

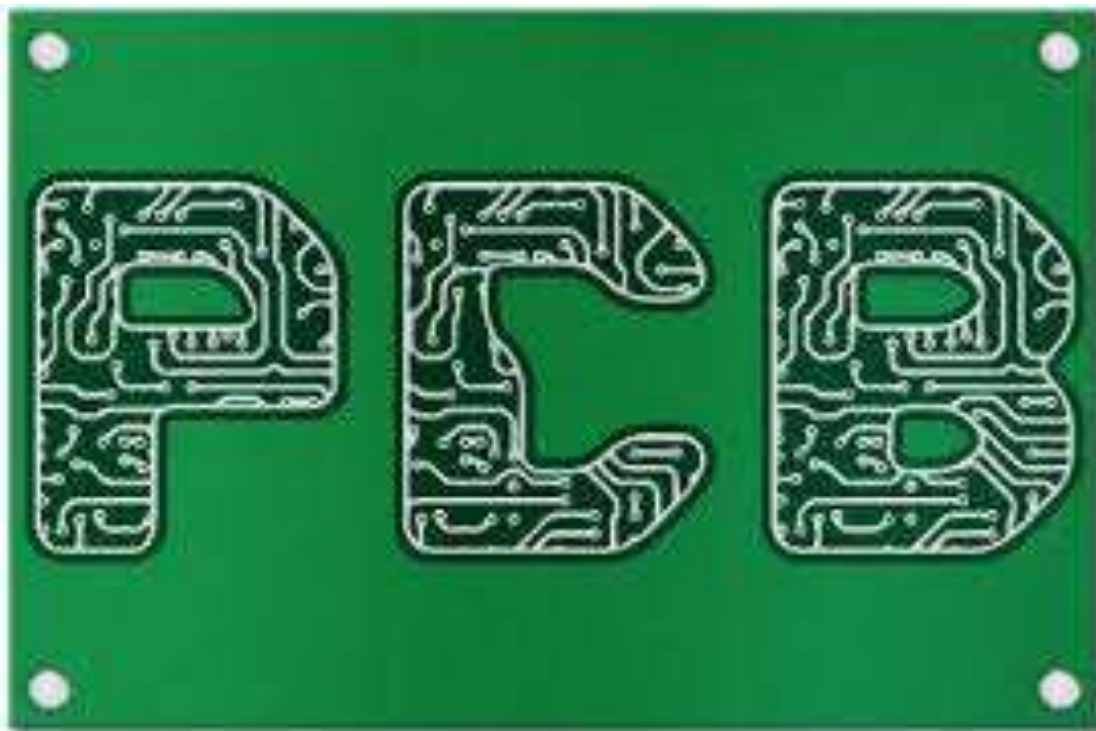
CAD package for electromagnetic and thermal analysis using finite elements and PEEC method

Altair[®] Flux[®]



ECAD import tutorial: Gerber format

PEEC technical example



Proprietary Information of Altair Engineering

Altair® Flux® is a registered trademark.

Copyright © 1983 – 2022 Altair Engineering, Inc.

These best practices were edited on 8 November 2022

Altair

15 Chemin de Malacher - Inovallée
38246 Meylan Cedex
FRANCE

Phone: +33 (0)4 76 90 50 45

Fax: +33 (0)4 56 38 08 30

Web: <http://www.altair.com>

Foreword

*(Please read before starting this tutorial)

Description of the document

A very simple application is illustrated in this technical paper to explain how ECAD imports are managed within Flux PEEC; this document goes through all steps of the process allowing the user to import PCB layers and vias described in Gerber format.

Moreover, description of actions required to define physical properties of imported objects is also presented.

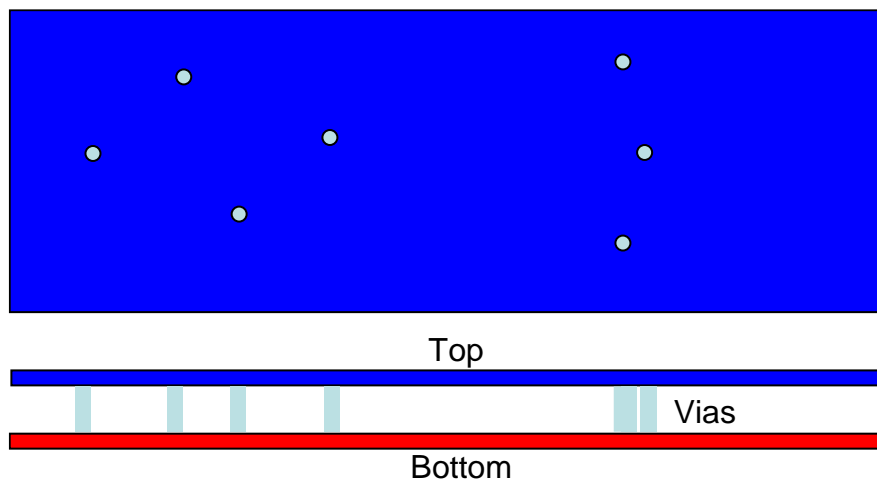
Table of contents

1. A basic application to understand the procedure.....	1
1.1. Choice and definition of the suitable application.....	2
1.1.1. Start and define the suitable application.....	2
1.1.2. Define the solving scenario.....	2
2. Import of Gerber files describing TOP and BOTTOM layers.....	3
2.1. Import of TOP layer.....	3
2.2. Import of BOTTOM layer.....	9
3. Geometry and physics.....	13
3.1. Geometry of TOP and BOTTOM layers.....	13
3.2. Physics of TOP and BOTTOM layers.....	14
3.3. Import of vias.....	15

1. A basic application to understand the procedure

Studied device The aim is to import Gerber files describing a basic two layers PCB connected by perpendicular vias and build the entire project in Flux PEEC software using dedicated tools.

