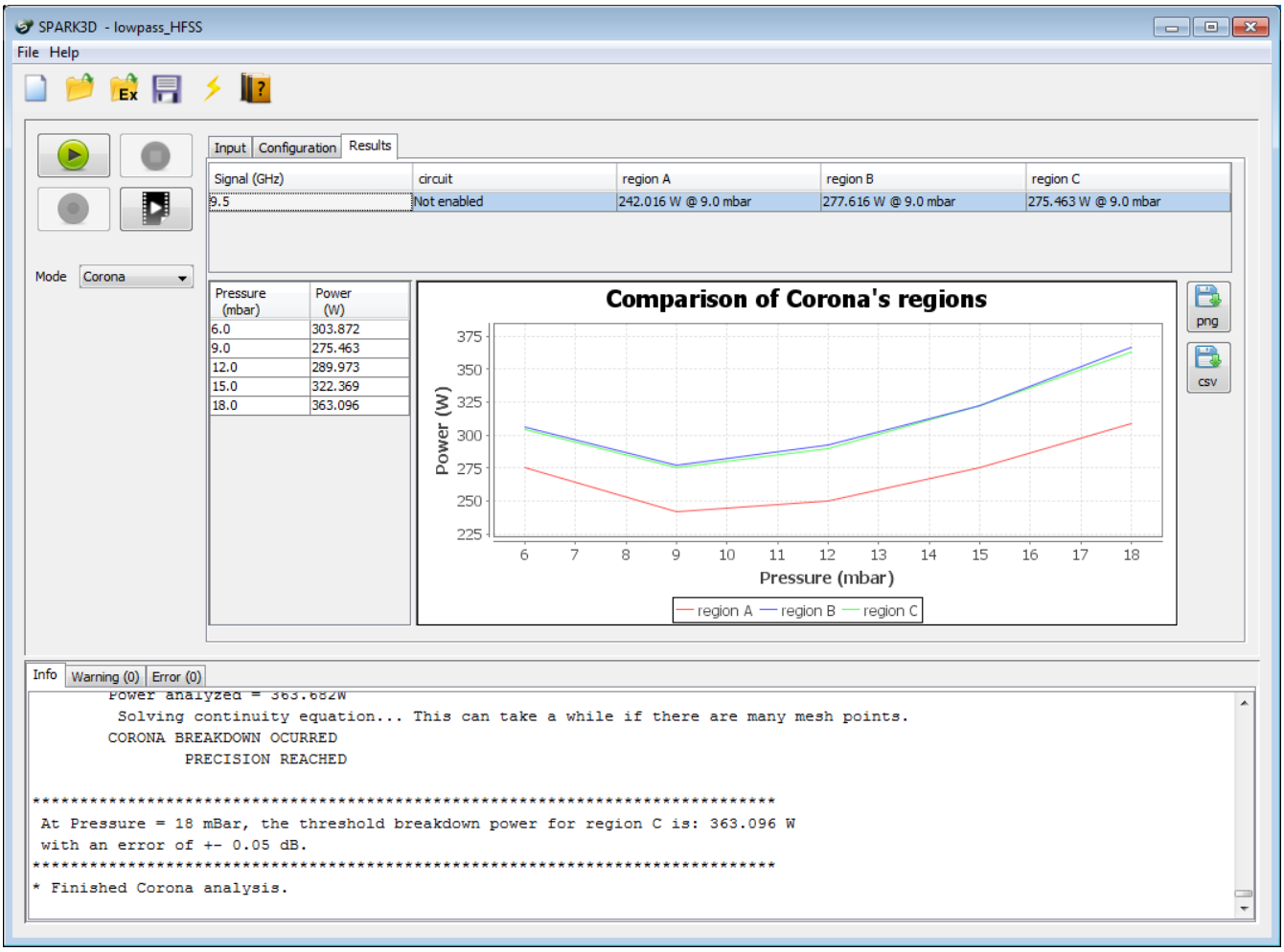
We are now ready to launch the simulation. Press the **play** button  . The simulation starts now. Click on the **Results** tab to see the output in run-time.



For each analyzed region, the results include:

- The representation of the Paschen curve, that is, the threshold breakdown power versus pressure.
- A table located in the left side of the window that corresponds to the points of the Paschen curve.

Besides, in the table situated on the top of the window, for each region of study it is shown the minimum breakdown power in the whole pressure sweep. With this information it is straightforward to identify which is the critical region of the device where breakdown onset is most likely to occur. For further details on how to interpret the results see Analysis of Results tutorial.



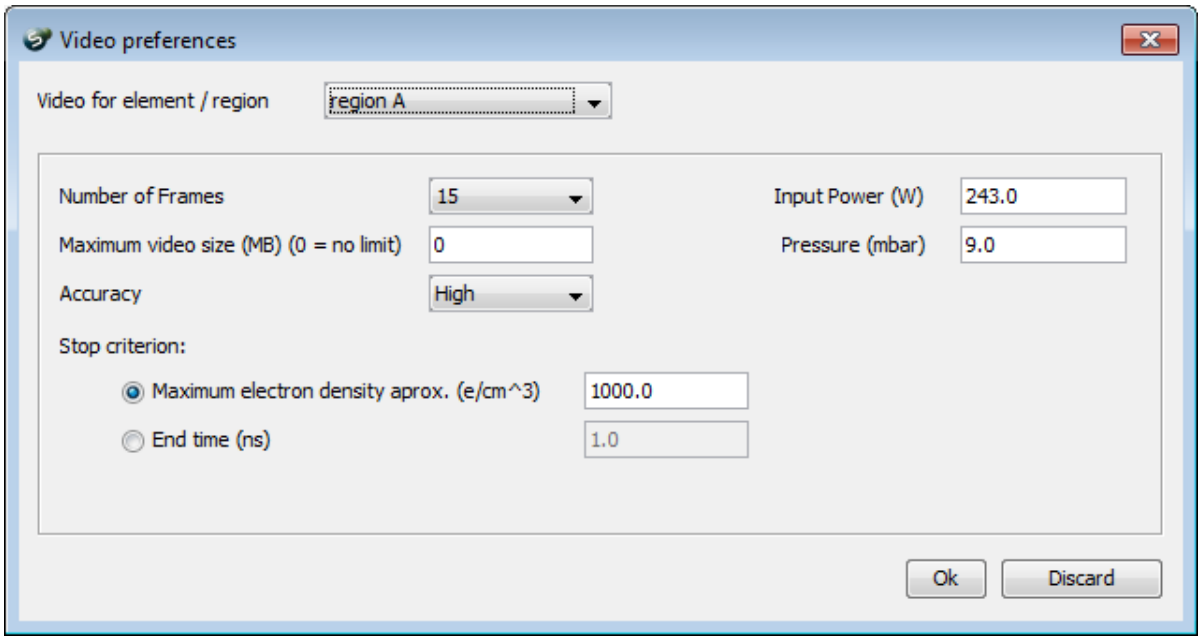
In this example, the Corona simulation shows that at the frequency of study (9.5 GHz) the region with lowest breakdown power in the chosen range of pressures (critical element) is region A. It is located in the center of the circuit and has a Corona breakdown of 242 W at 9 mBars, which is, finally, the limiting power of the device.

Running Corona Video

Alternatively to a corona analysis, it is possible to record a video of the electron density growing inside the 3D structure for a particular input power above the breakdown threshold.

Follow the provided steps for **opening the example file** (lowpass_hfss), setting the **simulation parameters**, and optionally define simulation regions.

Now, press the record video button  . The following window will appear:



Select:

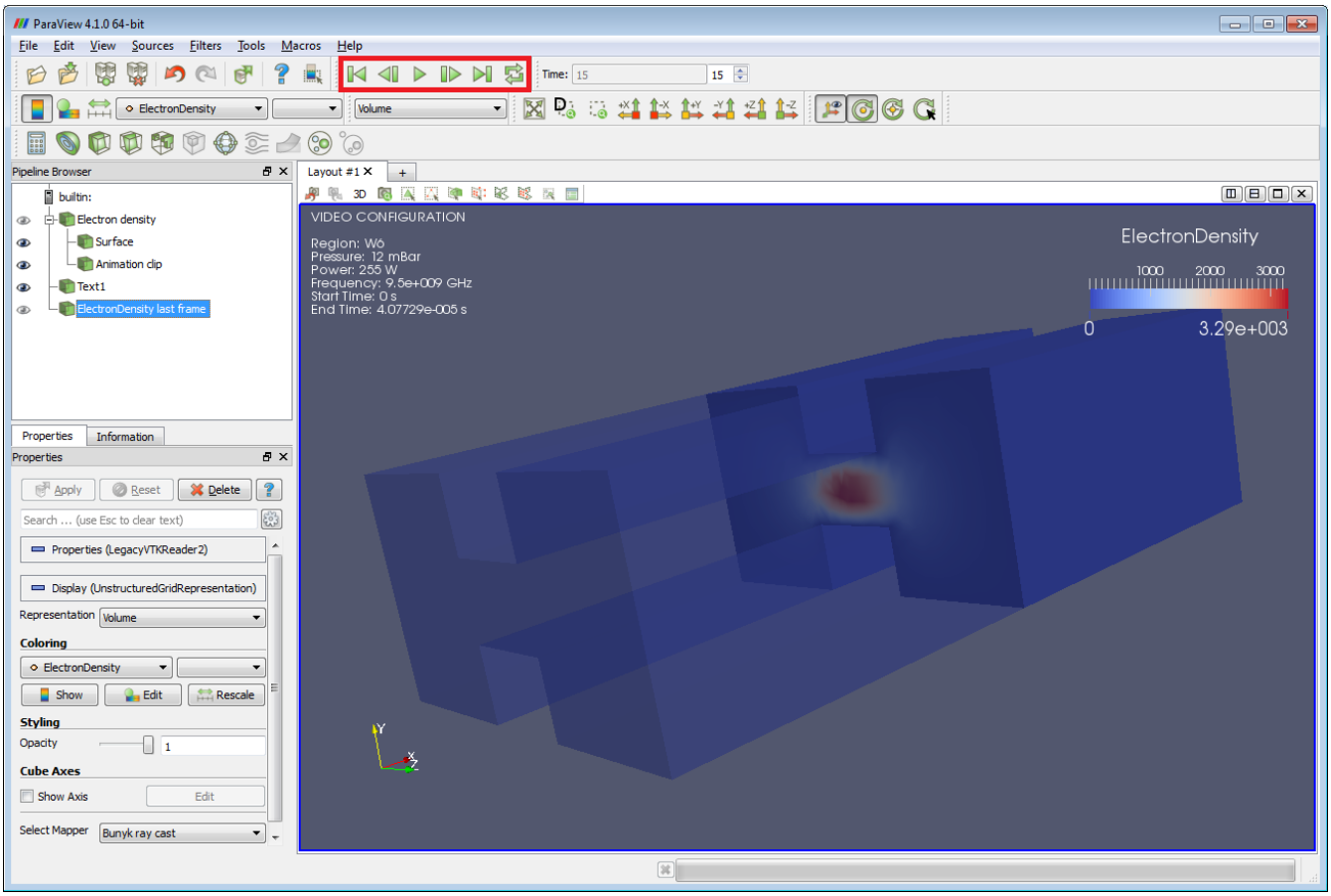
1. Video for element / region : Region A
2. Number of Frames: 15
3. Maximum video size (MB) (0 = no limit)
4. Accuracy: High
5. Stop criterion: Maximum electron density aprox. (e/cm^3): 1000
6. Input Power (W): 243
7. Pressure (mBar): 9

The remaining parameters, such as gas and temperature, are defined in the Configuration tab. Press Ok button and choose the output file (*.v3d format).

After that, the video generation will start. Note that the video is saved outside the project and will not erase previous simulation results.

When the video simulation is finished, you can choose to immediately visualize it, or open it later by pressing the play video button  in the main window at any time.

The video is visualized with the 3D CAD viewer Paraview. 3D rotations, perspective customization and zoom are allowed on recorded animations. Play, pause, forward and backward buttons can be found on top.

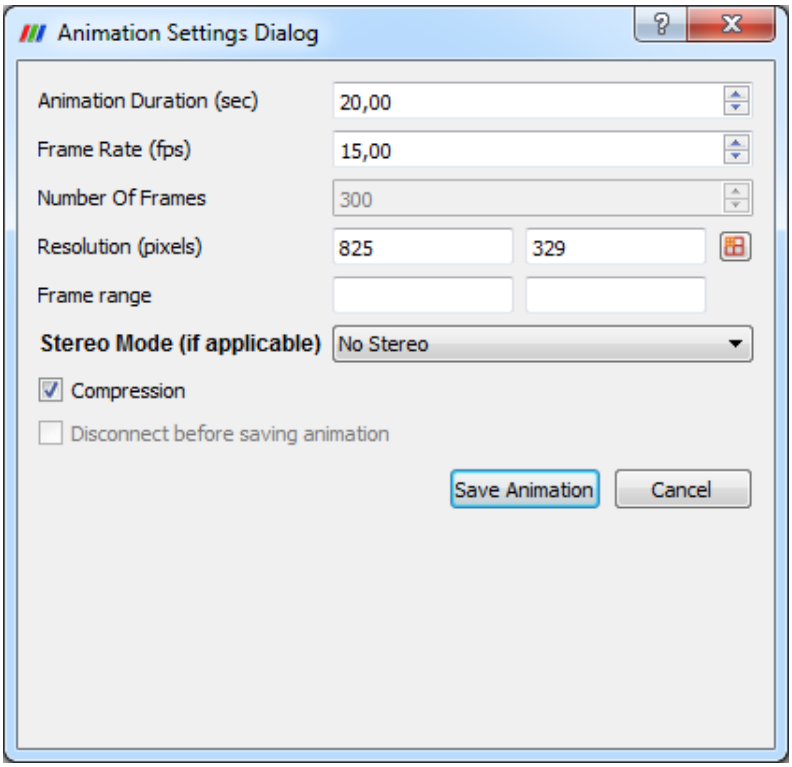


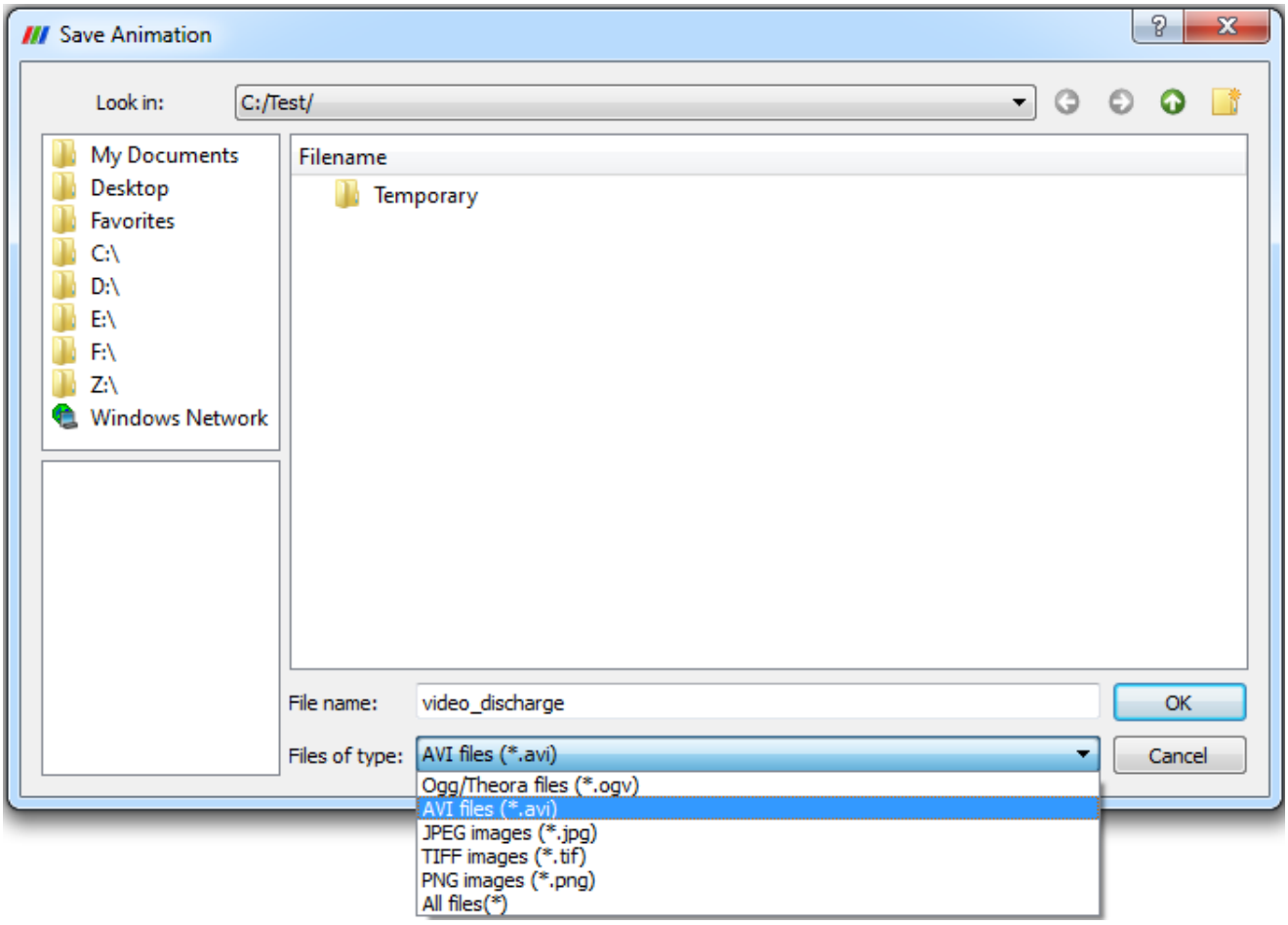
In the tree located on the left of the 3D CAD viewer Paraview window, there are different visualizations of the electron density evolution:

- Electron density: it corresponds to the electron density in the volume of the device at different video frames.
- Animation clip: it is a clip made on the electron density volume in order to visualize the discharge inside the device in a proper way. You can change the plane of the clip to center it in the proper place where the maximum of the discharge occurs by using the "Properties" tab or by dragging the plane on the visualization panel.
- ElectronDensity last frame: it corresponds to the last frame of the volume electron density.

You can enable/disable each one by clicking on the eye located in their left side.

The video can be exported in Paraview using File -> Save Animation... The video size, duration and format can be chosen as shown in the following pictures.



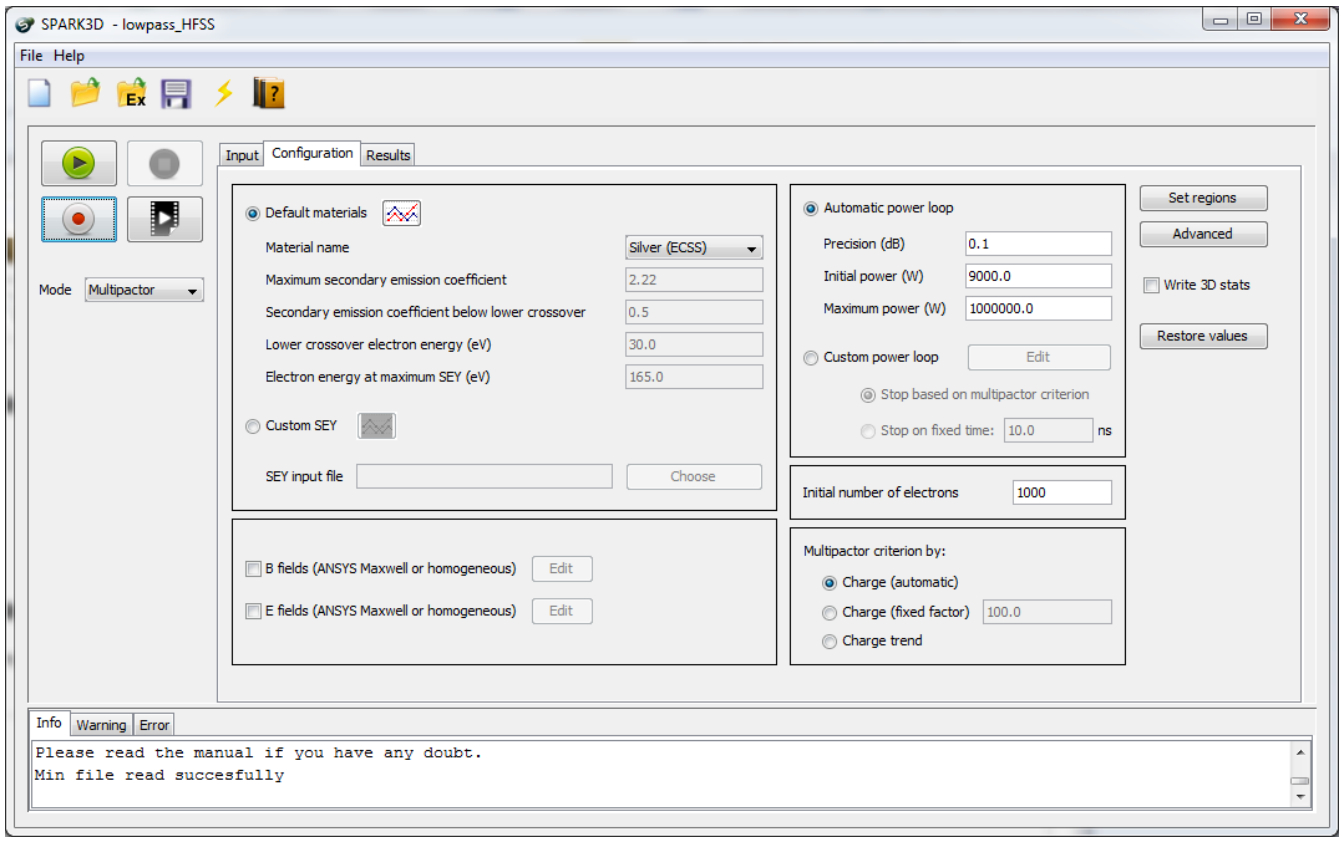


Running Multipactor mode

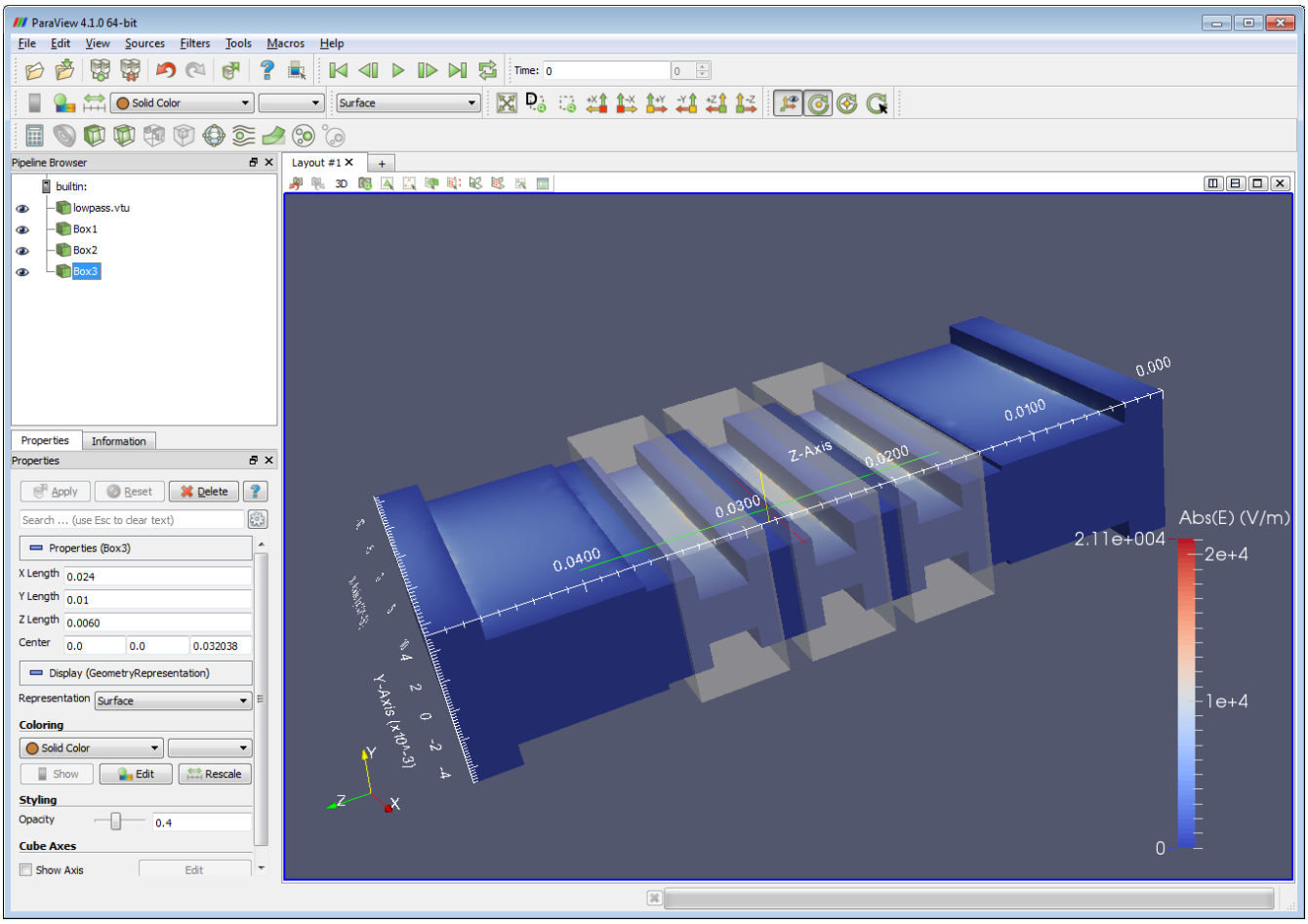
Once you have chosen Multipactor simulation mode, click on the **Configuration** tab to set up the configuration parameters. See Multipactor Analysis manual for a description of all the available options. In the configuration tab, fill in

1. Material name: Silver
2. Precision (dB): 0.1
3. Initial power (W): 9000

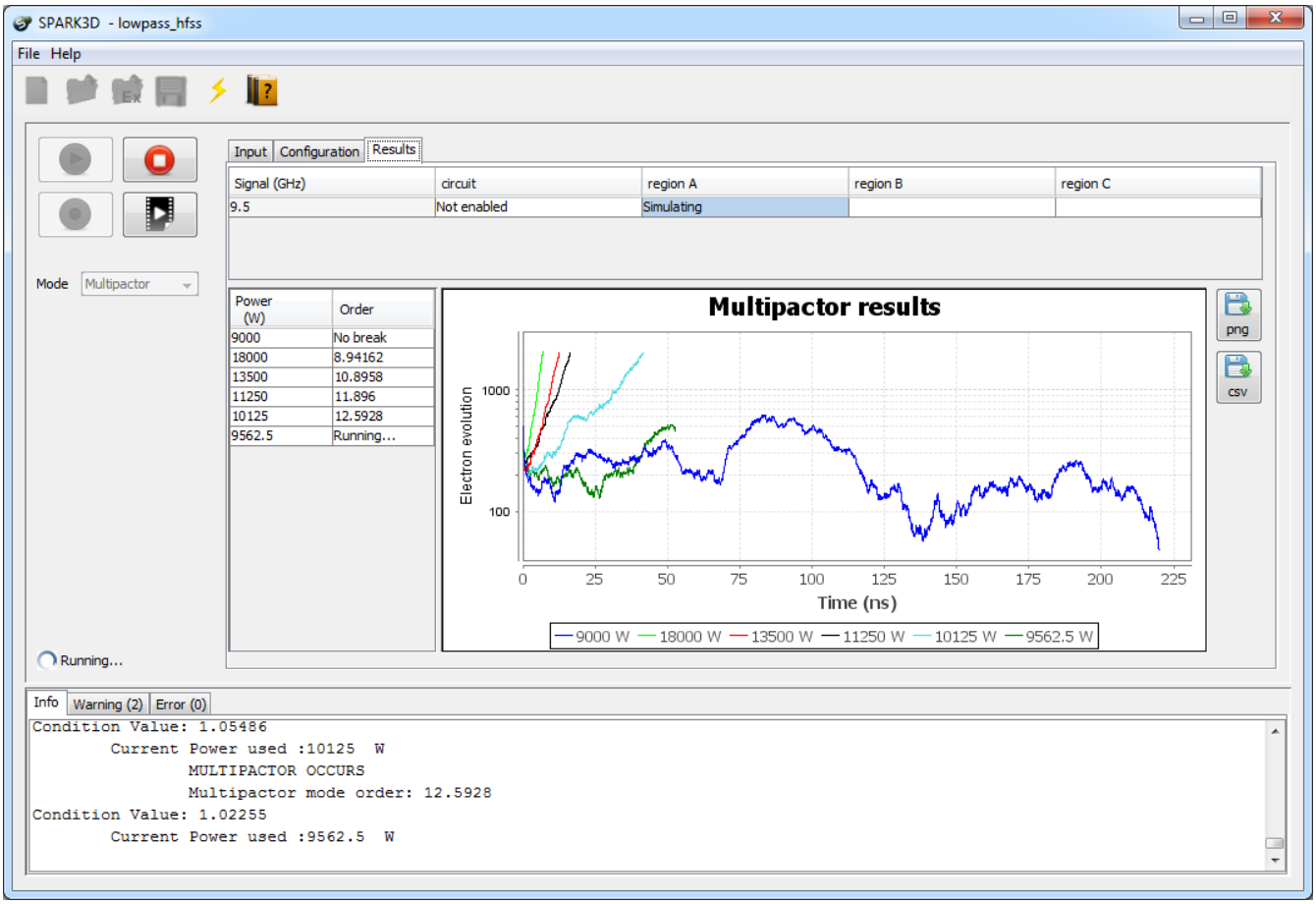
4. Maximum power (W): 100000
5. Initial number of electrons: 1000



Press **Set Regions** button. You can see the defined regions for the analysis of these examples by clicking on the **Visualize regions** button. The 3D CAD viewer Paraview will open and we will see the EM field of the device together with the defined regions, represented by boxes. For further information on how to work with regions see Specifying Regions tutorial.



We are now ready to launch the simulation. Press the **play** button  . The simulation starts now. Click on the **Results** tab to see the output in run-time.



For each analyzed region, the results include:

- A table located in the left-side of the window, which shows the analyzed powers in the process of searching the threshold breakdown power. For each power, depending on whether multipactor occurs or not, it appears either the order of multipactor or the message "No break", respectively.
- A graph, where for each analyzed power the electron population evolution is represented versus time.

Multipactor output data also includes a table situated on the top of the window, where it is shown the threshold breakdown power for the regions under study. With this information it is easy to recognize which is the most critical region in the device for multipactor onset and the limiting power. For a detailed description on multipactor results interpretation read the Analysis of Results tutorial.

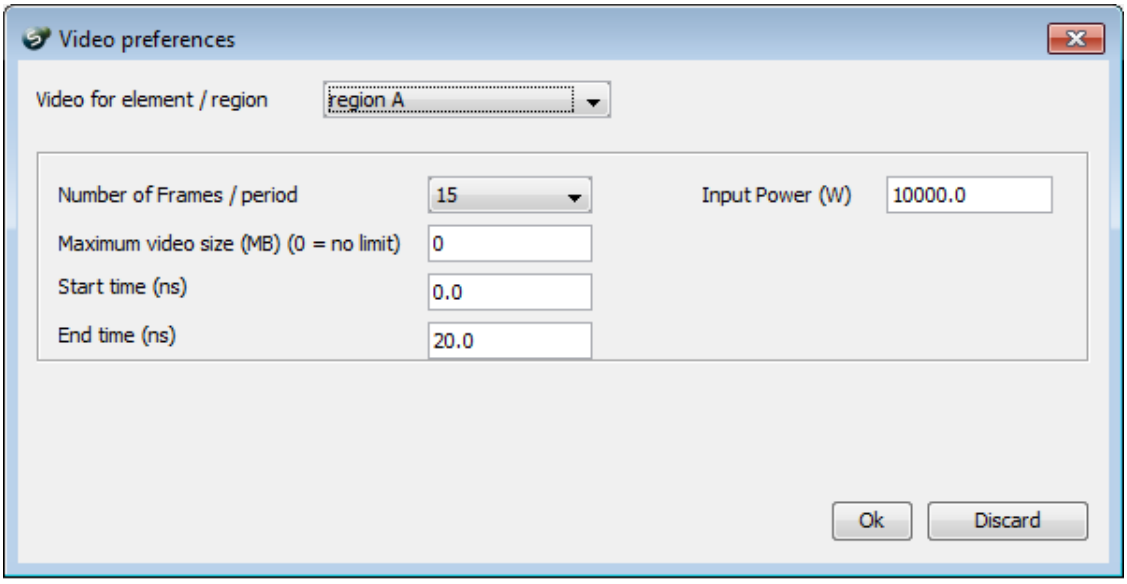
For the current example, we find that at the frequency of study, which is of 9.5 GHz, region A has the lowest breakdown threshold power, which is of 8859 W. Thus, this is the limiting power of the device.

Running Multipactor Video

Alternatively to a multipactor analysis, it is possible to record a video of the electrons moving inside the 3D structure for a particular input power.

Follow the provided steps for **opening the example file** (lowpass_hfss), setting the **simulation parameters**, and optionally define simulation regions.

Now, press the record video button  . The following window will appear:



Select:

1. Video for element / region : Region A

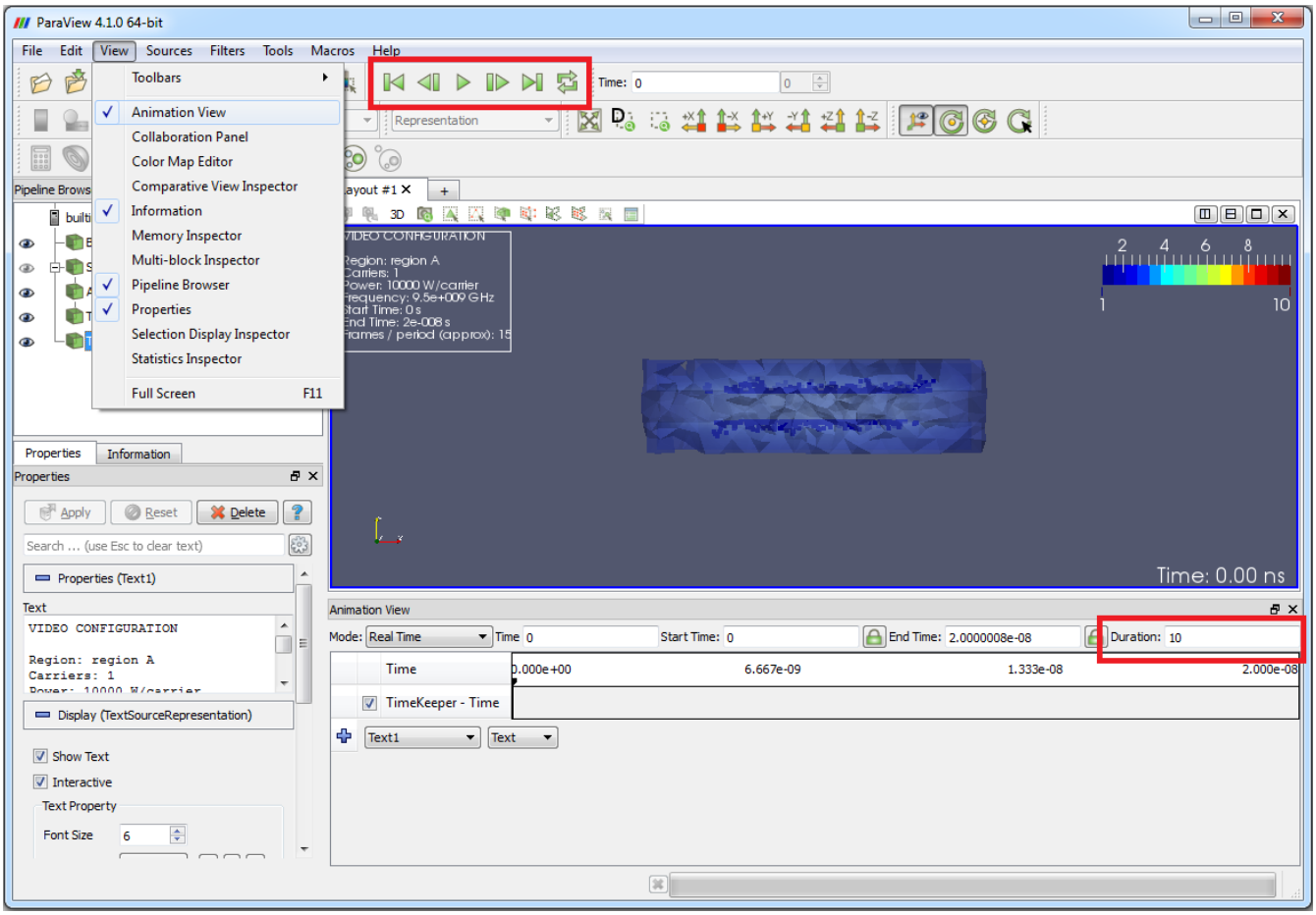
2. Number of Frames / period : 15
3. Maximum video size (MB) (0 = no limit)
4. Start time (ns): 0
5. End time (ns): 20
6. Input Power (W): 10000

The remaining parameters, such as SEY properties, number of initial electrons, multipactor criterion, etc. are defined in the Configuration tab. Press Ok button and choose output file (*.v3d format).

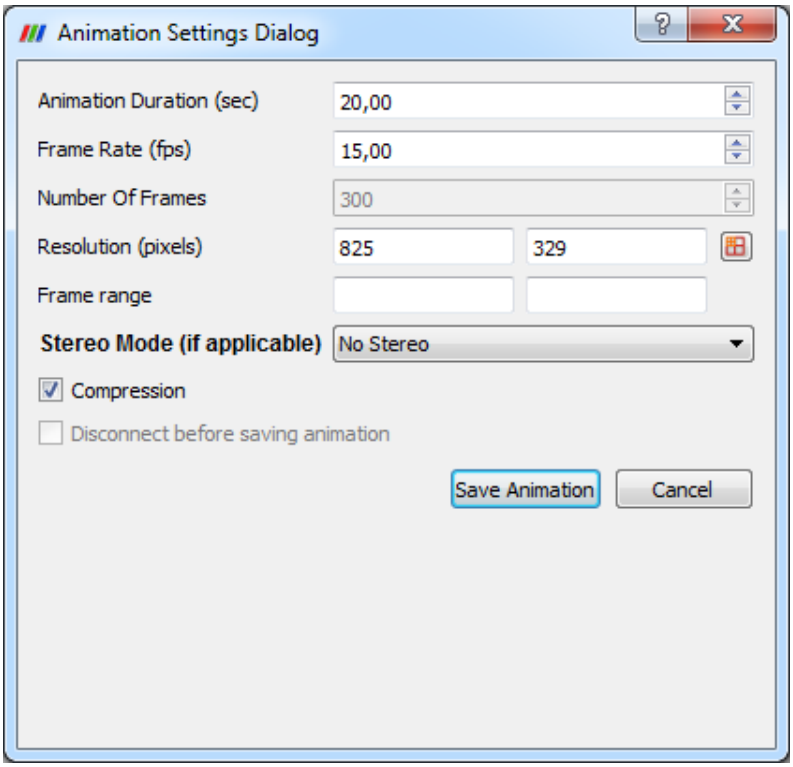
After that, video generation will start. Note that video is saved outside the project and will not erase previous simulation results.