Geometry to be imported Two rectangular planes situated on TOP and BOTTOM layers respectively are connected with seven perpendicular vias. The image below shows a top and a side view of the considered geometry.



1.1. Choice and definition of the suitable application

Studied device This section presents the preliminary steps necessary for defining the Flux PEEC application to be used and the computation scenario.

1.1.1. Start and define the suitable application

Goal First, the Flux PEEC application is defined: goal of this example is the computation of the equivalent impedance of the structure described above. This simulation will consider the resistive, the inductive and the capacitive behaviour of the two-layer PCB.

Definition of the application The chosen application is therefore **Conductor impedances with capacitive effects** and the default material for conductors is **copper** whose resistivity is 17.2 nΩ·m. Properties of the application are reported in the table below.

Type of application	Default material for new conductors
Conductor impedances with capacitive effects	copper

1.1.2. Define the solving scenario

Goal The solving frequency values are set via the solving scenario (only one scenario per project is allowed).

Definition of the scenario For simplicity, a mono-value scenario will be solved in this technical example: the considered frequency will be 1 MHz. Properties of the solving process scenario are reported in the table below.

Name	Piloted variation parameter	Control type	Value
Scenario_1	FREQUENCY	Mono-value	1e6

2. Import of Gerber files describing TOP and BOTTOM layers

Introduction

Geometrical description of the PCB is provided by means of three Gerber files: one for each of the two layers and one for the vias. The procedure to be used within Flux PEEC requires to firstly import all layers and at the end the vias. In fact, such a procedure will automatically establish the connection between conductors of the layers and vias, thus making easier the definition of the global model.

This section describes the import of two Gerber files: *Gerber-example-top-7vias.ger* and *Gerber-example-bottom-7vias.ger*. These files are supplied to the user in the folder

```
...|flux|Flux|DocExamples|ExamplesPEEC|Tutorial_Basic|
ECADImportGerberFormat|ECADImportGerberFormat.zip|
ECADImportGerberFormat_PEEC_Case1
```

2.1. Import of TOP layer

Introduction

Thickness of both layers of this PCB is fixed at 35 µm, their centre-to-centre distance is 250 µm.

Gerber import functionalities are accessible within Flux PEEC by the menu:

 Project → ECAD import → Import of a Gerber layer

Action (1)

To import the TOP layer, fill in the dialog box with the following information:

Gerber file containing the layer
Gerber-example-top-7vias.ger
Name of layer
TOP
Coordinate system for the layer positioning
XYZ1
Lower Z-coordinate of the layer
0.250
Layer thickness
0.035
Color of the layer
Black

Continued on next page

Remark

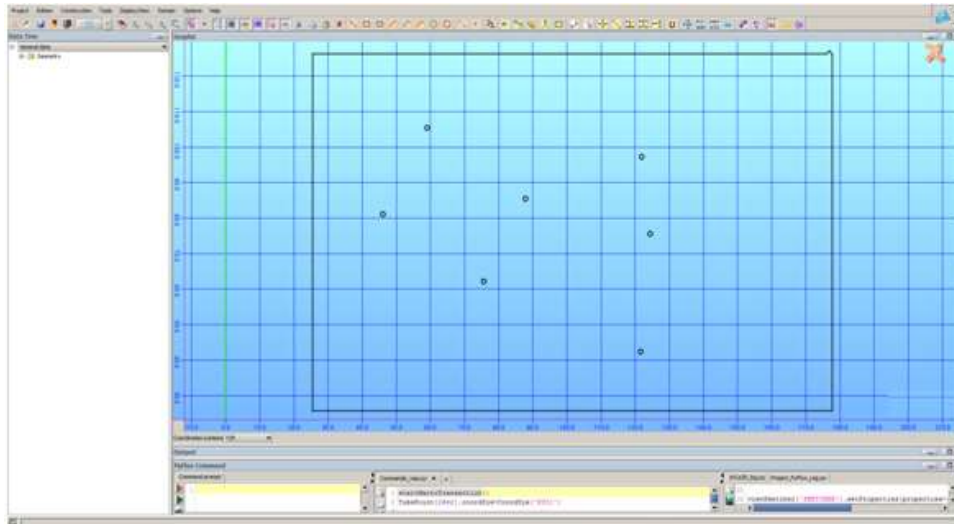
The coordinate system XYZ1 is a standard Cartesian system automatically created by Flux PEEC for each new project; its length and angle units are by default millimetre and degree, respectively. That's why in the previous dialog box it is necessary to convert Z-coordinate and thickness of the layer from micrometre to millimetre.

User who prefers to work in micrometres as length unit has just to modify the properties of XYZ1 coordinate system or, better, to create a new coordinate system.