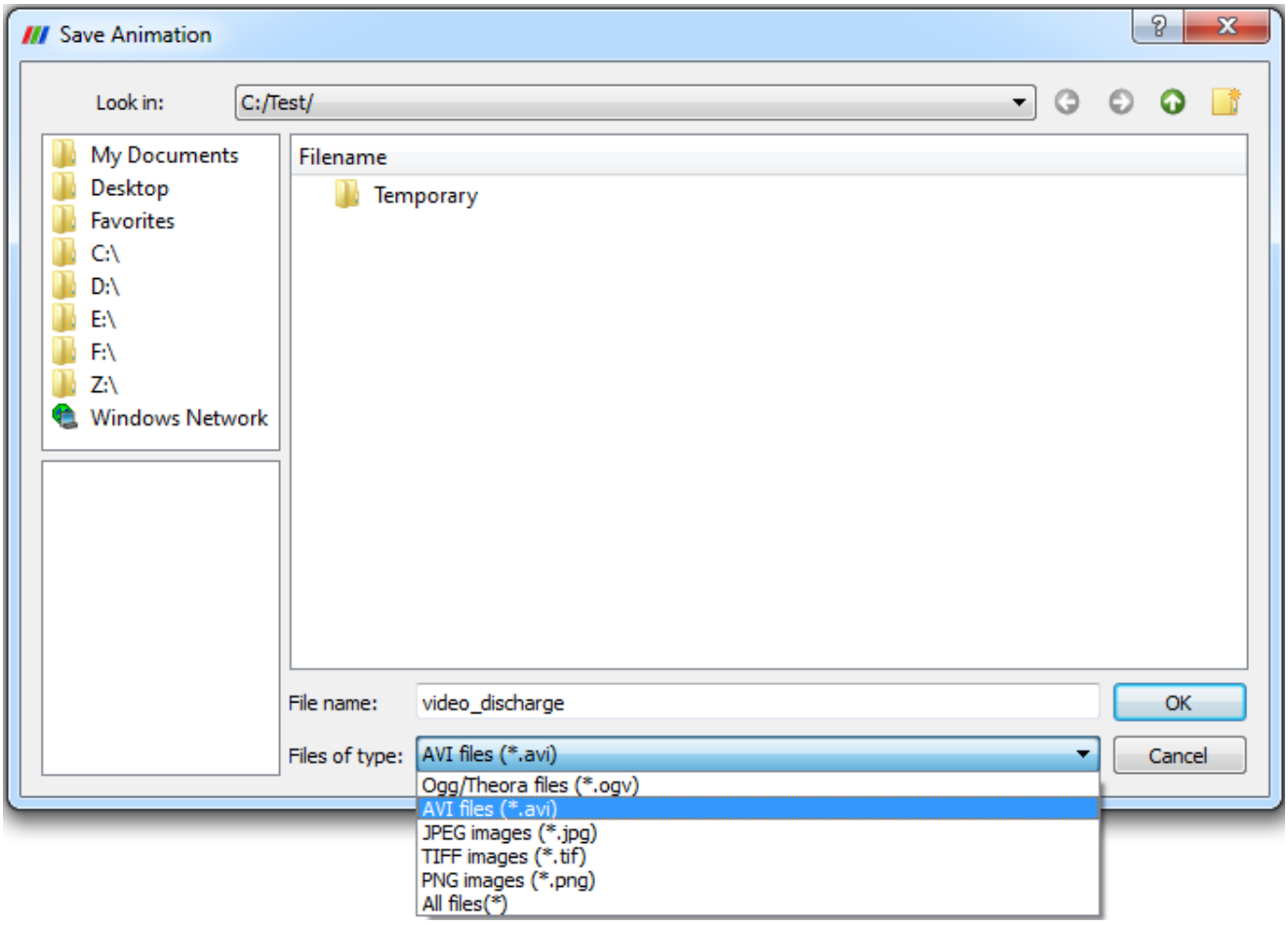
When the video simulation is finished, you can choose to immediately visualize it, or open it later by pressing the play video button  in the main window at any time.

The video is visualized with the 3D CAD viewer Paraview. 3D rotations, perspective customization and zoom are allowed on recorded animations. Play, pause, forward and backward buttons can be found on top. Animation parameters can be changed in the Animation View panel (View -> Animation View). Concretely, the video duration can be changed in the Duration textbox.



Video can be exported using File -> Save Animation... The video size, duration and format can be chosen.





2.2.3 Specifying Regions

The high power analysis of a device can be carried out in two different ways:

- analyzing the whole device as one or
- focusing the simulation on critical regions defined by the user, where the breakdown onset is more likely to occur.

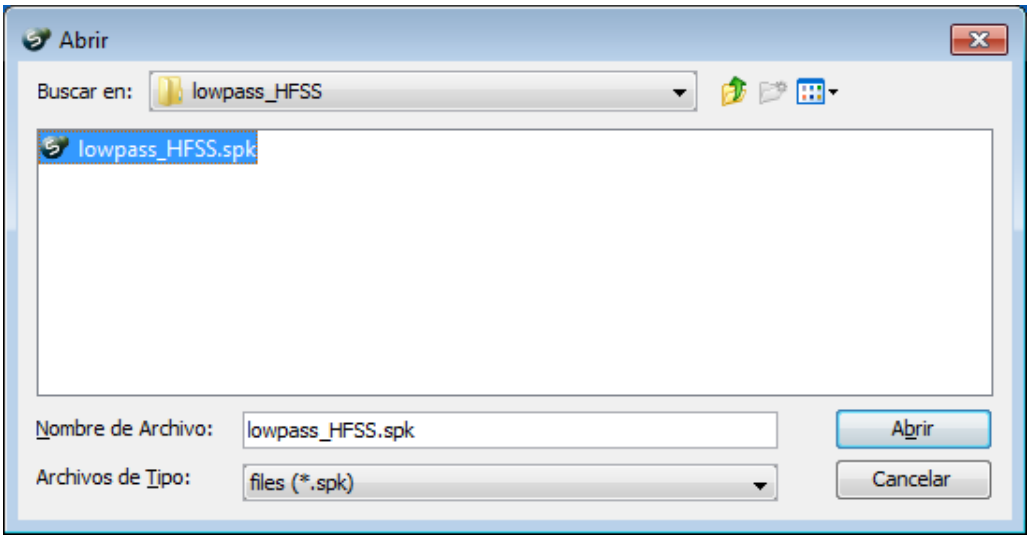
There are different reasons to take advantage of user-defined regions. As long as the device is divided in several areas it is possible to compare the breakdown threshold of each one and determine where the discharge will take place. Besides, computing the breakdown onset on specific regions is faster than taking into account the whole circuit. Finally, the user can increase the mesh density involved in the solution of the problem improving the precision of the calculation and avoiding memory overflow limitations.

In this tutorial you will learn how to define regions of study and how to deal with them through a guided example. The way to do it is the same for Multipactor and Corona modes, so we will just consider one of them. This tutorial is divided in 3 parts:

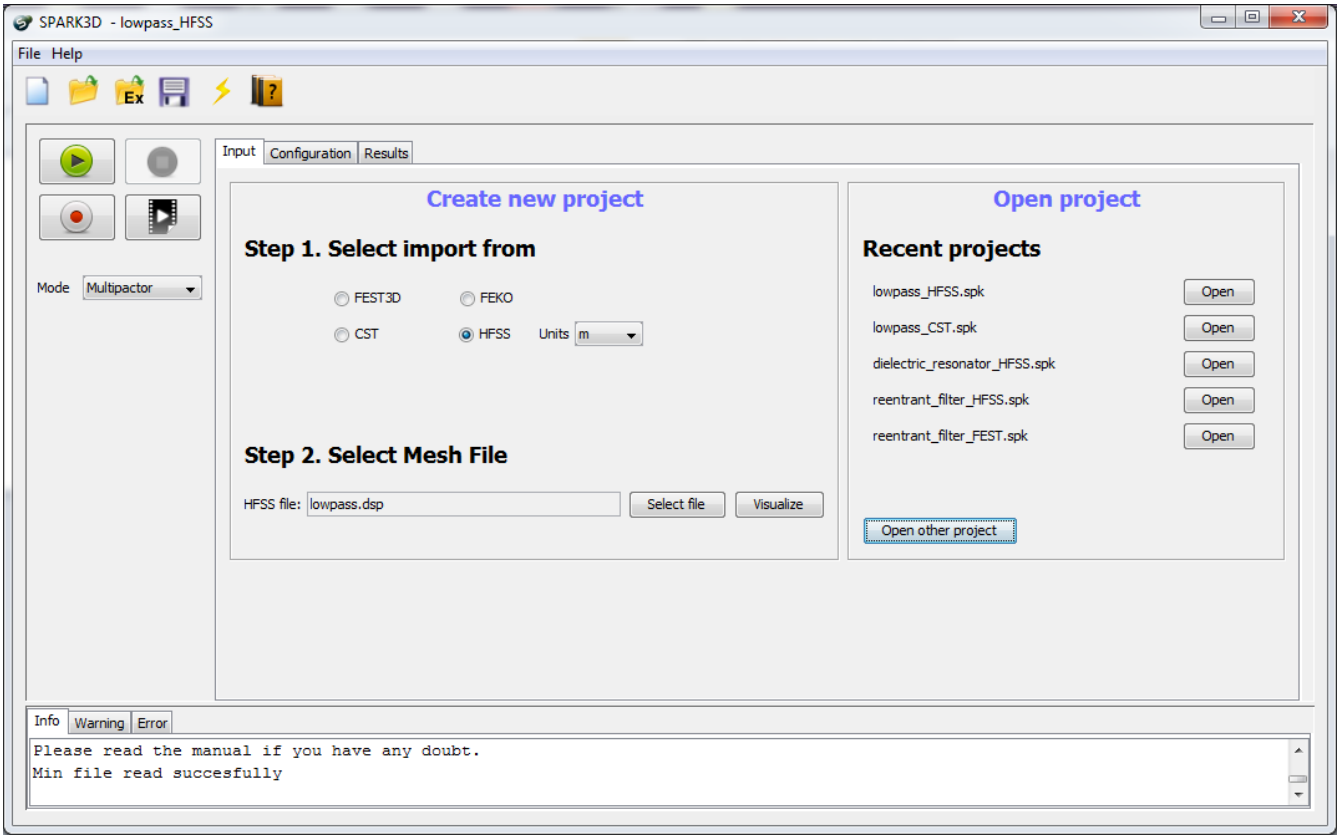
1. **Preliminaries.** We open an example and discuss which considerations should be taken prior to regions definition.
2. **Working with regions.** We show the different capabilities that are available in this feature.
3. **Errors.** We deal with possible errors produced when defining regions and give some solutions or workarounds to them.

Preliminaries

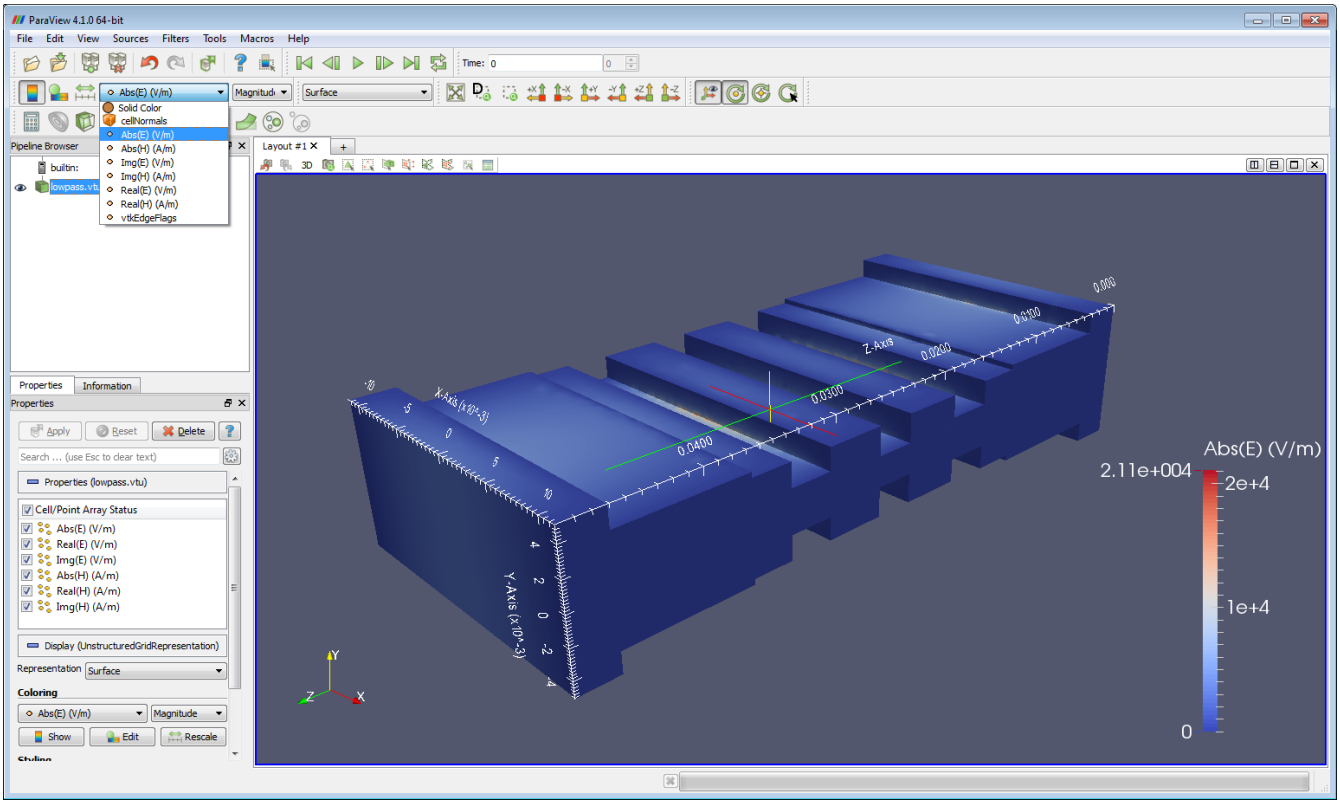
Click on the examples icon  and open Lowpass->lowpass_hfss.spk file.



Prior to the creation of simulation regions it is advisable visualizing the electromagnetic field in order to detect the potential areas of the structure where the breakdown onset is more likely to occur. In the input window, click on **Visualize** button.



A 3D CAD viewer included with SPARK3D distribution, Paraview, will pop up and you will be able to inspect the EM field. In the menu bar, choose the visualization of the Abs(E) (V/m) so that you can determine which are the potential regions of analysis. In this example, it is easy to recognize that the potential regions for breakdown onset lie in the waveguides corresponding to the fourth, sixth and eighth elements of the circuit.



A region of study corresponds to a box, which is defined through its center and size. These input variables can be determined from the visualization window, where a cube axis helps us to obtain their values. In our example, the regions will be defined through the following values:

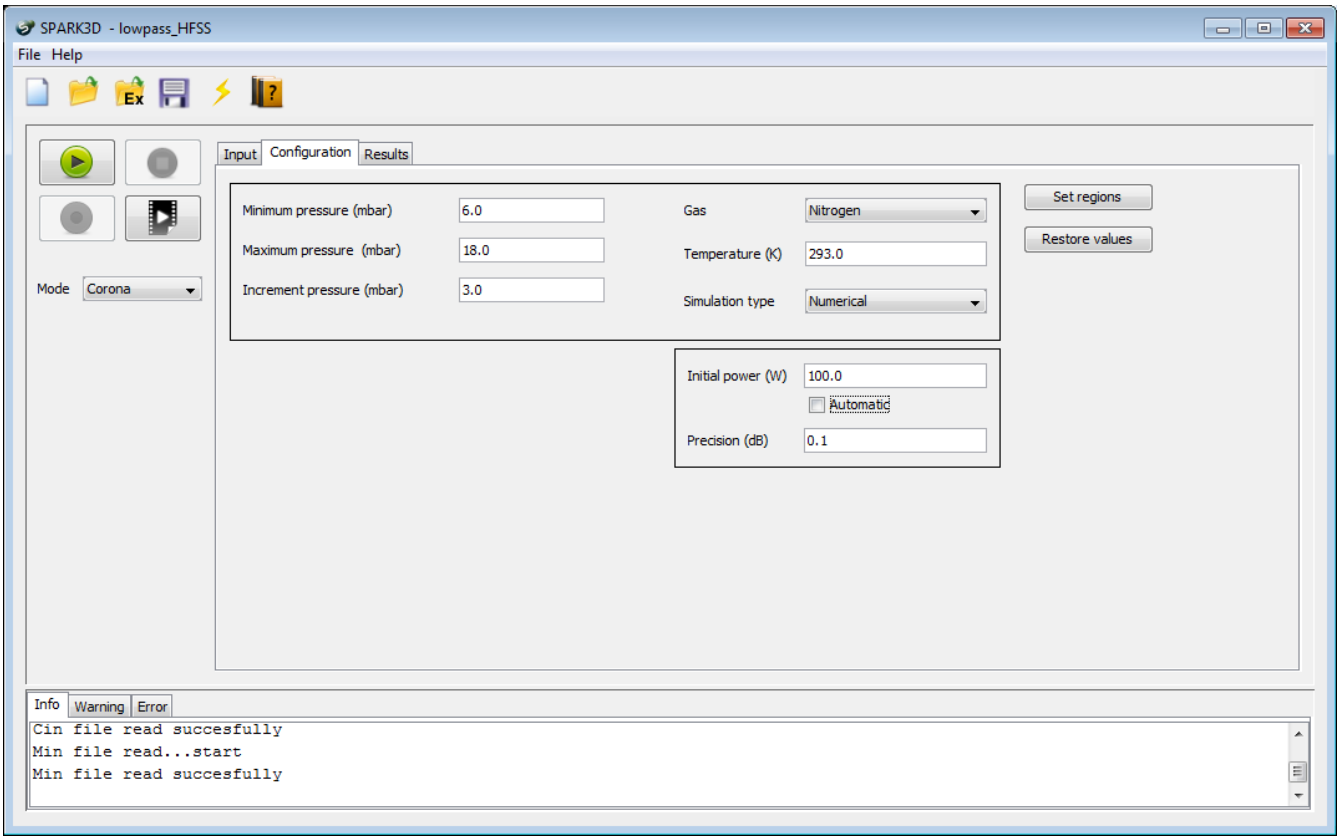
Region A	x	y	z
Center (m)	0	0	0.0245
Size (m)	0.024	0.01	0.006

Region B	x	y	z
Center (m)	0	0	0.017128
Size (m)	0.024	0.01	0.006

Region C	x	y	z
Center (m)	0	0	0.032038
Size (m)	0.024	0.01	0.006

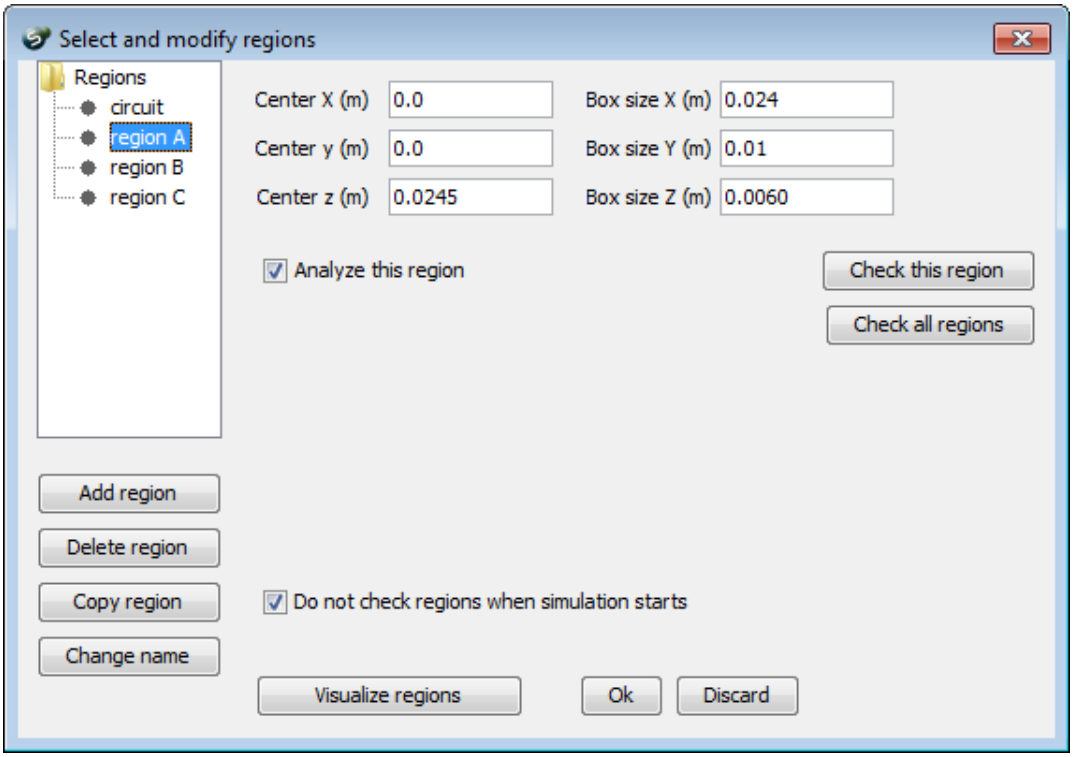
Working with regions

In order to define a region of interest, you should go to the **Configuration** window of either Multipactor or Corona modes.



Click on **Set regions** button and a new window will open from which you will be able to:

- create/copy a region,
- modify/delete the existing ones,
- change a region's name,
- enable/disable for analysis the existing regions,
- check the validity of each region or disable the region's validation made at the beginning of the simulation,
- and visualize all of them together with the device.



On the left hand side of the window, you see the folder tree corresponding to all existing regions. By default, there is a predefined region named **circuit**, which takes into account the whole device and is enabled for analysis.