Result

The layer named TOP is imported into a 2D sketcher context, with many points and lines describing shapes of the conductors.

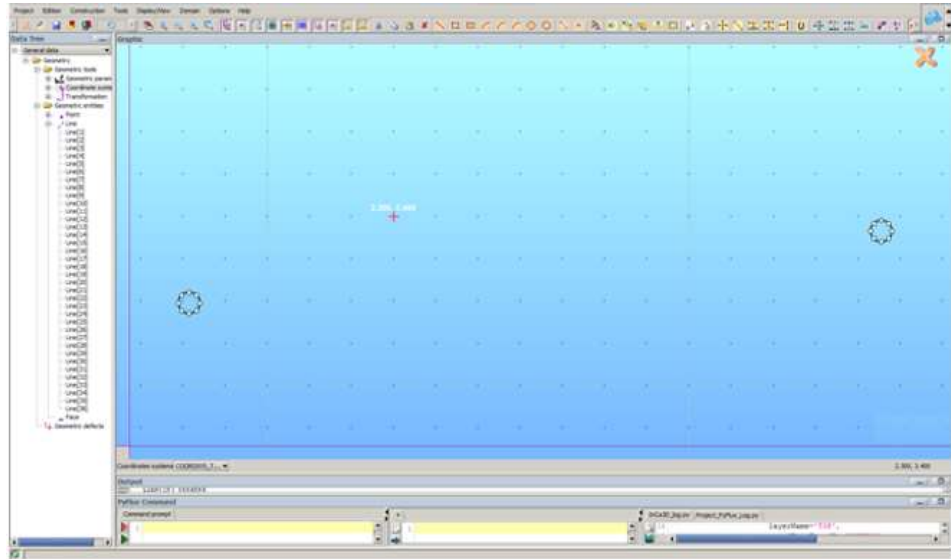
This 2D context is used to easily clean and/or modify details of 2D layers in order to prepare with dedicated tools the project to simulate: it is advised for example to remove parts that require extensive computational resources without providing useful electromagnetic results.



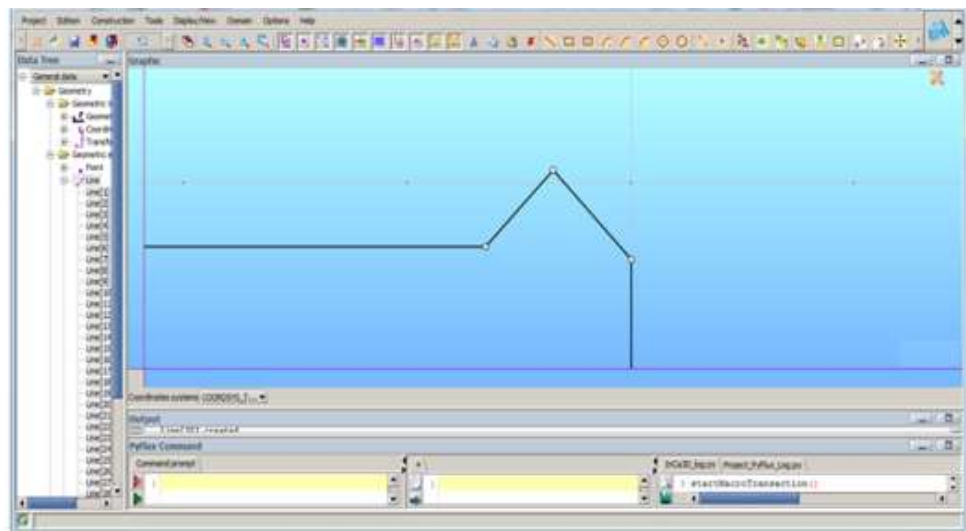
Continued on next page

Sketcher context

Some imported elements seem to be very small, but a zoom-in by using the wheel of the mouse makes it possible to note that several segments are linked together to create circles representing the places where vias are positioned.

**Defects**

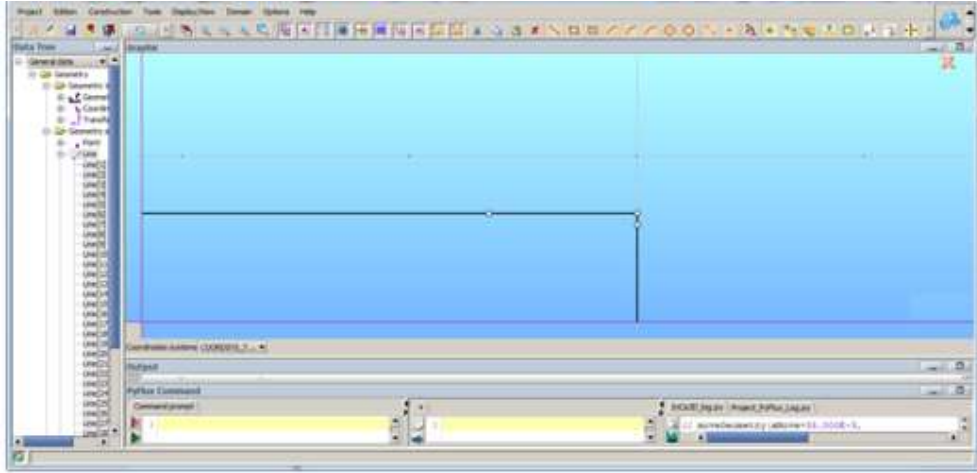
By zooming-in on the right top part (or on the bottom left part) of the imported TOP layer, two geometric defects appear: an additional protuberance exists at each corner of the rectangle.