By click on a specific region you can modify its defining properties. Click on region A and you see that in our example the

input variables:

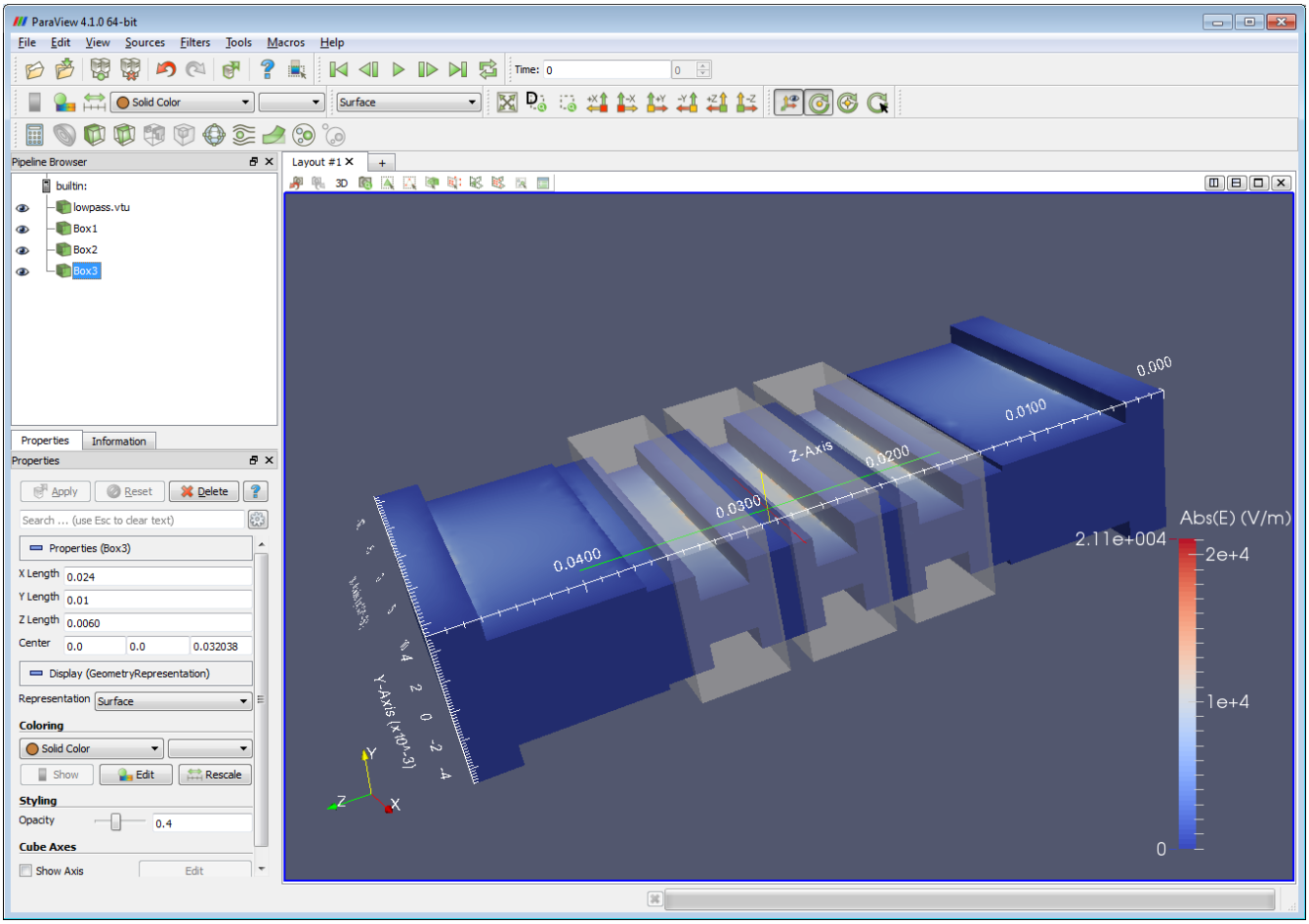
Center x, Center y, Center z

Size x, Size y, Size z

have the values given in the table above corresponding to one of the regions of interest. Note that the units of these variables are meters.

You can enable or disable a region through its checkbox **Analyze this region**. In our example, the region named circuit is disabled and region A, region B and region C are enabled.

It is possible to verify the validity of a certain region by clicking on the button **Check this region**, that checks if the region contains some mesh points of the device. On the other hand, if the user has the certainty that all regions are valid, it is also possible to disable the validation process made by default at the beginning of each simulation, which reduces the computing time. Besides, you can also visualize the relative position of all regions with respect the structure under study. Press **Visualize regions** button and you see that the defined regions correspond to the critical areas previously recognized in the lowpass filter.



Through the 3D CAD viewer it is possible to modify the defined boxes and visualize at once the changes. From the Pipeline Browser located on the left side of the window, select the box you want to modify (see the figure above, where box3 is chosen). Then in the Object Inspector window select Properties tab, where the geometrical parameters of the box, that is, its dimensions and center position, will be displayed. You can change them and by clicking on Apply button you can see the result of the modification. It is important to point out that the changes made in the 3D CAD viewer will not be automatically transferred to the defining parameters of the regions. Once you have found the proper values that suit your problem, you have to write them in the "Select and modify regions" window of SPARK3D.

If you want to create a new region you just click on **Add region** button. Then, you choose the name of the new region and fill in the input parameters. On the contrary, if you want to erase a region, you should select it and press **Delete region** button. You can also copy one existing region through the **Copy region** option or change its name through the **Change name** button.

Once you have done all your modifications, you can either preserve them through **OK** button or discard them through the **Discard** button (or alternatively closing the window).

In Multipactor simulations, when regions are specified, all non-metallic surfaces (corresponding to vacuum condition) of the mesh enclosed in a specific region are considered as open boundaries. As a consequence, all electrons impacting with such regions are automatically absorbed.

Errors

When checking the validity of a region it may occur that it is not correct, that is, there are no mesh points inside it. The reason of this could be one of the following:

- The region does not lie inside the device. You should check the defining input variables.
- The region is smaller than the mesh elements. You should enlarge the region or mesh thickly.

2.2.4 Analysis of Results

In this tutorial the capabilities of SPARK3D GUI to handle and better analyze the high power results are shown. As general features you can find that:

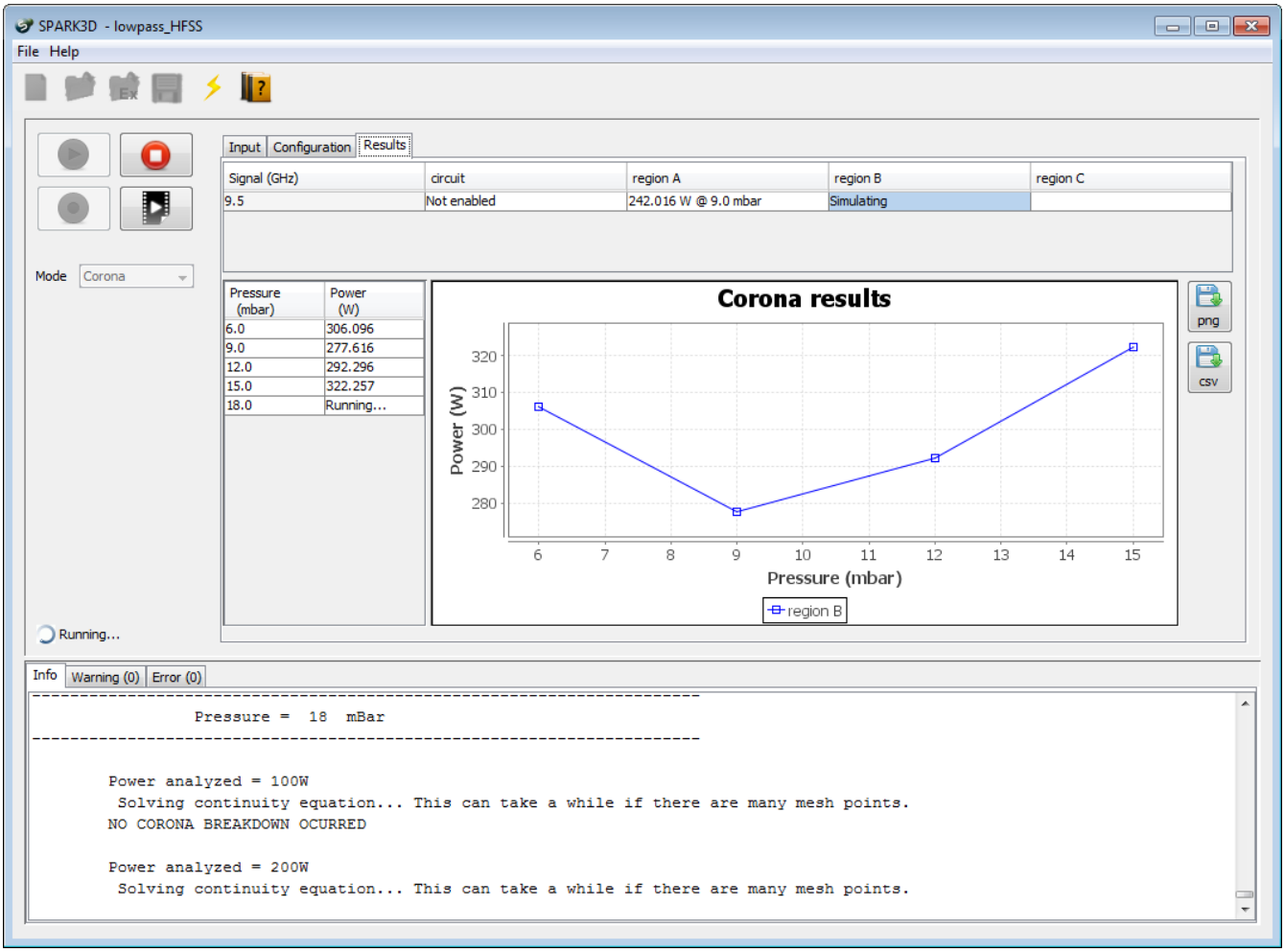
- results are given both in tabular and graphic form for a better understanding and can be saved in .csv or .png format respectively.
- partial results can be seen in run-time, that is, both tables and graphs are updated as results are being obtained
- for both Corona and Multipactor modes, the simulation process can be followed in the info tab of the GUI where a sweep in input power is shown as the simulation runs, indicating how the simulator tries to approach the breakdown threshold level.

This tutorial deals separately with:

1. **Corona results.**
2. **Multipactor results.**

Corona results window

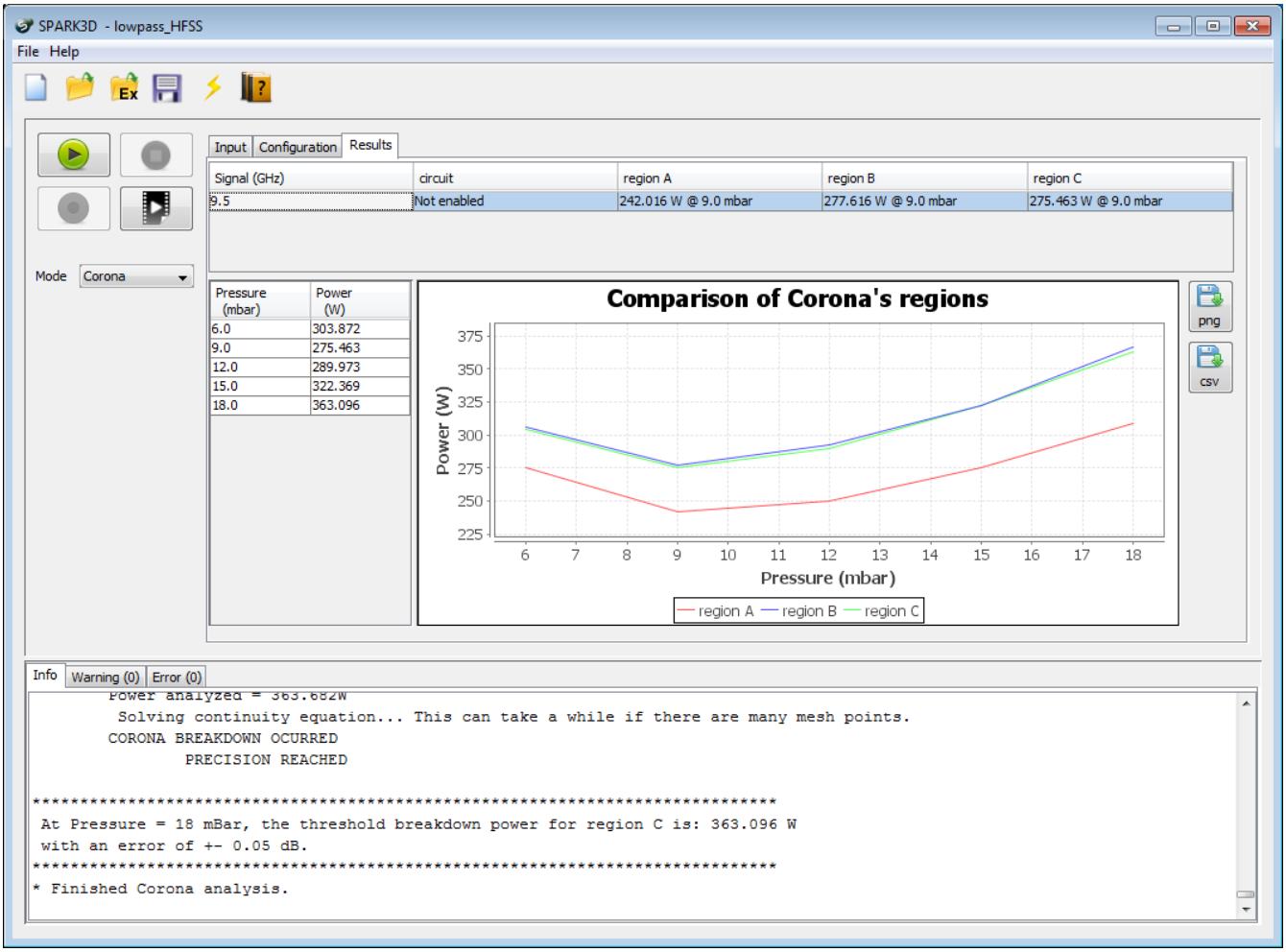
Corona results can be consulted in run-time. For the particular example followed in the tutorial, lowpass_HFSS.spk, Corona results window looks as follows during the simulation:



There are two tables and one graph that are refreshed as new results are obtained:

1. The left-side table contains the threshold breakdown power for each pressure point corresponding to a certain region. If both the high pressure analytical rule and the numeric simulation type (see Corona Analysis Manual for further explanation) have been selected for evaluation, the table will have three columns instead of two, where the last one corresponds to the analytical rule.
2. In the graph is represented the Paschen curve corresponding to the data showed in the left-side table. If both the high pressure analytical rule and the numeric simulation type (see Corona Analysis Manual for further explanation) are enabled, there will be two curves, one corresponding to the numerical analysis and the other one to the analytical rule.

3. The table located on the top of the window shows the minimum breakdown power in the whole pressure sweep for each region analyzed. During the simulation process, the word "Simulating" appears in the cell corresponding to the region which is being analyzed. When a certain region has been not enabled for analysis, the message "Not enabled" is shown in its cell. Besides, through this table the user can handle the results shown both in the left-side table and the graph:
   - By left-clicking on a cell corresponding to a particular region both the graph and the left-side table update their values to the current element.
   - By left-clicking on the cell corresponding to the signal value, the whole row is selected and the graph shows together the Paschen curves of all the regions analyzed. With this information it is easy to recognize which is the most critical region for Corona discharge and the minimum breakdown power supported by the device.



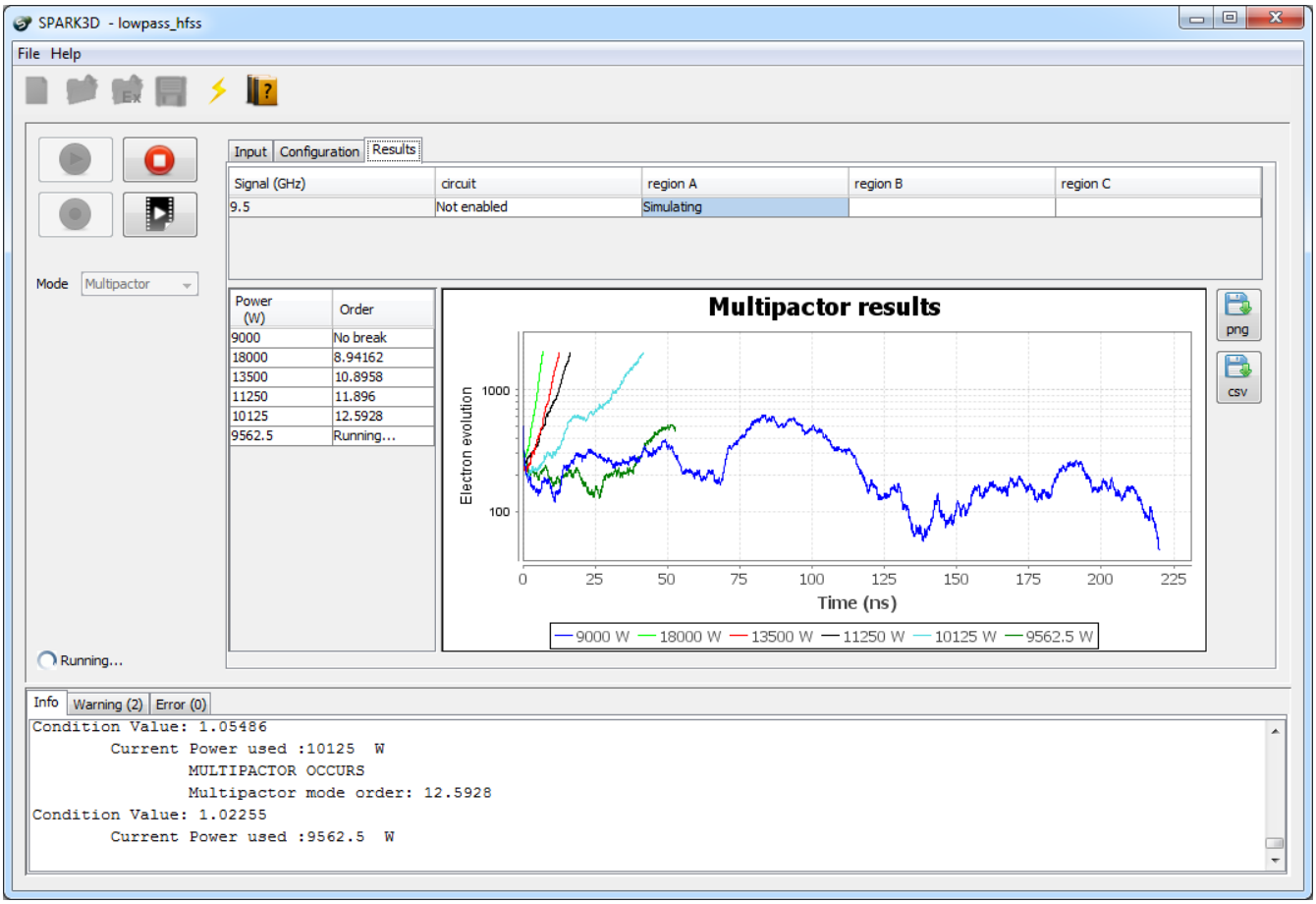
Using the buttons 

and 

you can save the graph in a .png file and its data, which corresponds to the left-side table, in .csv format, respectively.