Continued on next page

Action (2)

These defects can be easily corrected by the user by simply left-clicking on the central point of the protuberance and moving it down (or up, depending on the considered corner) towards the other point on the side to get the following geometric configuration:

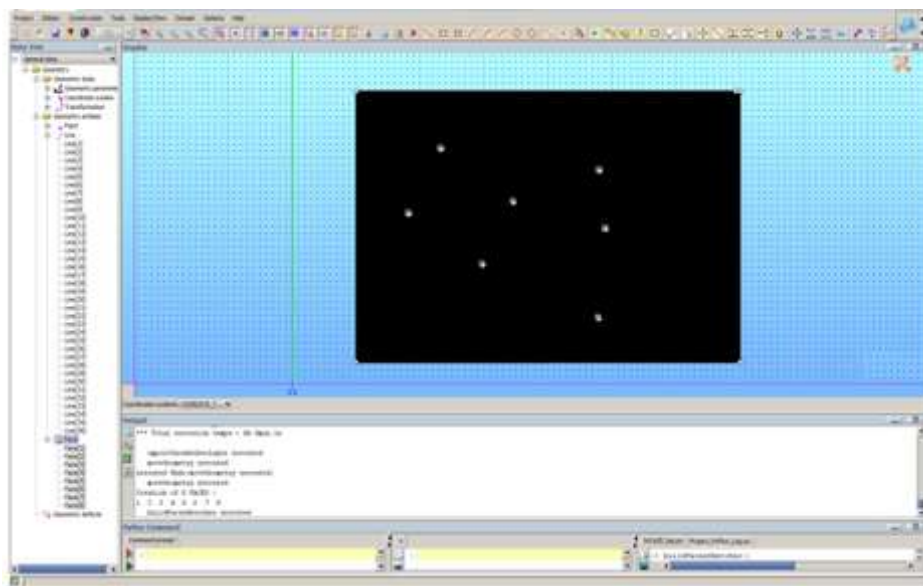


Action (3)

Once all corrections are carried out, creation of faces of the layer is required to carry on with the import operations. This functionality is accessible by the menu:



Eight faces are created: seven represent the area of vias and the latter the rest of the rectangle. Their color is black as previously indicated in the dialog box for the layer import.



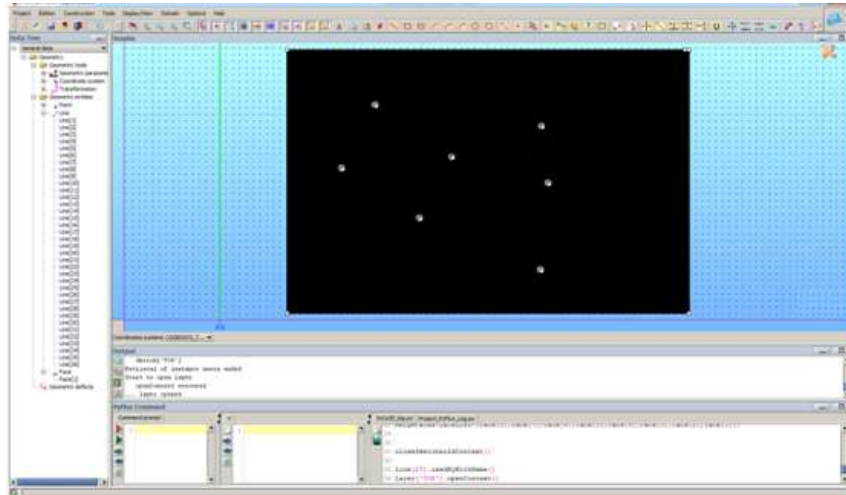
Continued on next page

Action (4) From a physical point of view, the PCB (ground or power) plane is constituted by the whole rectangle. The definition of such a conductor could be done by listing all these eight faces but it will be more efficient if they are merged together to create a global one.

In fact, the seven faces describing the vias can be disregarded since their position is also stored in their Gerber file to be imported at the end of this process. Moreover, the fact that these via areas touch the main face means that they are electrically connected to the plane and they are not holes. Consequently, the geometry can be simplified by means of the menu:



All faces need to be selected, either by graphic selection or in the data tree by keeping pressed the Ctrl button of the keyboard. As a result, only one face remains (see data tree), whereas points and lines identifying vias still appear in the graphical view.



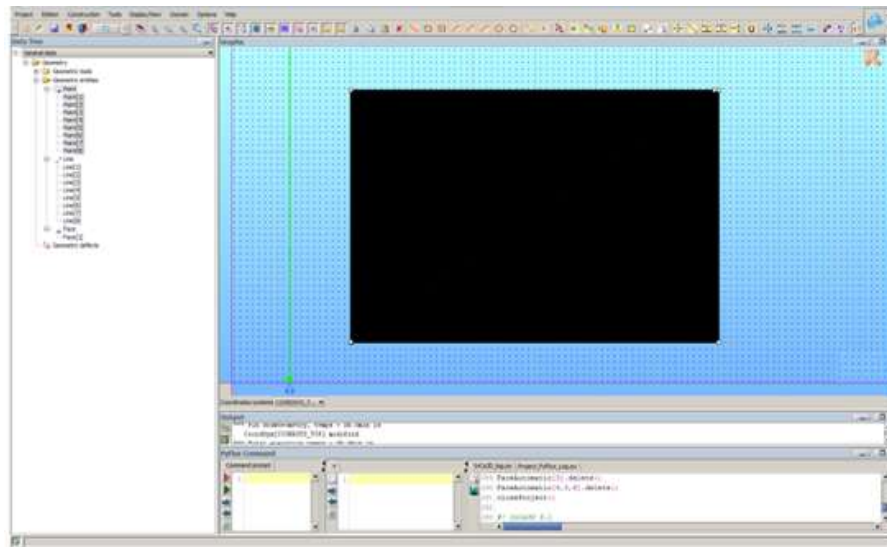
Remark These points and lines are also listed in the data tree, but they are now declared as “NO EXIST”, so that they do not contribute to the definition of the model. But, on the other side, in case the user needs to revise the geometry he can delete faces, modify the property of involved lines from “NO EXIST” to “STANDARD” and then rebuild faces in order to create the original geometry he had before the merging.

Action (5) In this technical example, no revisions of the geometry are planned, so that the suppression of useless points and lines is achieved. To this aim, select in the data tree points from [9] to [64] by means of the Shift button of the keyboard, then right-click and select the *Delete* command in the menu that appears.

Continued on next page

Result

As a result, all points and lines that described the position of vias are suppressed.



Action (6)

At corners where protuberances existed at the beginning of the process, some useless points still remain and some lines can also be merged since they are constituted by several segment having the same direction. To this aim, use the menu:

☞	Tools → Simplify all lines	
---	-----------------------------------	---

As a result, only four points and four lines still exist and they are enough for the definition of plane shape.

Action (7)

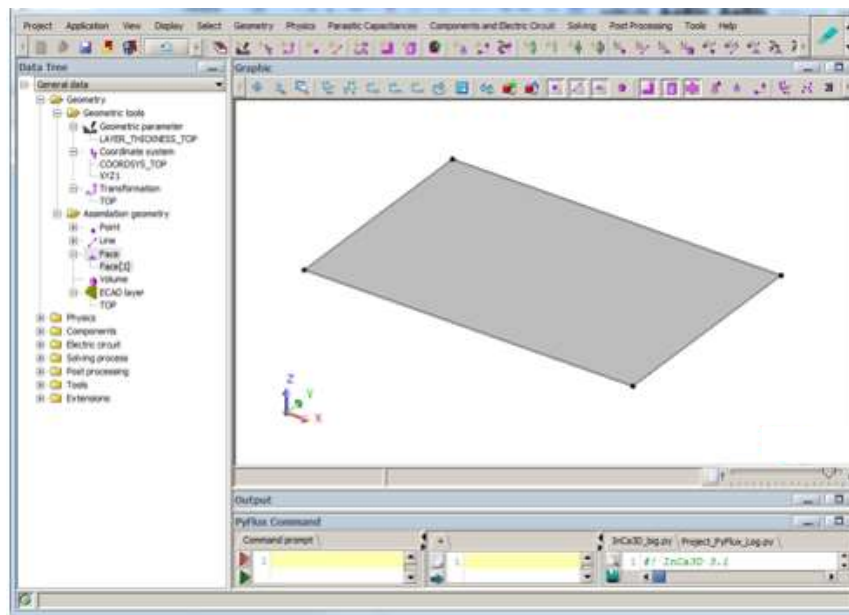
Operations in the 2D sketcher context for importing and simplifying the TOP layer of the PCB are concluded; coming back to the Flux PEEC standard working context is the next action that can be carried out by clicking on the orange cross at the right top corner of the window or by the menu:

☞	Project → Close Sketcher2DContext	
---	--	---

Continued on next page

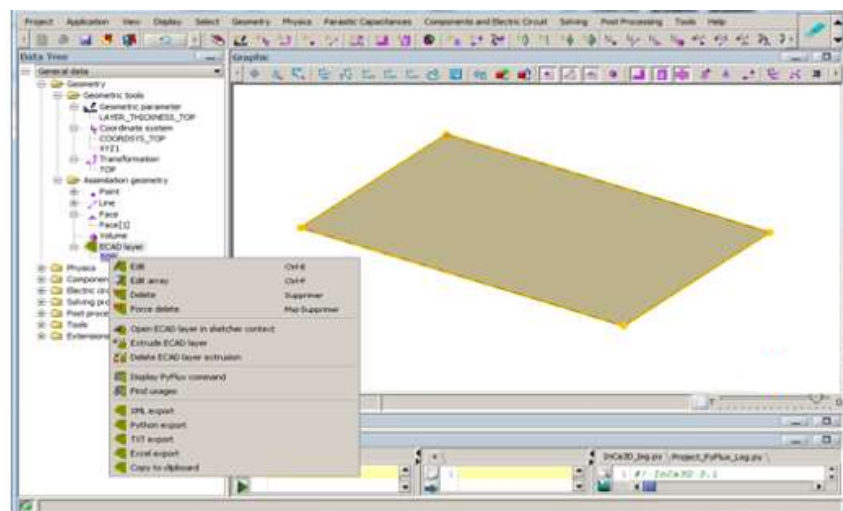
Result

The points, the lines and the faces (only one in this example) describing the lower surface of the TOP layer are automatically transmitted to the standard working context of Flux PEEC. Moreover, a geometric parameter `LAYER_THICKNESS_TOP`, a coordinate system `COORDSYS_TOP` and a transformation `TOP` are created in order that the user could reposition the imported layer in the 3D space and especially could easily generate later the volumes representing the PCB planes and tracks.