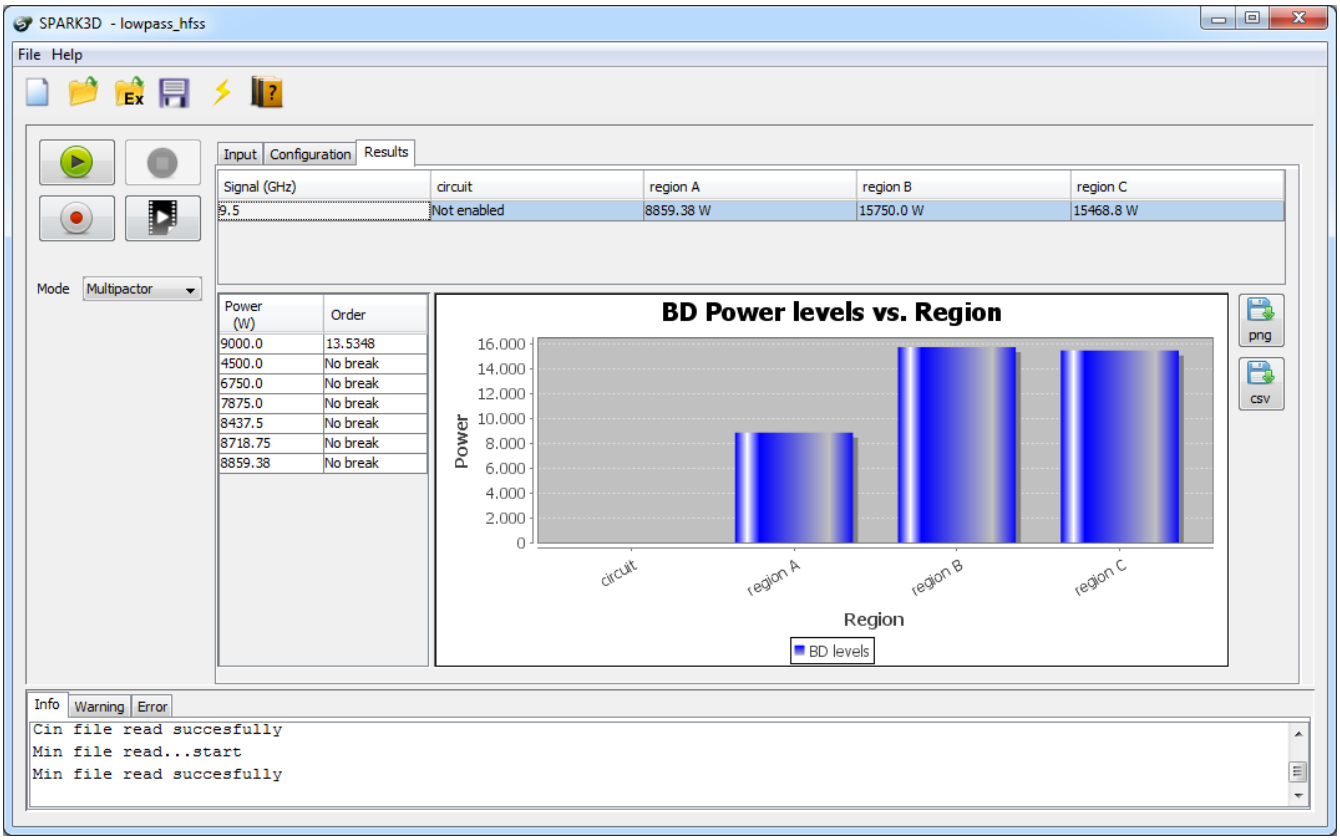
Multipactor results window

Multipactor results are given both in tabular and graphic form. They can be seen in run-time through the results window, which for the example treated in the tutorial looks as follows:



There are two tables and one graph:

1. The left-side table shows for each analyzed power whether there has been breakdown or not. When breakdown occurs for a certain input power, the multipactor order is given in the second column of the table whereas when there is no breakdown the message "No break" appears.
2. In the graph it is represented the electron evolution with time for each power analyzed. In this way it is easy to follow the increase/decrease of the electron population as the simulation runs. When left-clicking on the cell corresponding to a certain power of the left-side table, its corresponding curve is highlighted on the graph for a better recognition.
3. The upper table contains the threshold breakdown power for each region under study. Through this table the user can handle the results shown both in the left-side table and the graph:
   - By left-clicking on a cell corresponding to a particular region both the graph and the left-side table update their values to the current element.
   - By right-clicking on a cell corresponding to a particular region an option "visualize in paraview" appears. This option launches a paraview window and shows the position of the electrons in the structure, and the 3D stats (if enabled in the configuration tab).
   - By left-clicking on the cell corresponding to the signal value, the whole row is selected and a bar diagram appears in the graph comparing the threshold breakdown power for all regions. With this information it is easy to recognize which is the most critical region for Multipactor and the minimum breakdown power supported by the device.



Using the buttons 

and 

you can save the graph in a .png file and its data, which corresponds to the left-side table, in .csv format, respectively.

2.3 SPARK3D Manual

This section describes the structure of SPARK3D and documents the features of each subsystem SPARK3D is composed of (Graphical User Interface, Multipactor module, Corona module).

The SPARK3D manual contains the following topics:

Architecture	The top-level architecture of SPARK3D
Requirements	The minimum hardware and software requirements needed to run SPARK3D.
Graphical User Interface (GUI)	Description of the Graphical User Interface, its features and how to use it
Corona analysis	Description of the corona module
Multipactor analysis	Description of the multipactor module

Architecture

SPARK3D is a general software tool for Radio Frequency (RF) breakdown analysis, which allows predicting both Corona and Multipactor breakdown onsets in a great variety of RF structures. It has as input data the electromagnetic field distribution of the device under study at a single frequency. It allows the user defining the regions where the High Power analysis will be carried out and perform the result visualization using an intuitive, user-friendly graphical interface.

At the top-level, SPARK3D is composed of two subsystems:

- Graphical User Interface (GUI)
- High Power Computational Engine (HPCE), which includes Multipactor and Corona modules.

The GUI is a Java application. It is the part of SPARK3D program in charge of interacting with the user, also executes and coordinates the other subsystems at user's demand and represents the results data in tabular and graphic forms.

The HPCE implements the high power capabilities of SPARK3D. The HPCE is designed and tuned for performance and

exploits state-of-the-art techniques in multipactor, corona and information technology research fields.

Requirements

SPARK3D requires at least the following:

- Hardware: Dual core with 2GB of RAM and 3GB free disk space.
- Operating System: Windows 7, Windows 8. Special requests for Linux or other Windows versions.

Anyway, hardware requirements may vary: very complicated structures and high numeric accuracy can consume **much** more memory and computational time. It is recommended to have a 64 bit operative system and at least 4GB of RAM

2.3.1 Graphical User Interface (GUI)

This section describes the architecture of SPARK3D Graphical User Interface (GUI), documents its features and how to use it.

The GUI section contains the following topics:

Features	Description of GUI features and capabilities.
The Main Window	How to use the GUI to load a SPARK3D project.

Features

The GUI allows the user to interactively access all functionalities using mouse and dialogs:

- Projects can be created or loaded
- The Multipactor and Corona modules can be configured using dialogs and activated with button clicks
- The progress status of simulations is reported in real time, including graphical visualization in the results window
- The 3D CAD viewer program, Paraview, is used to visualize the imported EM fields and the regions defined by the user for simulation.

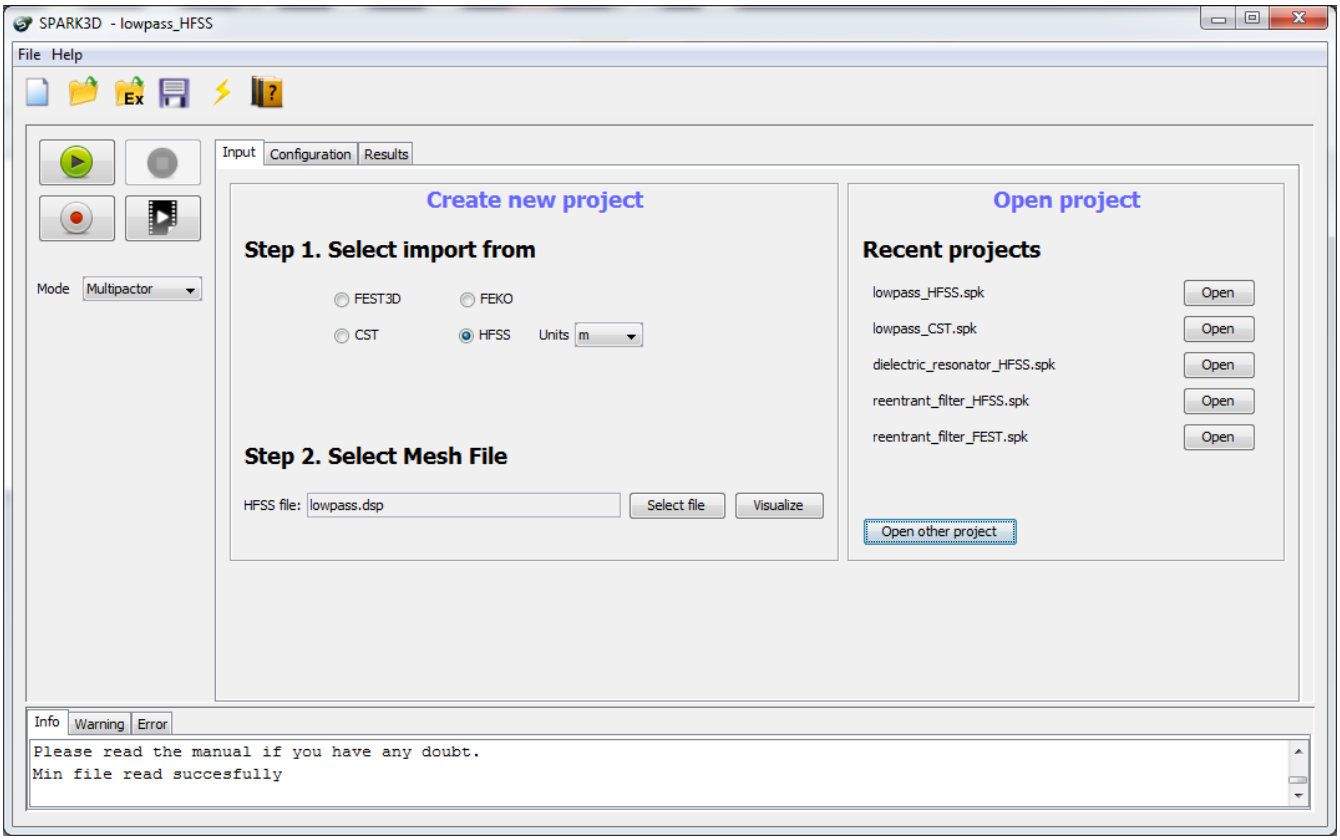
The Main Window

This section describes the SPARK3D Main Window and how to use it to load a project, select between multipactor or corona operation modes and open other facilities of SPARK3D.

The Main Window section contains the following topics:

Menubar	The top menu bar with standard commands: Load, Save, Exit and also SPARK3D help.
Toolbar	The toolbar on the top, containing buttons for frequently used Menu commands.
Output messages window	Messages are shown in different window tabs, depending on its type: Info, Warning, Error.

The FEST3D Main Window typically looks as follows



From the Main Window it is also possible to select the operation mode: Multipactor or Corona (arcing) modules, through the pull-down menu **Mode**.

In the left side of the main window, the buttons play  and stop , make the simulation start and finish correspondingly.

The buttons record  and play video  , are only enabled in multipactor mode and can be used to start a video recording or to visualize a previously stored animation. See Multipactor Analysis and Video Tutorial for further information.

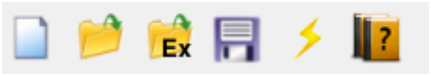
Menubar

The menubar contains the following menus:

1. **File**
   - **New project:** lets the user create a new project from the beginning.
   - **Load project:** a browsing dialog box for project selection appears. By default, the user can choose among *.spk files.
   - **Save project:** stores the project with the name defined before (written at the top of the window).
   - **Save project as:** stores the project with a new name, this name becomes the new current name.
   - **Exit:** ends the program (closing all windows) asking the user to save modifications if not previously saved.
2. **Help**
   - **About** shows SPARK3D version information.
   - **Help** opens SPARK3D Online Help.

Toolbar

The toolbar is the horizontal row of buttons at the top of the window. It duplicates the most frequently used menu commands, allowing to perform the basic functions: load project, open examples, save and help. The following figure shows the toolbar as it typically appears on the screen



From the button  of the toolbar, SPARK3D offers the possibility to access to an extra functionality: ECSS Multipactor Tool.

Input tab

The input tab is common for Multipactor and Corona Analysis and it allows for either importing a new mesh file or opening an existing project.

There are two steps to load a new mesh file.

Step 1: Select import from	Here the user defines the input format of the mesh. It can be chosen from the following: • **CST:** This option imports CST Microwave Studio® files in .f3e format. • **HFSS:** This option imports ANSYS® HFSS™ mesh files in .dsp format. The units must match with those of the HFSS model. • **FEKO:** This option imports FEKO files in .fse format. • **FEST3D:** This option imports mesh files generated by FEST3D in .mfe format.
Step 2: Select mesh file	By clicking on **select file** button an open file dialog appears where the user can select the input mesh according to the file format previously selected. The button **visualize** opens paraview and shows the mesh and fields of the selected file.
Recent Projects	A list of the last five opened projects is shown. The user can open any of them.

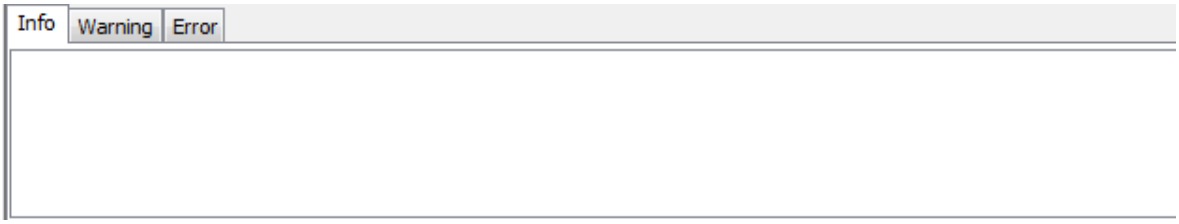
See the tutorial EM field exportation for more information on the different input mesh formats.

Output messages window

The GUI shows the simulation output messages in three different tabs of the output window. Depending on the type of message, it is shown in:

- the Info tab
- the Warning tab
- the Error tab

which look as follows



2.3.2 Multipactor Analysis

The Multipactor analysis section contains the following topics:

Definition	What is exactly done when using this SPARK3D feature.
Features	List of Features and New features included in this release.
Limitations	Some limitations of the Multipactor module.
Errors	The possible errors produced when performing the Multipactor analysis, and solutions or workarounds to them.
Using the Multipactor module	How to use this feature in SPARK3D.
Output of a Multipactor simulation	Description of the output of a multipactor simulation.
Hints	Non-trivial properties of the use of the Multipactor module.

Definition

The Multipactor analysis computes the multipactor breakdown power threshold of one or more particular regions of the structure. It supports single carrier operation.

For a more detailed information about multipactor theory and results see:

C. Vicente, M. Mattes, D. Wolk, H. L. Hartnagel, J. R. Mosig, and D. Raboso, "FEST3D: A simulation tool for multipactor prediction," *in Workshop on Multipactor, RF and DC Corona and Passive Intermodulation in Space RF Hard- ware*, pp. 11–17, ESTEC, Noordwijk, The Netherlands, Sept. 12-14 2005.

S. Anza, C. Vicente, B. Gimeno, V. E. Boria, and J. Armendariz, "Long-term multipactor discharge in multicarrier systems," *Physics of Plasmas*, vol. 14, pp. 082112–082112–8, Aug. 2007.

S. Anza, C. Vicente, D. Raboso, J. Gil, B. Gimeno, V. E. Boria, "Enhanced Prediction of Multipaction Breakdown in Passive Waveguide Components including Space Charge Effects", *in IEEE 2008 International Microwave Symposium* , June 2008, Atlanta (Georgia), USA.

E. Sorolla, S. Anza, B. Gimeno, A.M. Perez, C. Vicente, J. Gil, F.J. Perez-Soler, F.D. Quesada, A. Alvarez, V. Boria, "An analytical model to evaluate the radiated power spectrum of a multipactor discharge in a parallel- plate region", *IEEE Transactions on Electron Device Letters* , vol.55 no. 8, pp. 2252-2258 Aug. 2008.

S. Anza, C. Vicente, J. Gil, B. Gimeno, V. E. Boria, and D. Raboso, "Non-stationary Statistical Theory for Multipactor," *Physics of Plasmas*, vol. 17, June 2010.

Features

- Custom SEY curves. Possibility of using Predefined SEY materials (according to ECSS standards), user defined parameters or import from text file.
- Computation of electron evolution for each applied input power.
- Automatic multipactor threshold determination.
- Advanced 3D output statistics with average impact energy, average SEY, and emitted electron density for the different surfaces in the structure.
- Possibility to add external uniform DC magnetic field.
- Electron path algorithm with adaptive refinement which allows for faster and more accurate simulations. The electron trajectories are now computed with a certain error introduced by the user.
- Different multipactor criteria. The multipactor criteria allows for automatically stop the simulation and decide whether there is multipactor discharge or not. The election of one or another have implications on the accuracy and speed of the simulation. The user can easily change the criteria from the configuration window. The available criteria are: charge (automatic), charge (fixed factor) and charge trend.
- Impact angle dependence for SEY curves imported from text files.
- Multipactor video recording feature. The user can export videos of electrons moving in a 3D structure and open them at any time. 3D rotations, perspective customization and zoom are allowed on recorded animations. Final export to popular video formats (such as .avi) can also be done.
- Automatic power loop, in which input power levels are automatically computed to find the multipactor threshold, and Custom power loop, in which the user can specify as many arbitrary input power levels as desired
- Multipactor analysis can be run on the entire imported mesh or on different user defined regions to speed up the simulation.

New features

- Now, in addition to the magnetic case, homogeneous electric DC field can be added to the simulation.
- Arbitrary external electric and/or magnetic DC fields, with ANSYS® MAXWELL™ or rectangular CSV mesh formats, can be imported and incorporated to the simulation.
- New SEY default properties for Aerospace Corporation aluminium have been added (TOR-2014).

Limitations

Presently, all surfaces in the problem are considered to have the same SEY properties. This is, regarding to Secondary Emission Properties, there is no distinction between different metals or dielectrics within the same problem. They all will be assigned a common SEY curve defined by the user.

Due to numerical limitations on the electron path integration, in rare cases and for very high fields, false single-surface discharges may occur at very low multipactor orders (below 0.05). These are easily identified and must not be taken as real discharges. If this occurs, please contact technical support for possible solutions to this issue.

Errors

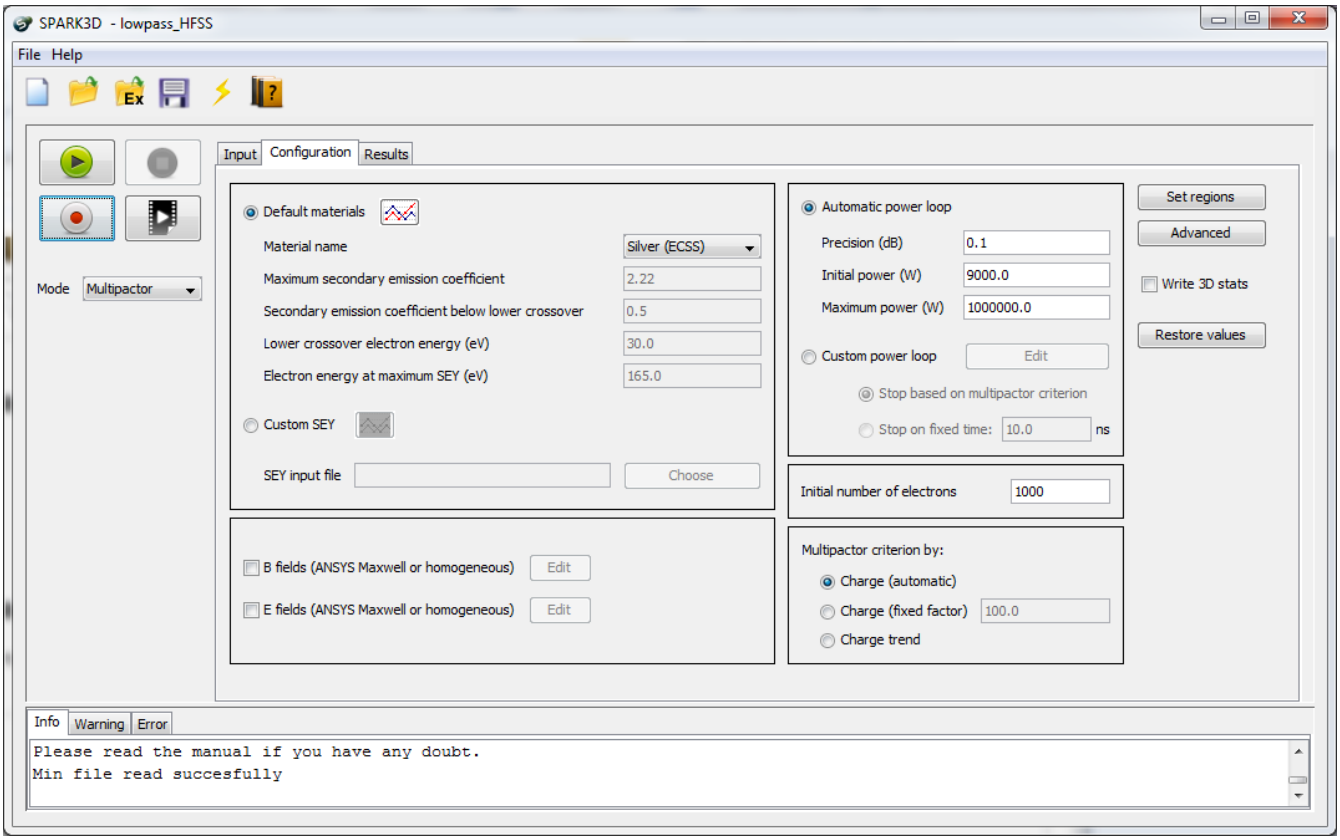
Due to the nature of the phenomenon, the results can slightly differ from simulation to simulation. This deviation can be considered an intrinsic error caused by the phenomenon itself. However, this error is normally so small that it is not relevant for practical applications.

Using the Multipactor module

The multipactor discharge analysis of the device under simulation is controlled through the GUI, that allows easily setting the input parameters.

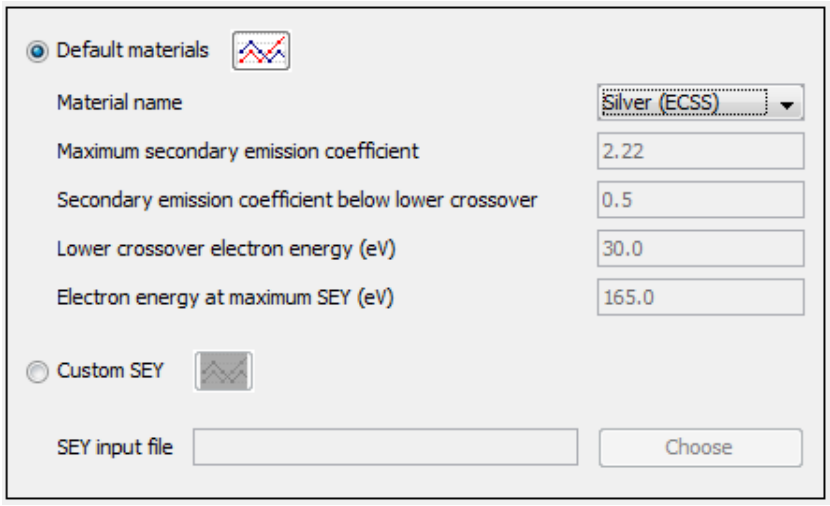
Configuration tab

The configuration window permits setting the rest of the simulation parameters as is shown in the next menu:



Material

Allows you to choose metals with well studied Secondary Electron Yield (SEY) properties. It also allows you to create new materials and save them for future simulations.



Material name Six materials are included with their SEY properties. User defined materials can be saved and loaded.

Maximum secondary Maximum SEY of the material. Typical values are between 1.5 and 3.

emission coefficient	
Secondary emission coefficient below lower crossover	SEY of elastically reflected electrons at low impact energies.
Lower crossover electron energy (eV)	The lowest electron impact energy at which the SEY crosses the value of 1. This is a typical value between 25 and 45 eV for the **materials** for space applications.
Electron energy at maximum SEY (eV)	The electron impact energy at which the SEY is maximum. Typical values are between 150 and 300 eV.

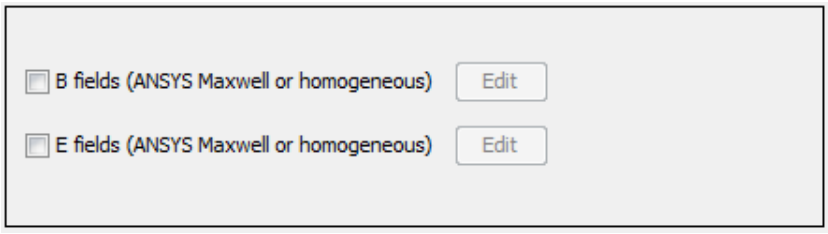
It is also possible to use a custom SEY by importing it from an input file. The file must be in CSV (comma-separated-value) format, which is text file with .csv extension that consists on tabulated data. The SEY file should have 2 columns: the first one contains the electron impact energy in eV and the second one corresponds to the SEY of the material at normal incidence. SPARK3D will automatically add the angle dependence for each electron impact. For energies outside the range defined in the input file, the SEY will be set to 0.

Press the button with the icon  close to a SEY definition to open a new window with a plot of the selected SEY curve.

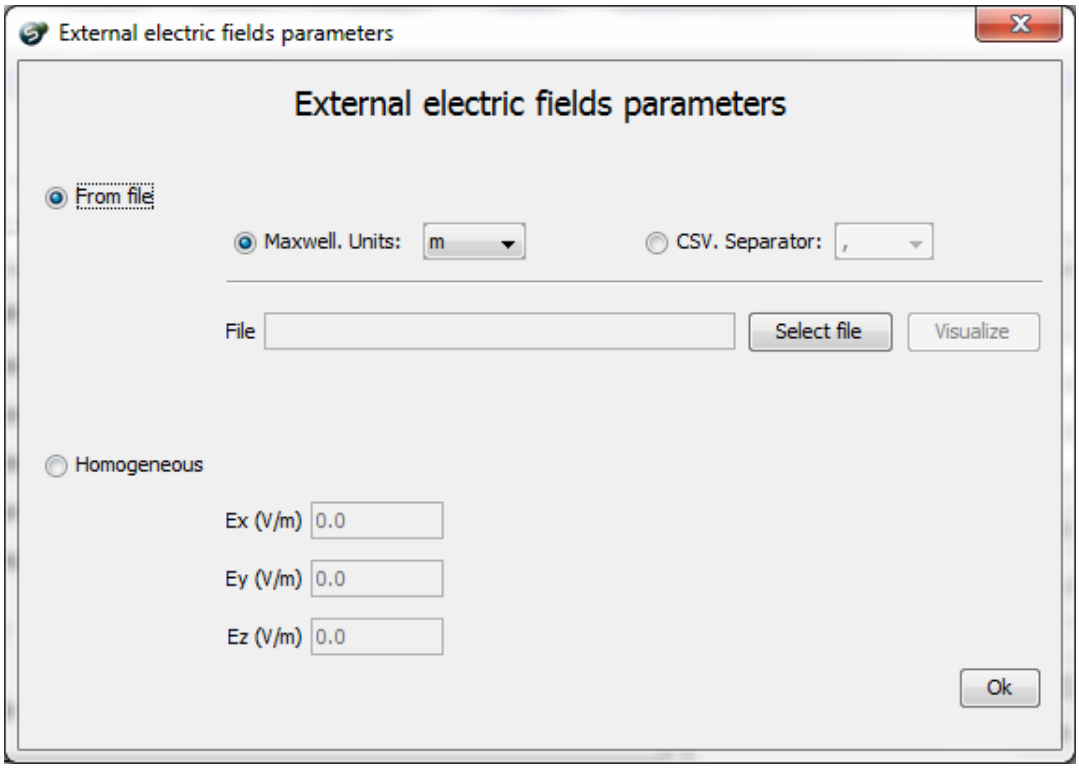
The selection and definition of the SEY curve has an important effect on the multipactor simulation. See some practical considerations when selecting the material properties.

DC fields

By selecting the B fields or the E fields check boxes, external DC fields are added to the simulation.



Clicking on the edit button an advanced dialog is opened (similar for electric and magnetic cases).



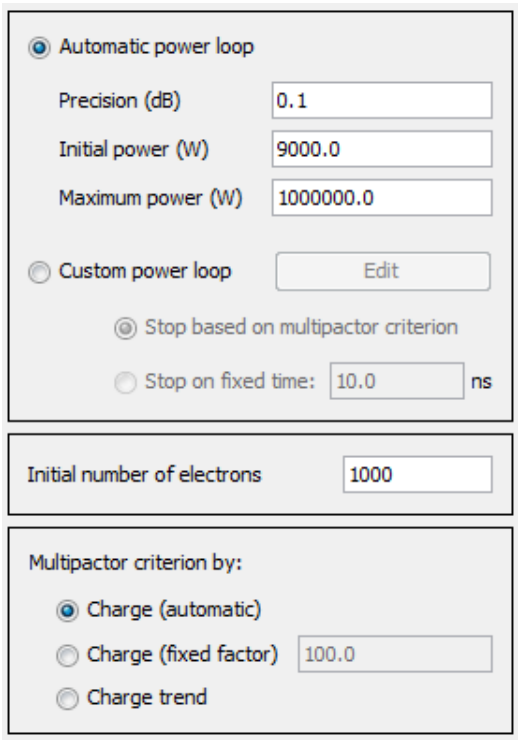
From File Selecting this option, the DC field values are imported from an external mesh file. Two options are available (see EM field exportation):

- **Maxwell:** ANSYS® MAXWELL™ Projects can export their results to .dsp format and then imported into SPARK3D. Units must be properly chosen to match those used in the MAXWELL™ project.
- **CSV:** Structured rectangular meshes in CSV format can be imported. The separator can be arbitrarily chosen. The file must follow a six column format specifying the x, y and z coordinates of the mesh node (first 3 columns) and the x, y and z values of the DC field (last three columns).

Once imported, the values of the DC fields will be interpolated to the RF mesh node locations. Therefore, it is important that the DC field mesh has non-zero volume intersection with the RF field mesh.

Homogeneous Uniform external DC fields are added to the simulation. The uniform B and E DC fields are given in Tesla and V/m respectively.

Simulation preferences



Automatic power loop If selected, the multipactor module will search automatically for the multipactor threshold, starting from the initial power and stopping when the desired precision is reached. Bisection method is employed, and the multipactor criterion (to determine if there has been a discharge or not) is set by the Multipactor criterion menu below. The parameters are:

- **Precision (dB):** This parameter sets the precision in power level desired for the multipactor breakdown onset. The default is 0.1 dB.
- **Initial power (W):** This will be the initial input power used to search the multipactor breakdown onset. This can be changed to an input power level close to the final breakdown onset if some information is known a priori.
- **Maximum power (W):** Sets the maximum allowed power for multipactor breakdown search. If the simulation reaches this power and no multipactor is observed the element is considered multipactor free. The default is 100 kW.

Custom power loop If selected, the input power steps are selected by user by pressing the edit button, see the figure below. A multipactor simulation will be done for each step. The criterion for stopping the simulation can be chosen from:

- **Stop based on multipactor criterion**: The simulation will stop if a discharge (or not discharge) is detected, using the selected criterion in the menu below.
- **Stop on fixed time**: The simulation time is fixed, no matter if there is a discharge or not, unless the number of electrons decrease to 0, or reach the maximum allowed number of electrons (1e15 for numerical stability reasons).

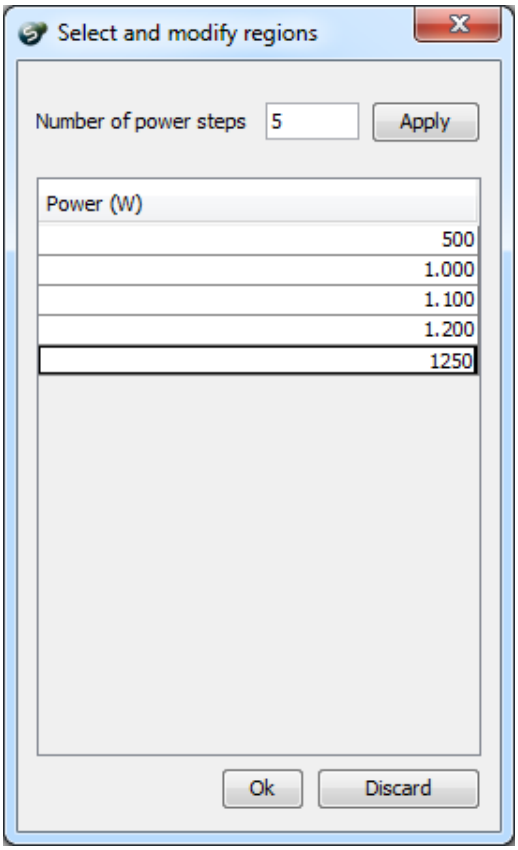
Initial number of electrons This defines the initial number of electrons launched in a particular component element. This number can vary in order to obtain reliable results. The default value of 100 electrons should be quite accurate in single-carrier mode and in waveguide elements where the parallel plate approximation holds. However, if the length of the waveguide element is of the order of its height more electrons could be necessary. For a complete simulation, the best idea is to start with a low number of electrons in order to get a fast idea of the approximated breakdown power level. After that, more electrons can be launched using an input power level close to the one obtained in the simulation with few electrons.

Multipactor Multipactor criterion is the mechanism that automatically decides if there is a discharge or not at a

criterion by: certain input power and stops the simulation. Then, next power is simulated until the precision is reached. There are three different criteria, all of them based on the electron population:

- **Charge (automatic):** This is the default mode. At each RF half-cycle, the ratio between the current number of electrons and the initial ones are checked. This criterion establishes a factor depending on the current number of simulated half-cycles. If the number of electrons is above such a factor, multipactor is detected. Basically, it sets higher factors for lower number of half-cycles (beginning of the simulation) and more relaxed ones for larger number of half-cycles (longer simulations). This is done in order to avoid false detection during the initial stages of the simulation. Additionally, if after a certain number of cycles, the ratio is below a certain number, the simulation is stopped and no multipactor is detected. This is done in order to avoid excessively large simulations in which there is not a clear electron growth.
- **Charge (fixed factor):** It is equivalent to the automatic one, but the factor is not automatic but set by the user. Gives more control on the simulation but needs of more trim and knowledge from the user side. It does not have any check for low number of electrons. Only populations decreasing to zero are considered no discharges. Therefore there is a risk of long simulations.
- **Charge trend:** It fits the electron evolution to a exponential curve and checks whether there is positive or negative growth. It detects both discharges and no discharges. In general, this method detects multipactor much faster than the others. However, it may suffer from higher variability between consecutive simulations. In such cases, it is advisable to use a high number of initial electrons.

If Custom power loop is selected, the user can choose the input power steps by pressing the edit button. The following window appears.



Other simulation options

Write 3D stats It writes advanced statistics in paraview mesh format that can be visualized from the results tab (see **output** section):

- **Average SEY:** It shows the average SEY of the impacting electrons in each surface of the mesh.
- **Average Impact Energy:** It shows the average impact energy of the impacting electrons in each surface of the mesh.
- **Impact Density:** It shows the electron impact density (impacts/m2) for each surface of the mesh.
- **Emission Density:** It shows the electron emission density (emitted electrons/m2)for each surface of the mesh. It can be positive (more electrons were emitted than absorbed) or negative (more electrons were absorbed than emitted).

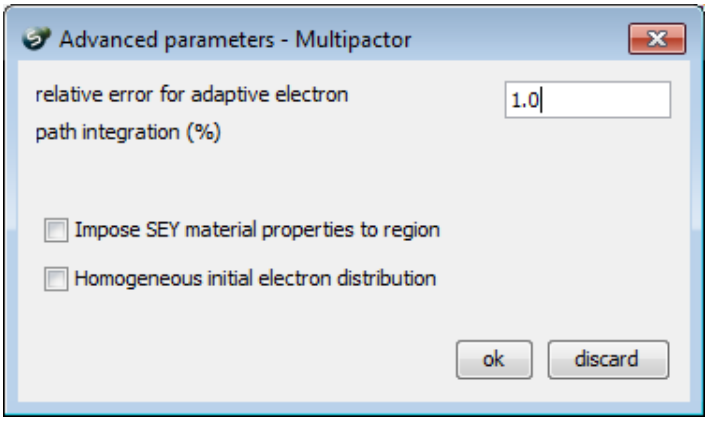
Set regions Through this option the user can define the specific regions of the structure where the analysis will be carried out. For a detail description of its usage see specifying regions section of the tutorial. When regions

are specified, all non-metallic surfaces (corresponding to vacuum condition) of the mesh enclosed in a specific region are considered as open boundaries. As a consequence, all electrons impacting with such regions are automatically absorbed.

Restore values Gives back to the configuration parameters used in the last simulation.

Advanced Dialog

The Advance Dialog allows for setting extra simulation parameters which are not usually needed for typical simulations but that provides extra control for advanced users.



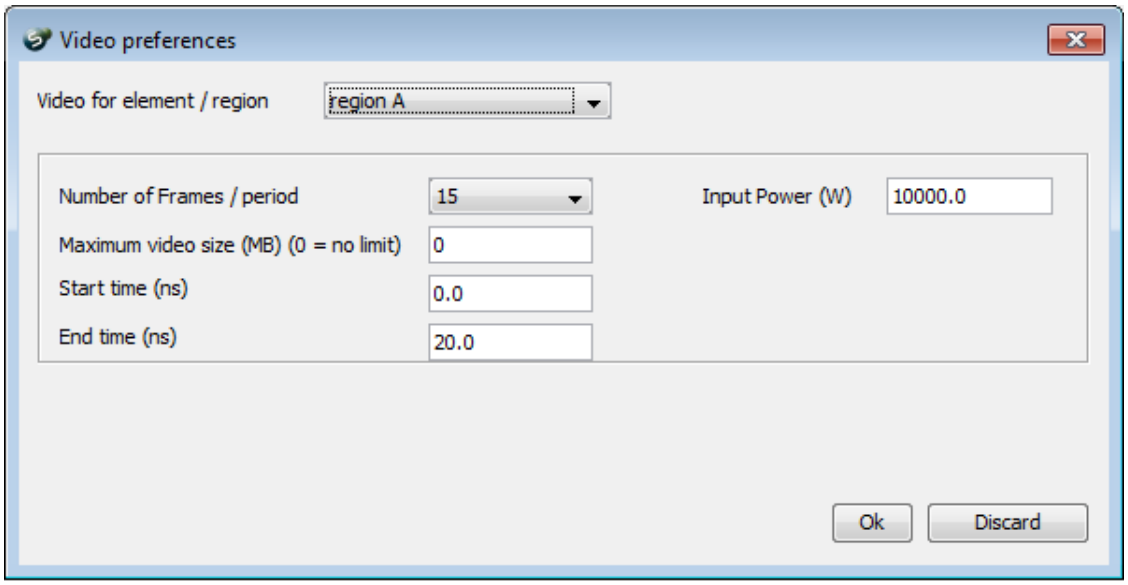
The Advance Dialog allows for setting the following parameters:

- **Relative error for adaptive electron path integration**: This parameter specifies the maximum error in the electron path integration. The SPARK3D electron tracker incorporates an automatic step refinement for each electron at each time step. This implies that the integration step for electrons in high field regions will be smaller than for those in low field regions, ensuring a maximum error for all of them. This process is iterative. Large values imply less accurate simulations but less adaptive iterations and thus faster simulations. Small values imply more accurate but slower simulations. The default value (1%) is normally a good trade-off for most cases.
- **Homogeneous initial electron distribution**: Normally, initial electrons are located on high electric field locations on metallic surfaces. If this option is checked, initial electrons will distribute uniformly on all metallic surfaces. This can be useful in situations where high electrical field is present in reduced areas (metal edges or corners) and multipactor is known to occur in other places.
- **Impose SEY material properties to regions**: If checked, the boundaries of selected regions will be considered as solid surfaces in all their extent (instead of opening the boundaries that correspond to vacuum) . This consideration does not affect the EM fields (since they are computed with an external solver). It only has implications on electron interaction with surfaces and SEY. This option may be useful to model some kind of special interfaces such as dielectric-vacuum (since current version of SPARK3D does not support dielectrics yet).

Video Record

If multipactor mode is selected videos can be saved by pressing the record button  and opened at any time with the play button 

The record video dialog is the following



Video for element / region	Here, the region in which the video is going to be recorded is selected
Number of Frames / period	Specifies the frame rate of the recording. The higher the smoother the animation, but bigger video sizes will be generated.
Maximum video size (MB)	If different from zero, the video will be cropped when the video file (approximately) exceeds the set size.
Start time (ns)	Sets the initial time for video recording.
End time (ns)	Sets the maximum time for video recording.
Input Power (W)	Sets the input power for this specific video recording.

Other parameters are taken from current configuration, such as SEY definition, number of electrons, multipactor criterion, single or multi-carrier simulation, etc . Autofill button completes the fields based in the last simulation (using results such as breakdown power, multipactor order, etc.

When the Ok button is pressed, the an output file must be selected (*.v3d extension) and the simulation starts. Results from previous simulations will not be deleted. The video will be stored outside of the project and it is independent of it (can be opened from other projects, for example).

Once the simulation is finished, the user can select to immediately open the video, or to open it at any time with the play video button  . The videos are opened with the 3D CAD viewer software Paraview, which allows for 3D rotations, perspective customization and zoom on the saved animations. It also allows for exporting the animation to popular video formats, such as avi format for instance. See the Video Tutorial for further information.

Running Multipactor Analysis from command prompt

It is also possible to execute Multipactor Discharge Analysis from command prompt. The executable name is *spark3d.exe* on Windows platform and *spark3d* on Unix-like platforms, and is located in the directory where SPARK3D is installed.