Remark

An ECAD-layer entity named TOP has also been created (see data tree in the image below): it is fundamental to generate later the volumes representing the PCB planes and tracks, but it also enables to go back to the ECAD working context and continue defeaturing operations on the layer if some details have been forgotten.



2.2. Import of BOTTOM layer

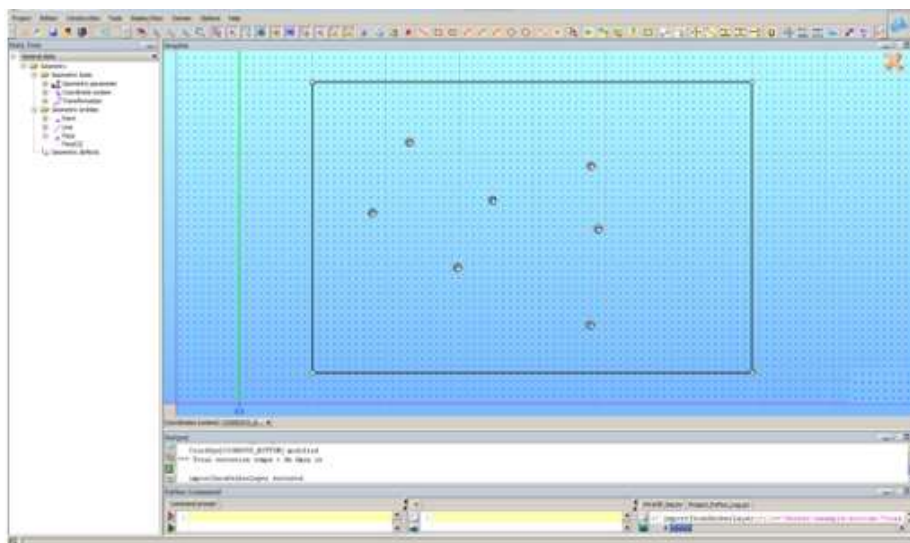
Action To import the file describing the BOTTOM layer, go to the menu:



and then fill in the dialog box with the following data:

Gerber file containing the layer	Gerber-example-bottom-7vias.ger
Name of layer	BOTTOM
Coordinate system for the layer positioning	XYZ1
Lower Z-coordinate of the layer	0
Layer thickness	0.035
Color of Layer	Red

Result The layer named BOTTOM is imported into the 2D sketcher context:



Defects As it was for the TOP layer, also the BOTTOM presents two geometric defects: a chamfer on the left top corner and a protuberance of the right bottom corner of the rectangle. To correct them, the user has to left-click on points and move them to the location allowing to have a perfect rectangular shape.

Continued on next page

Faces, lines and points

Once all corrections are carried out, the user can proceed with the creation of the faces by the menu;



Construction → Face → Build faces



This command generates eight new faces, whose color is red as it was previously indicated in the dialog box for the layer import.

Merging these faces is the next action to perform by the menu:



Tools → Merge faces



that compacts all faces of the BOTTOM layer in only one and modifies to “NO EXIST” properties of internal points and lines that identify the via positions.

To further simplify the imported geometry, in this example, deletion of such useless points and lines is executed by selecting in the data tree points from [12] to [67] and then by pressing the *Delete* button of the keyboard. Please note that in the data tree also the entities belonging to the TOP layer (previously imported) are listed.

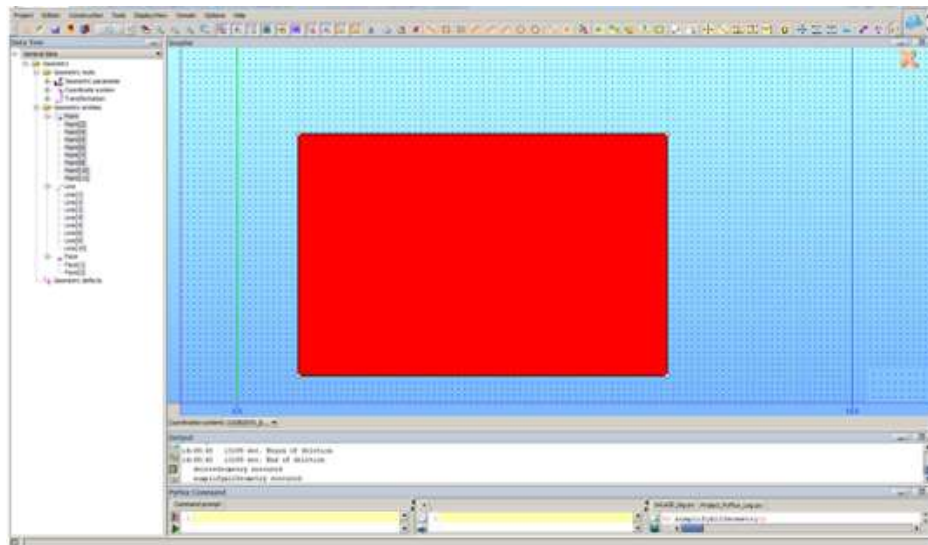
Final action to complete BOTTOM-layer import is the merging of lines at the rectangle corners where defeaturing operations had been carried out: to this aim, use the menu:



Tools → Simplify all lines



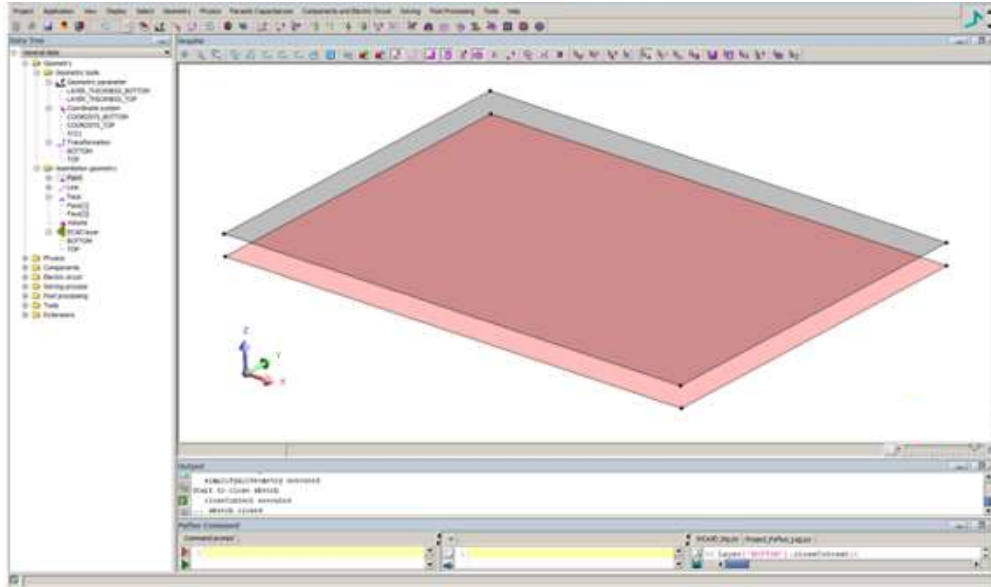
As a result, only four points, four lines and one face still exist for this BOTTOM layer and they are enough for the definition of the plane shape.



Continued on next page

Action

Operations on BOTTOM layer in the 2D sketcher context are concluded, so that user can quit this working environment by clicking on the orange cross at the right top corner of the window and come back to Flux PEEC standard view where the two planes (representing the TOP and the BOTTOM layers) are correctly placed. Moreover, all parameters, coordinate systems and transformations are automatically defined to easily continue the process.

**Advice**

At this stage of the process, it is highly recommended to save a copy of the project and keep it separated from the following actions. In fact, from now on, operations will be performed in the standard Flux PEEC context, but in case the geometry of the imported layers has to be revised, the modifications must be done in the 2D sketcher context.

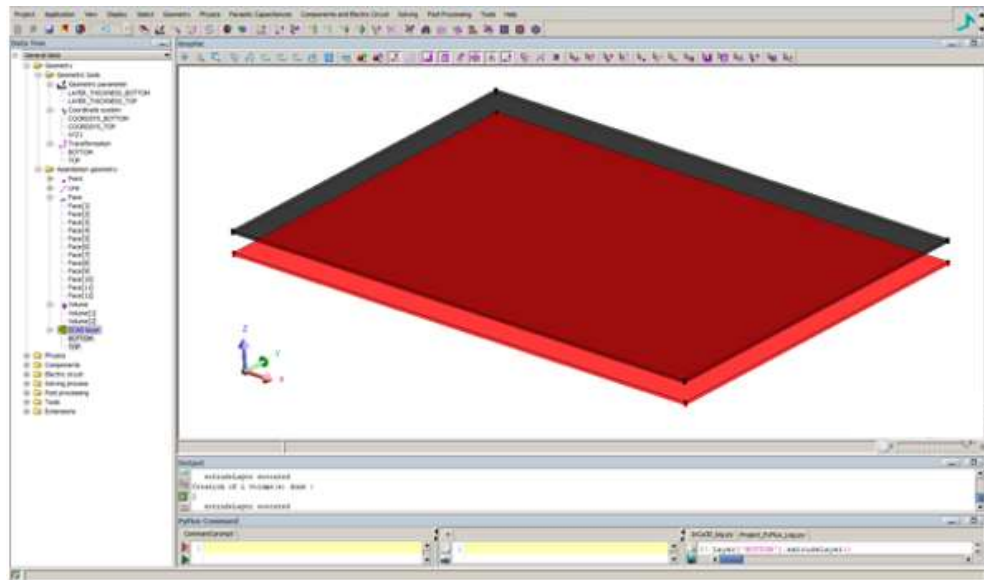
3. Geometry and physics

Introduction This section describes how to complete the geometric definition of the layers, their physical descriptions and the import of PCB vias from Gerber file.