Before running the simulation from command prompt, a text file must be created to keep all configuration parameters for Multipactor simulation. The extension of the file must be *.min* and its format looks as follows:

```
begin "multipactor"
  begin "regions"
    num_regions             2
    not_analyze_all_when_simulate       1
    begin "region"
      region_type           "box"
      region_name           "circuit"
      region_number         1
      analyze               0
      center_x              0.0
      center_y              0.0
      center_z              0.0
      size_x                0.0
      size_y                0.0
      size_z                0.0
    end "region"
    begin "region"
      region_type           "box"
      region_name           "region A"
      region_number         2
      analyze               1
```

```
      center_x              0.0
      center_y              0.0
      center_z              0.023
      size_x                0.03
      size_y                0.03
      size_z                0.01
    end "region"
  end "regions"
  begin "power_steps"
    num_steps               1
                0.0
  end "power_steps"
  multipactor_criterion_sc              "charge_automatic"
  criterion_fixedfactor_sc              100.0
  multipactor_criterion_mc              "charge_fixed_factor"
  criterion_fixedfactor_mc              1.0E7
  material_name             "silver"
  ext_DC_BField             0
  ext_DC_Bfield_type        "maxwell"
  ext_DC_Bfield_file        ""
  ext_DC_Bfield_units       "m"
  import_delimiter_B        ","
  B_DC_x        0.0
  B_DC_y        0.0
  B_DC_z        0.0
  ext_DC_EField             0
  ext_DC_Efield_type        "maxwell"
  ext_DC_Efield_file        ""
  ext_DC_Efield_units       "m"
  import_delimiter_E        ","
  E_DC_x        0.0
  E_DC_y        0.0
  E_DC_z        0.0
  write_3D_stats            0
  m_pdf_report              0
  maximum_secondary_emission_coefficient            2.22
  secondary_emission_coefficient_below_lower_crossover          0.5
  lower_crossover_electron_energy       30.0
  electron_energy_at_maximum_SEY        165.0
  initial_number_of_electrons           1000
  sc_precision              0.1
  sc_initial_power          0.4
  sc_max_power              1000000.0
  iteration_type            "bisection"
  custom_fixed_time         0
  custom_max_time           1.0E-8
  mc_precision              0.1
  mc_initial_power          -1.0
  mc_max_power              1000000.0
  SEE_statistics            "maxwellian_velocity"
  path_rel_precision        1.0
  homogeneous_emission      0
  metallic_contours         0
  videoNumFrames            15
  videoMaxSize              0
  videoStartTime            0.0
  videoEndTime              1.0E-8
  videoInputPower           100.0
  videoRegionsLabel         "1"
  videoOutputPath           "Select output file"
  sey_type      "vaughan"
end "multipactor"
```

All these parameters are described in the **Usage** of Multipactor Discharge from User Interface and in the Specifying Regions section.

Executing the command *spark3d -h* (prefixed by SPARK3D installation directory) will show all command-line arguments and options supported, including how to specify input and output files.

A typical invocation of Multipactor Discharge Analysis looks as follows:

```
<full-path-to-spark3d/spark3d.exe> --mode=multipactor --project_path=<working-directory> --tmp_path=<relative-path-to-temporal-directory> --config_file=<relative-path-to-config-directory/mycircuit.min> --output_path=<relative-path-to-output-directory> --data_file=<relative-path-to-mesh-directory/EM_fields_file> --file_type=<fest,cst,hfss>
```

In the case of an HFSS file, it is also mandatory to include an extra argument:

```
--HFSS_units=<m,mm,inch>
```

See the example below:

```
C:/Program Files (x86)/SPARK3D1.5.1FULL/dist/spark3d --config_file="paramsFolder/conf1.min" --output_path="outputFolder" --mode=multipactor --project_path="Libraries/Documents/SPARK3D_Example/Dielectric_resonator" --HFSS_units="m" --tmp_path="tempFolder" --data_file="meshFolder/dielectric_resonator.dsp" --file_type=hfss
```

The **full** path to SPARK3D executable is required since SPARK3D will search for the license file in the same directory as the full path specified.

The project_path indicates the folder where the other directories are located. In the example above, *meshFolder*, *paramsFolder*, *outputFolder* and *tempFolder* are located inside Dielectric_resonator directory and relative paths are given with respect to it.

It is also necessary to indicate a directory where temporal files will be stored (`tmp_path`) and a directory where the results will be kept (`output_path`).

The argument `data_file` corresponds to the file where the EM fields have been exported, which in the example is stored in *meshFolder*. The argument `file_type` refers to the type of EM software used to export the EM fields and `HFSS_units` indicates the units used in the HFSS project to define the dimensions.

The possibility to launch SPARK3D from command line allows using it in combination with external programs. This way, scripts to launch several executables sequentially can be generated or it can be combined with a synthesis or optimization tool.

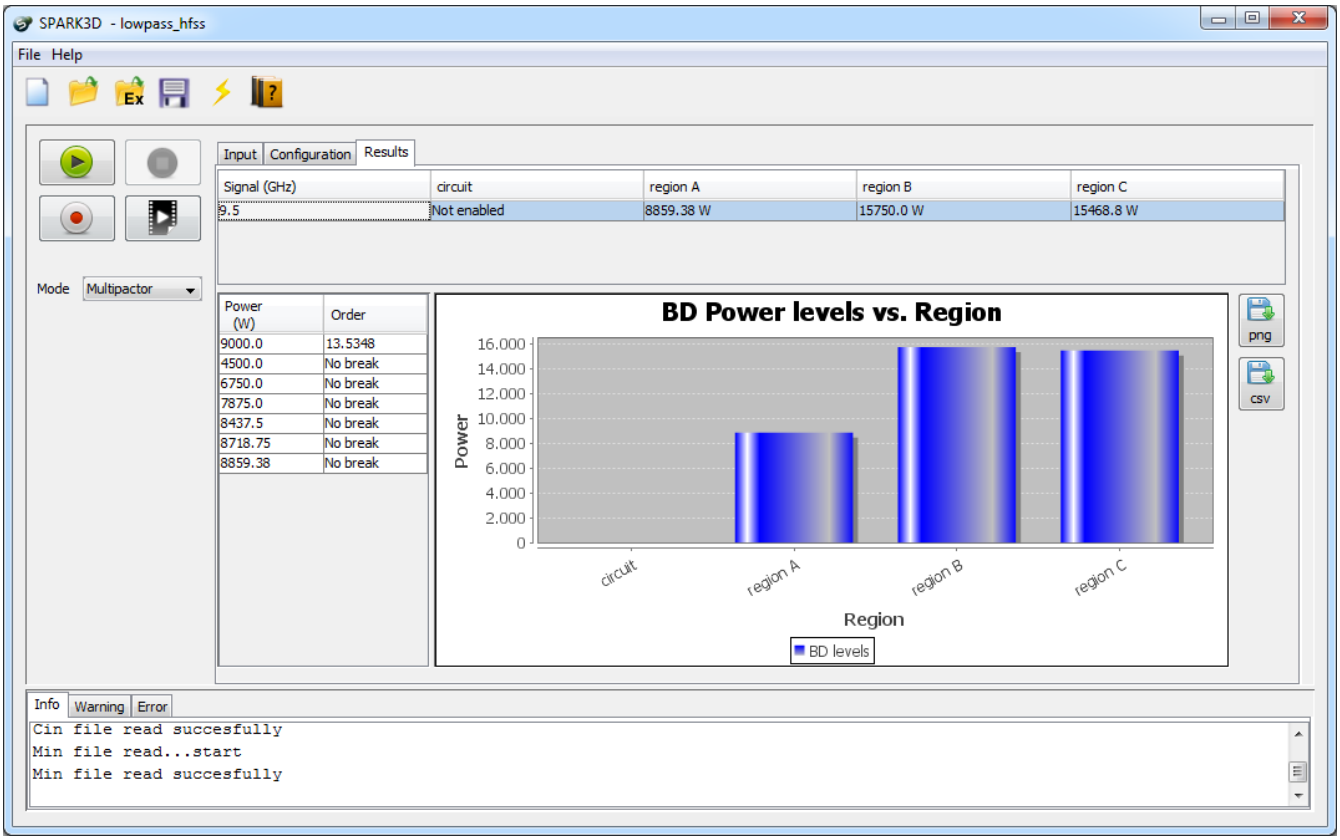
Output

The multipactor module provides the input power breakdown threshold per carrier of the selected regions of the structure. The simulation process can be visualized in the info window of the main SPARK3D canvas, where a sweep in input power is shown as the simulation runs, showing how the simulator tries to approach the breakdown threshold.

Multipactor results are given both in tabular and graphic form. They can be seen in run-time through the results window, which looks as follows:

There are two tables and one graph:

1. The left-side table shows for each analyzed power whether there has been breakdown or not. When breakdown occurs for a certain input power, the multipactor order is given in the second column of the table whereas when there is no breakdown the message "No break" appears.
2. In the graph it is represented the electron evolution with time for each power analyzed. This way it is easy to follow the increase/decrease of the electron population as the simulation runs. When left-clicking on the cell corresponding to a certain power of the left-side table, its corresponding curve is highlighted on the graph for a better recognition.
3. The upper table contains the threshold breakdown power for each region under study. Through this table the user can handle the results shown both in the left-side table and the graph:
   - By left-clicking on a cell corresponding to a particular region both the graph and the left-side table update their values to the current region.
   - By right-clicking on a cell corresponding to a particular region an option "Show 3D Stats" appears. This option launches a paraview window and shows the position of the electrons in the structure, and the 3D stats (if enabled in the configuration tab (see **Usage**)).
   - By left-clicking on the cell corresponding to the signal value, the whole row is selected and a bar diagram appears in the graph comparing the threshold breakdown power for all regions. With this information it is easy to recognize which is the most critical region for Multipactor and the minimum breakdown power supported by the device.



The data represented in the graph can be saved into a .png file through the button 

or into a .csv file through the button 

.

Hints

- To speed up the simulation use the multipactor module with the minimum accuracy possible to have a rough idea about the breakdown level.
- Set the multipactor criterion to charge-trend. This will speed up the simulations significantly. Only if high variability is found between simulations change back to charge (automatic), or charge (fixed factor) criteria.

2.3.2.1 Multipactor Practical Considerations

Secondary Emission Yield (SEY)

The multipactor discharge is a complex physical phenomenon which is strongly related to many factors. Concretely, the most important one is the **Secondary Emission Yield (SEY)** of the surfaces of the device.

The correct modeling of the SEY properties of the surface is crucial for having reliable simulations. SPARK3D multipactor module, allows for using custom SEY parameters or even import ASCII SEY definition files (see material definition).

Unfortunately, in the real world, there is a **high uncertainty** with the real values of the SEY:

- First of all, the SEY of a certain surface depends not only on the material itself but on the microscopic roughness, impurities, cleanness, and oxidization processes. This means that **there are no "universal" SEY curves** for the different materials. For example the SEY of the silver coating of a company may differ from the SEY of the silver coating of another company.
- In addition, there are more caveats. **The SEY properties of a material may change with time** in which is known as **Ageing** process. That means that a certain sample may present important deviations of the SEY measured at a particular time, and the SEY measured some time later. In [1] it has been reported the variations of the SEY during 6 and 18 months of many types of coatings coming from different companies. As a result, it has been observed that the Ageing can cause an important variation in the multipactor breakdown (2dB-7dB).

See below an example in Table 1, where the measured SEY properties of silver coatings coming from different companies are compared (extracted from [3], company names are confidential). A big difference can be observed. Values measured at different moments are also

presented, showing a noticeable variation.

Table 1: Comparison of Silver SEY for different companies and variation with time (Ageing).

	Initial			After 6 months			After 18 months		
	E1	SEYmax	Emax	E1	SEYmax	Emax	E1	SEYmax	Emax
Company1	20	2.8	380	20	3.1	298	20	3.1	268
Company2a	40	1,9	410	29	2,1	322	24	2,6	288
Company2b	44	2,0	484	39	2,3	376	39	2,2	376
Company3	43	1,7	210	34	2,1	366	34	2,1	385

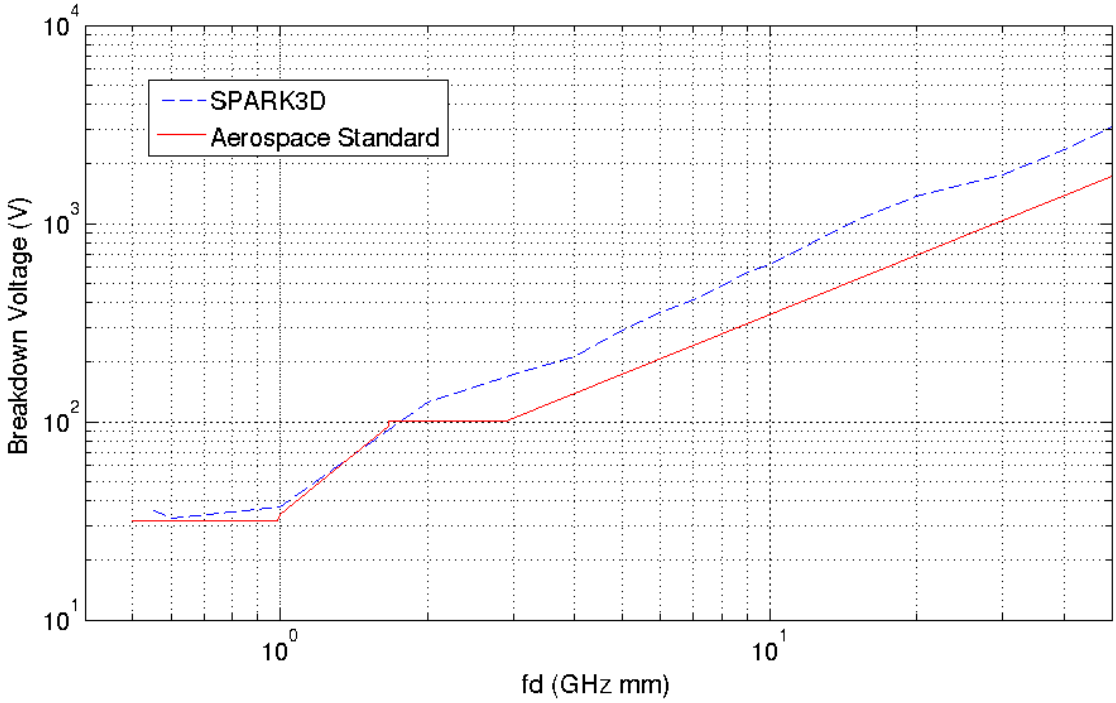
With all this in mind, the engineer must interpret the breakdown discharges given by the software with caution, expecting some margin in experimental measurements. Our recommendation is to do a **SEY sensitivity analysis**, simulating the same structure with **different SEY curves**, to see the impact on the breakdown power, since this impact will strongly depend on the particular component under analysis.

Standard SEY materials

SPARK3D includes typical **SEY parameters for most relevant materials**, extracted from European ECSS standard [2] and American Aerospace Corp. standard [3]. Both standards give worst-case multipactor breakdown charts which may be useful to easily estimate the breakdown levels for the parallel-plate case. For real structures, numerical simulation with SPARK3D provides more accurate results.

The ECSS standard figures correspond to different materials and come from the fitting of the multipactor breakdown results to a particular test campaign done in [4]. For that reason, numerical simulations with SPARK3D (with simple structures, close to parallel-plate geometry) and ECSS SEY parameters, provide results similar to those of the ECSS standard.

In turn, the SEY parameters provided by the Aerospace standard do not correspond to real measurements, but correspond rather to a single material which represents theoretically the worst-case (lowest breakdown levels). On the other hand, the Aerospace Corp. standard is based on the classical multipactor theory for parallel plates without experimental data fitting. As a result, numerical simulations with SPARK3D (with simple structures, close to parallel-plate geometry) and the Aerospace Corp. SEY, provide more realistic (higher) breakdown levels. Fig. below shows the difference (around 3 dB).



References

[1] ESA-ESTEC TRP AO/1-4978/05/NL/GLC "SEY Database", Final Report, December 2011.

[2] "Space Engineering: Multipacting Design and Test", volume ECSS-20-01A, edited by ESA-ESTEC. ESA Publication Division, The Netherlands, May 2003.

[3] AEROSPACE REPORT NO. TOR-2014-02198, "Standard/Handbook for Radio Frequency (RF) Breakdown Prevention in Spacecraft Components"

[4] A. Woode and J.Petit. "Diagnostic investigations into the multipactor effect, susceptibility zone measurements and parameters affecting a discharge". Technical report, ESTEC working paper No. 1556, Noordwijk, Nov. 1989.

2.3.3 Corona Analysis

The Corona Discharge analysis section contains the following topics:

Definition	What is exactly done when using this SPARK3D feature.
Limitations	What are the limitations you should be aware of.
Errors	The possible errors produced when performing the Corona Discharge analysis, and solutions or workarounds to them.
Using the Corona Discharge module	How to use this feature in SPARK3D from the User Interface or, in case you need, from the command prompt.
Output of a Corona Discharge simulation	Description of the output of a Corona Discharge simulation.
Hints	Non-trivial properties of the use of the Corona Discharge module.

Definition

The Corona Discharge analysis computes the corona breakdown power threshold for a range of pressures of one or more regions defined by the user from the current device. The breakdown power calculated is the input power at the entrance of the device.

Limitations

It is not feasible to exactly determine inside the chosen region where Corona onset takes place.

With respect to precision, on the one hand the results given by the software have limited accuracy if the pressure (in mBar) significantly differs from the pressure at which the breakdown threshold is minimum (critical pressure). The value of such a minimum (in mBar) is normally very close to the frequency value (in GHz).

On the other hand, depending on the particular geometry, a too dense mesh could be necessary in order to achieve a convergent result in the threshold breakdown power. This situation could then lead to a memory overflow, which ultimately would fix the limit in the results' accuracy.

Errors

Errors can occur when importing the EM fields. Considering a mesh too dense there could be a memory overflow in the numerical simulation. When this occur, a coarser mesh for the EM fields should be taken, even though this could lead to an accuracy loss.

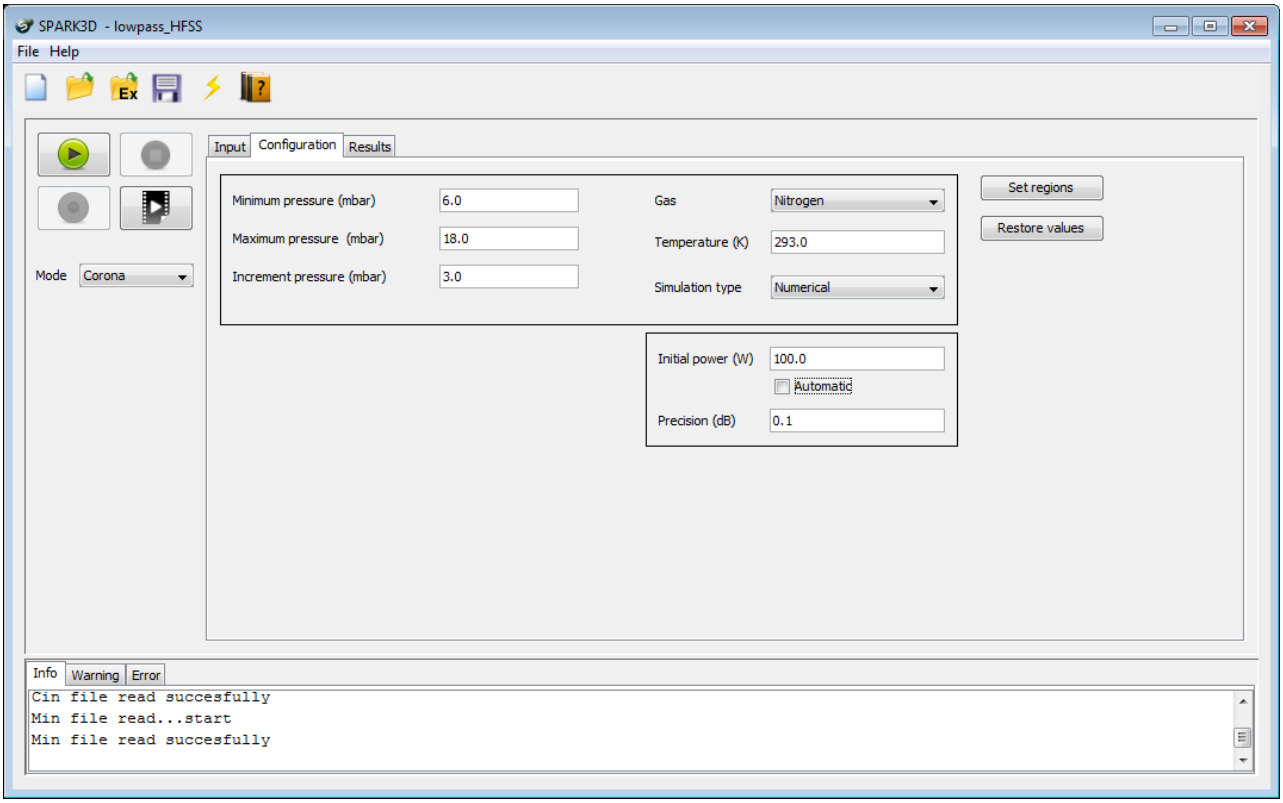
Alternatively, the user can focus the simulation on regions of analysis specified inside the device (see Specifying Regions tutorial). In this way, the mesh used in the computation is still dense, so that the accuracy is maintained, but a memory overflow is avoided.

Usage

The corona discharge analysis of the device under simulation is controlled through the GUI, that allows easily setting the input parameters.

Configuration tab

The configuration window permits setting the rest of the simulation parameters as is shown in the next menu:

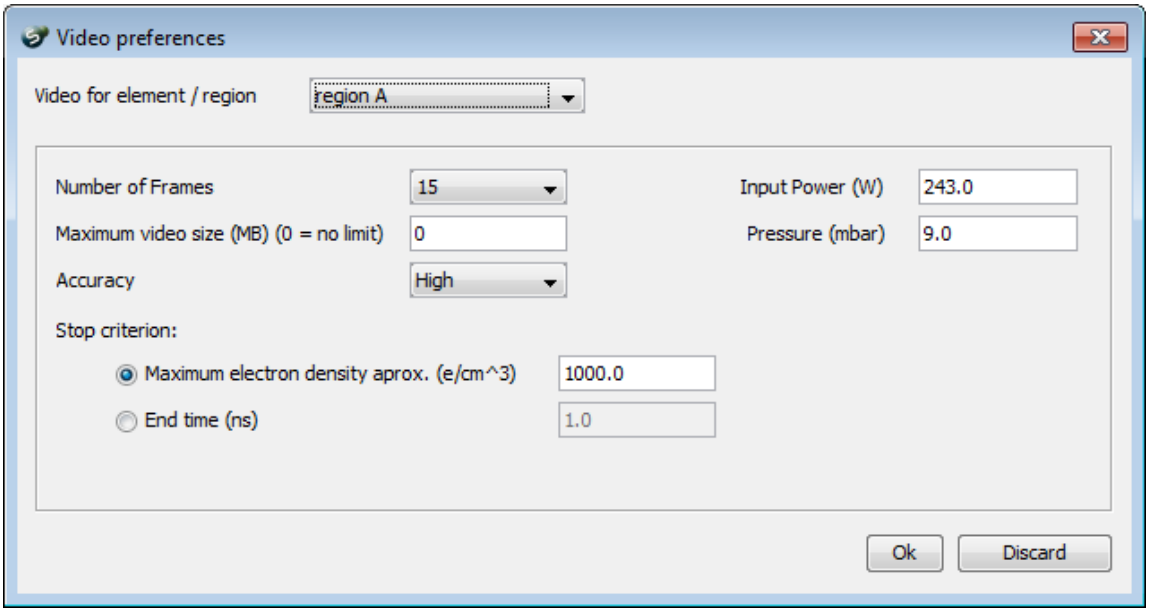


Minimum pressure (mBar)	Pressure at which the pressure sweep will start.
Maximum pressure (mBar)	Pressure at which the pressure sweep will finish.
Increment pressure (mBar)	Step in pressure for the pressure sweep.
Gas	Two gases can be considered in the simulation: nitrogen and air. Nitrogen will always provide a lower corona breakdown power level.
Temperature (K)	Ambient Temperature. The reference is taken as the room temperature of 293 K.
Simulation type	Three different simulation types can be considered: • Numerical, which corresponds to a numeric algorithm that uses an adapted FEM technique to solve the free electron density continuity equation. • Analytical rule, which is detailed in **high pressure analytical rule section**. • Numerical & analytical, which enables both simulation types.
Initial power (W)	Power from which the threshold breakdown power is looked for. It must be set for both "Numerical" and "Numerical & analytical" simulation types. Its value may be set by the user or it may be taken automatically (enabling the "Automatic" check box) from the **high pressure analytical approach**.
Precision (dB)	This parameter sets the desired precision in power level for the corona breakdown onset.
Set regions	Through this option the user can define the specific regions of the structure where the analysis will be carried out. For a detailed description of its usage see specifying regions section of the tutorial.

Video Record

If corona mode is selected videos can be saved by pressing the record button  and opened at any time with the play button 

The record video dialog is the following



Video for element / region	Here, the region in which the video is going to be recorded is selected
Number of Frames / period	Specifies the frame rate of the recording. The higher the smoother the animation, but bigger video sizes will be generated.
Maximum video size (MB)	If different from zero, the video will be cropped when the video file (approximately) exceeds the set size.
Accuracy	Sets the level of accuracy that will be used in the electron density computation. The higher is this level, the more accurate, time and memory consuming is the computation.
Stop criterion	Sets the criterion used in the last frame of the video to stop the computation of the electron density: • If "Maximum electron density approx." is selected, the maximum value of the computed electron density in the last frame of the video will be approximately the value fixed by the user. • If "End time" is chosen, the electron density time evolution will be calculated till the time specified by the user.
Input Power (W)	Sets the input power for this specific video recording.
Pressure (mBar)	Sets the pressure value for this specific video recording.

Other parameters are taken from current configuration, such as gas type and temperature.

When the Ok button is pressed, an output file must be selected (*.v3d extension) and the simulation starts. Results from previous simulations will not be deleted. The video will be stored outside of the project and it is independent of it (can be opened from other projects, for example).

Once the simulation is finished, the user can select to immediately open the video, or to open it at any time with the play video button  . The videos are opened with the 3D CAD viewer software Paraview, which allows for 3D rotations, perspective customization and zoom on the saved animations. It also allows for exporting the animation to popular video formats, such as avi format for instance. See the Video Tutorial for further information.

High pressure analytical rule

At high pressures, where diffusion is negligible, it is also possible to include the breakdown power threshold corresponding to a **high pressure analytical rule** by enabling its corresponding check-box. The obtained results are based on the well-known relation for ionization breakdown at sea level (*W. Woo and J. DeGroot, Microwave absorption and plasma heating due to microwave breakdown in the atmosphere", IEEE Physical Fluids, vol. 27, no. 2, pp. 475-487, 1984*), which in the case of air corresponds to:

$$E_{breakdown} = 30.17\ (pressure^2 + 2 \cdot frequency^2)^{0.5}\ (V/cm)$$

This rule is conservative at all pressure ranges. At high pressures, it gives an estimation for the breakdown power threshold whereas at low pressures - where diffusion losses are much more important- this rule only results in a very conservative breakdown onset.

It is important to point out that the results are extremely dependent on the maximum value of the Electric field magnitude, Emax. This means that if this value changes, the high pressure analytical results will also change. Such a modification usually occurs in problems where the maximum electric field is concentrated on small localized regions, like in devices where metal

corners are present. There are several reasons for such a variation:

- Change in the mesh used to compute the EM field. If the mesh is not dense enough, the maximum value found for Emax may not be the absolute maximum and small changes in the mesh may lead to different results.
- The use of non-convergent results for EM field calculation. If the EM field computation has not converged, a change in the simulation parameters may lead to different values of Emax and consequently to different results.

Running Corona Discharge Analysis from command prompt

It is also possible to execute Corona Discharge Analysis from command prompt. The executable name is *spark3d.exe* on Windows platform and *spark3d* on Unix-like platforms, and is located in the directory where SPARK3D is installed.

Before running the simulation from command prompt, a text file must be created to keep all configuration parameters for Corona simulation. The extension of the file must be *.cin* and its format looks as follows:

```
begin "corona"
  begin "regions"
    num_regions             2
    not_analyze_all_when_simulate       1
    begin "region"
      region_type           "box"
      region_name           "circuit"
      region_number         1
      analyze               0
      center_x              0.0
      center_y              0.0
      center_z              0.0
      size_x                0.0
      size_y                0.0
      size_z                0.0
    end "region"
    begin "region"
      region_type           "box"
      region_name           "region A"
      region_number         2
      analyze               1
      center_x              0.0
      center_y              0.0
      center_z              0.023
      size_x                0.03
      size_y                0.03
      size_z                0.01
    end "region"
  end "regions"
  configuration_name        "config1"
  gas           "air"
  minimum_pressure          1.0
  maximum_pressure          6.0
  increment_pressure        1.0
  temperature               293.0
  precision     0.1
  initial_power_type        "automatic"
  initial_power             1.0
  iteration_type            "brent"
  simulation_type           "numeric"
  videoNumFrames            15
  videoAccuracy             "high"
  videoMaxSize              0
  videoPressure             1.0
  videoInputPower           100.0
  videoNmax     1000.0
  videoTmax     1.0
  videoStopCriterion        "Nmax"
  videoRegionsLabel         "1"
  videoOutputPath           "Select output file"
end "corona"
```

All these parameters are described in the **Usage** of Corona Discharge from User Interface and in the Specifying Regions section.

Executing the command *spark3d -h* (prefixed by SPARK3D installation directory) will show all command-line arguments and options supported, including how to specify input and output files.

A typical invocation of Corona Discharge Analysis looks as follows:

```
<full-path-to-spark3d/spark3d.exe> --mode=corona --project_path=<working-directory> --tmp_path=<relative-path-to-temporal-
directory> --config_file=<relative-path-to-config-directory/mycircuit.cin> --output_path=<relative-path-to-output-directory> --
data_file=<relative-path-to-mesh-directory/EM_fields_file> --file_type=<fest,cst,hfss>
```

In the case of an HFSS file, it is also mandatory to include an extra argument:

```
--HFSS_units=<m,mm,inch>
```

See the example below:

```
C:/Program Files (x86)/SPARK3D1.5.1FULL/dist/spark3d --config_file="paramsFolder/conf1.cin" --output_path="outputFolder" --
mode=corona --project_path="Libraries/Documents/SPARK3D_Example/Dielectric_resonator" --HFSS_units="m" --tmp_path="tempFolder" --
data_file="meshFolder/dielectric_resonator.dsp" --file_type=hfss
```

The **full** path to SPARK3D executable is required since SPARK3D will search for the license file in the same directory as the full path specified.

The project_path indicates the folder where the other directories are located. In the example above, *meshFolder*, *paramsFolder*, *outputFolder* and *tempFolder* are located inside Dielectric_resonator directory and relative paths are given with respect to it.

It is also necessary to indicate a directory where temporal files will be stored (`tmp_path`) and a directory where the results will be kept (`output_path`).

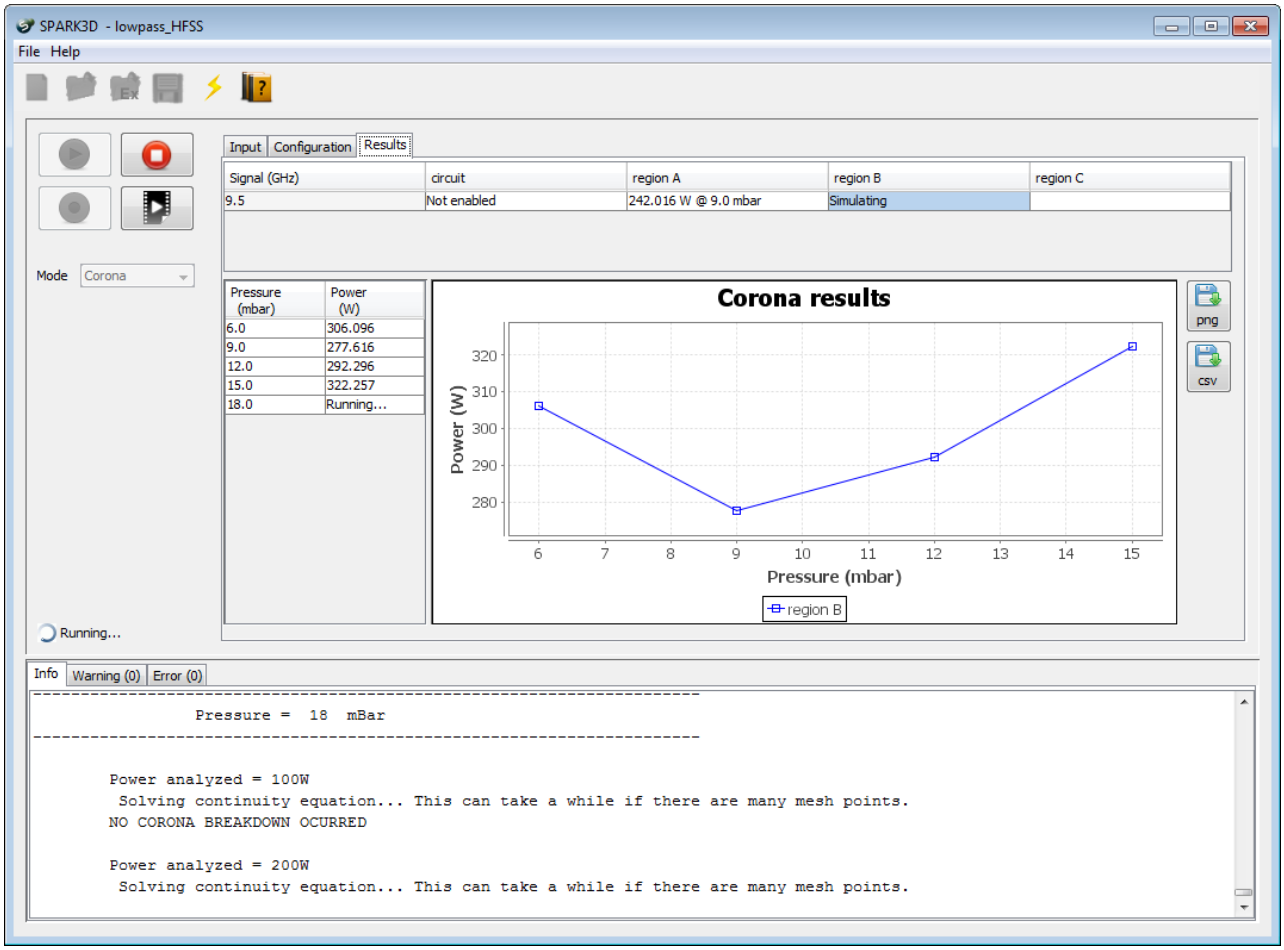
The argument `data_file` corresponds to the file where the EM fields have been exported, which in the example is stored in *meshFolder*. The argument `file_type` refers to the type of EM software used to export the EM fields and `HFSS_units` indicates the units used in the HFSS project to define the dimensions.

The possibility to launch SPARK3D from command line allows using it in combination with external programs. This way, scripts to launch several executables sequentially can be generated or it can be combined with a synthesis or optimization tool.

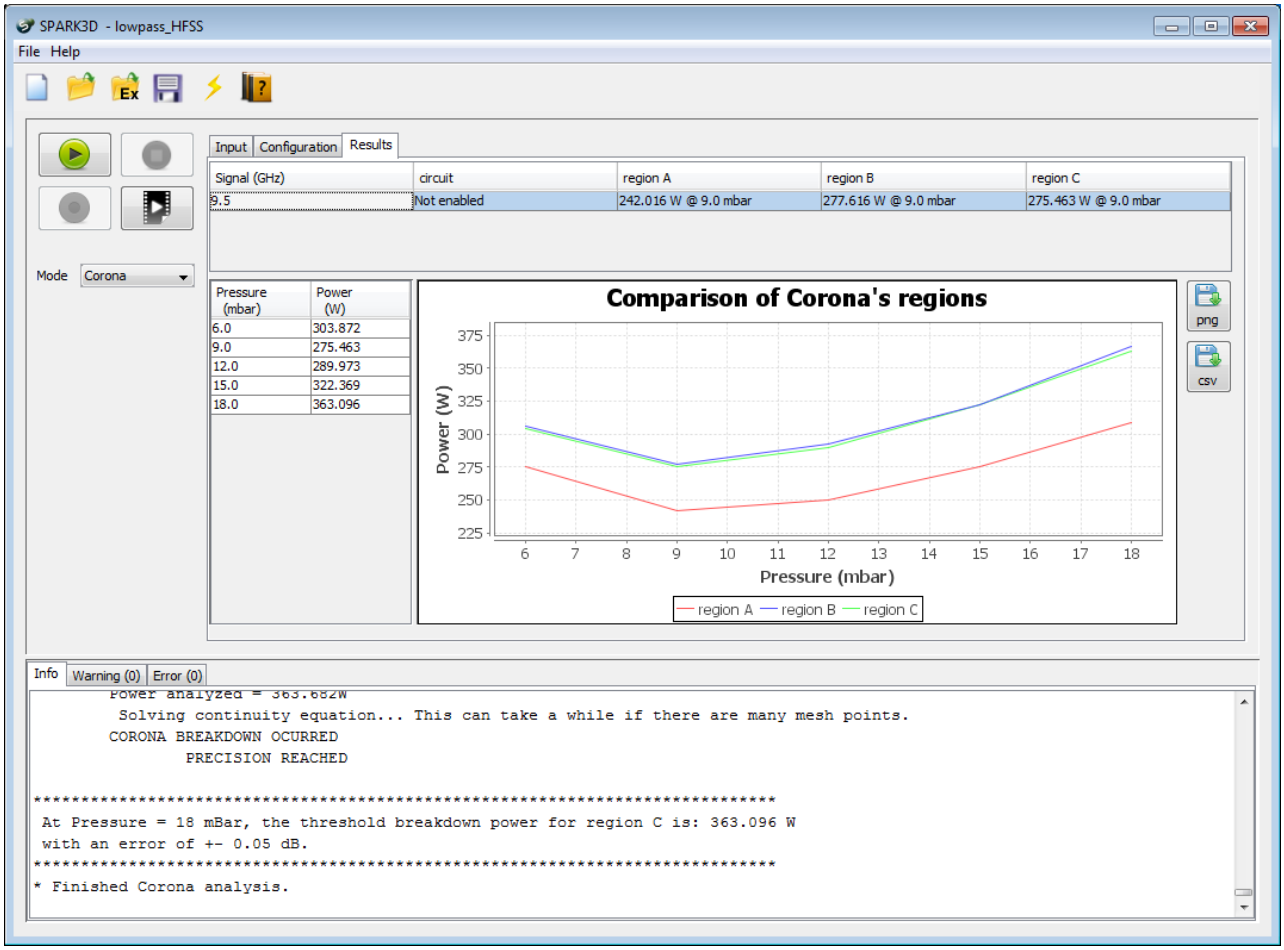
Output

Corona module provides the threshold input breakdown power of the selected regions of the structure. The simulation process can be visualized in the info window of the GUI where a sweep in input power is shown as the simulation runs, indicating how the simulator tries to approach to the corona breakdown threshold level.

The results of the analysis are given both in graphic and tabular form to make their interpretation easier. There are two tables and one graph, as can be seen in the following figure:



- In the left-side table is represented the threshold breakdown power for each pressure point corresponding to a certain region, which is selected by left-clicking on its corresponding cell in the upper table. If the high pressure analytical rule has been also selected for evaluation, the table will have three columns instead of two, where the last one corresponds to the empirical rule.
- The data of the left-side table correspond to the Paschen curve, which is represented in the graph. If the high pressure analytical rule is enabled, there will be two curves, one corresponding to the numerical analysis and the other one to the analytical rule.
- In the table located in the top of the results window the minimum breakdown power in the whole pressure sweep is represented for each region analyzed. Besides, through this table the user can handle the results shown both in the left-side table and the graph:
  - By left-clicking on a cell corresponding to a particular region both the graph and the left-side table update their values to the current element.
  - By left-clicking on the cell corresponding to the signal value, the whole row is selected and the graph shows together the Paschen curves of all the regions analyzed. With this information it is easy to recognize which is the most critical region for Corona discharge and the minimum breakdown power supported by the device.



The data represented in the graph can be saved into a .png file through the button 

or into a .csv file through the button 

.

Hints

- The minimum of corona discharge breakdown occurs at pressure levels (in mBar) close to the frequency value (in GHz). It is therefore recommended to include such a value in the pressure interval to be given.
- It is necessary to carry out a convergence study of the threshold breakdown power as a function of the mesh used in the description of the EM fields. It should be pointed out that for certain structures a too dense mesh, that would lead to a memory overflow, should be required in order to achieve convergent results. In theses cases it is highly advisable to use regions of study.
- Whenever user-defined regions are considered, a convergence study with the size of the region should be carried out. Once you have simulated Corona effect with a certain region, you should enlarge the region till the results remain unaltered or slightly change.

3 Index