3.1. Geometry of TOP and BOTTOM layers

Goal Aim of this part of the process is the creation of conductors describing PCB layers; this requires the creation of volumes from imported faces.

Action To build volumes for a layer, in the data tree choose the ECAD layer, right-click and select *Extrude ECAD layer*. Thanks to previously-created parameters and transformations, all faces of the considered layer are extruded to compose volumes whose thickness is equal to the value indicated in the Gerber-import dialog box. New faces and volumes are listed in the data tree: they have the same color of the layer. By repeating this extrusion for both layers, the geometry shown in the figure below is obtained.



3.2. Physics of TOP and BOTTOM layers

Goal Define physical properties of the layers imported by Gerber files.

Bidirectional conductors In this example, dimensions and shapes of volumes clearly indicate for both layers that they represent PCB power or ground planes. Their best model within Flux PEEC is thus the bidirectional-conductor type, since the current could freely flow in the two main directions on their surface.



Physics → Bidirectional conductor → New



Properties for TOP and BOTTOM conductors associated to TOP and BOTTOM layers respectively, are presented in the table below.

Bidirectional conductor defined by a list of volumes						
Name	Description		Meshing			Thickness
	List of volumes	Face of definition of the current flow plane	Surface			
			Type	Side 1	Side 2	
TOP	1	7	Uniform - Piecewise	10	7	None, i.e. one element
BOTTOM	2	12	Uniform - Piecewise	10	7	None, i.e. one element

Result Two bidirectional conductors are created: their material is equal to the default one, set up at the beginning of the project, i.e. the copper. Once all interesting volumes belonging to imported layers are processed, the import of vias can be executed.

3.3. Import of vias

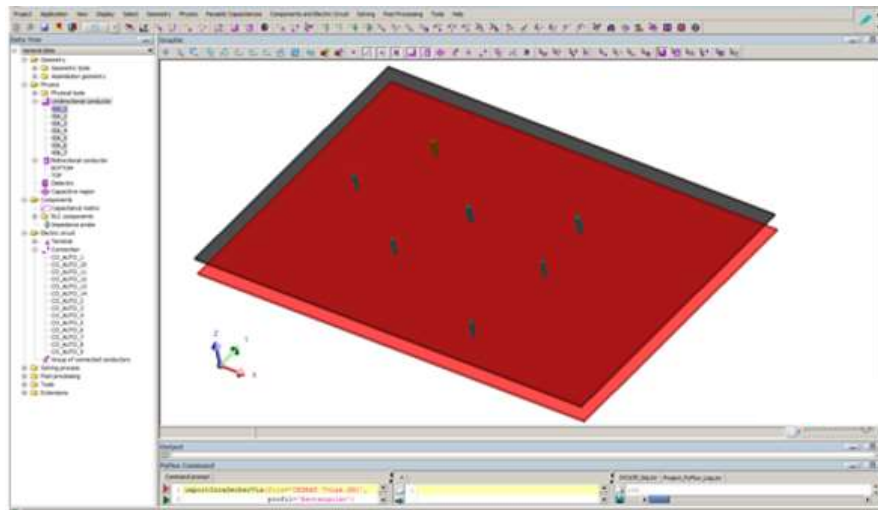
Goal The last step of the PCB import procedure deals with vias stored in a Gerber file, which is supplied to the user in the folder
`...|flux|Flux|DocExamples|ExamplesPEEC|Tutorial_Basic|ECADImportGerberFormat|ECADImportGerberFormat.zip|ECADImportGerberFormat_PEEC_Case1`

Action This functionality is accessible within Flux PEEC by the menu:

☞ Project → ECAD import → Import of Gerber vias

For this example, fill in the dialog box with the following data:

Gerber file containing the drill drawing
Gerber-example-vias-7vias.ger
Profile for vias
circularVia



Remark (1) Flux PEEC represents by default all imported vias as full-section conductors, even if in reality they are hollow: this choice makes it possible to tremendously limit the number of meshing elements and thus computation time, without downgrading the accuracy of the simulations, since frequencies analysed in power electronics applications are not so high and the main function of vias is the connection of layers. For signal integrity models, this assumption could not be true. The user who prefers to use hollow sections for vias can easily modify properties of all involved conductors by himself after import operations.

At the same manner, the user has the possibility to define vias as full rectangular conductors in order to save much more meshing elements without compromising result precision.

Continued on next page

Remark (2)

During import of vias, not only unidirectional conductors are automatically created, but a lot of useful electrical entities are also generated: in particular, on TOP and BOTTOM bidirectional conductors, new points are created at the locations where vias intersect their volumes. Moreover, at each of these points an electrical terminal is created (see the Terminal menu in the data tree); their name is TERM_BOTTOM_1, etc.
Since vias have also terminals (TERM_VIA_1_1, etc.) at each location where they intersect layers, all needed electrical connections (CO_AUTO_1, etc.) are set up between vias and bidirectional conductors. This automated procedure is of great help for the user since it accelerates the workflow and avoids potential manual errors.

Result

Gerber import of this simple example is now concluded. The user can carry on with construction of dielectric substrates if capacitance effects have to be considered in the simulation and/or with the connection of active (voltage and current source or impedance probe) and passive (RLC elements) components before solving.
