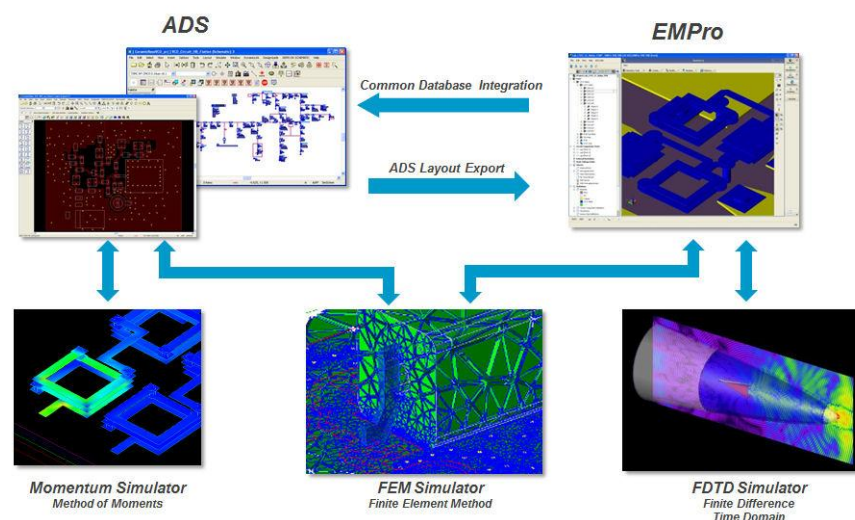


Laboratory no. 1

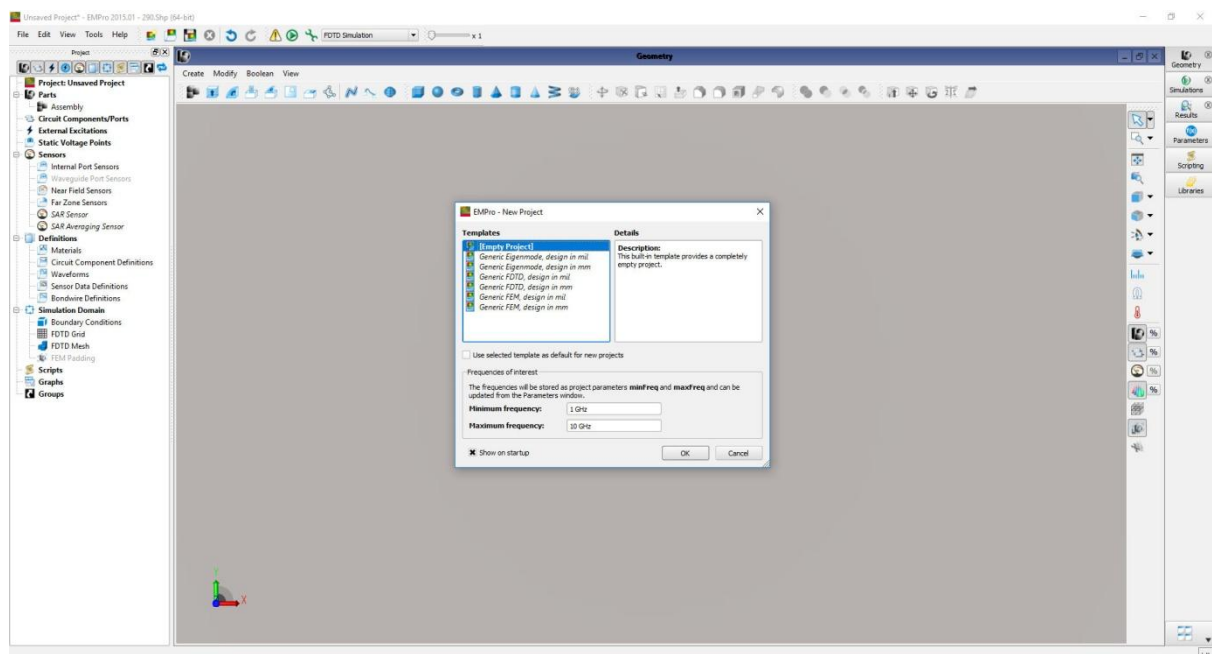
Electromagnetic simulation in EmPro

1. EmPro interface

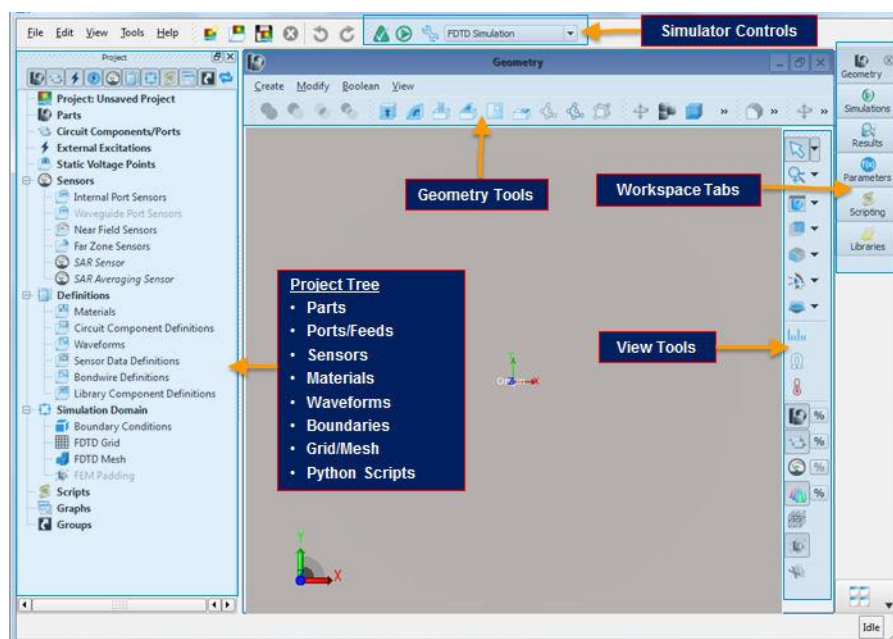
EmPro is a 3D electromagnetic simulation software created by Keysight (<http://www.keysight.com>). It is designed to provide to Advanced Design System (RF simulator created also by Keysight) with 3D electromagnetic simulation capabilities (ADS natively offers 2.5D simulation by MoM - method of moments).



To start the software click the Start button or use the shortcut on the desktop. The software starts in the structure drawing interface with a "create a new project" window.



When a new project is created, you can choose the particular simulator to be used (FDTD, FEM or Eigenmode solver), the measurement units to use in the project (particularly important are those for length, typically mm or mil) and also the frequency range in which the simulation will be performed. The frequency range can be changed later, but because a large number of (physical) elements depend on it, it's important to estimate it as accurately as possible from the beginning to avoid massive corrections later.



On the left side of the interface we find the **Project Tree**, a tree structure in which almost all the elements of interest in the current project can be found. The central window provides access to the structure for drawing/viewing (the **View Tools** button bar from the right provides quick commands to control in detail how different elements are plotted on the central window). The **Workspace Tabs** on the far right can display, on top of the **Geometry** window, the **Simulations** control, the view **Results**, the view/change **Parameters**, **Scripting** (Python), upload **Libraries** windows. At the top of the main interface, near the main menu, we find the **Simulator Control** area.

It should be noted that although FDTD and FEM are general-purpose simulators, thus both capable to simulate any type of structure (as a consequence any structure can be analyzed by either method or by both to compare the results), however changing the active simulator means more than only selecting it in the interface, as this operation has side effects that lead to more complex changes (changing the sources, changing the grid, etc.).

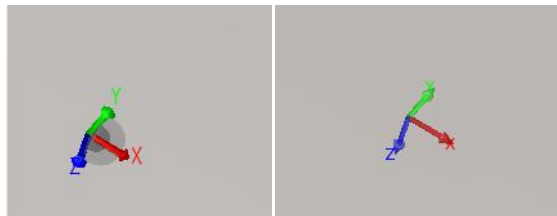
2. Design using EmPro

2.1. Materials

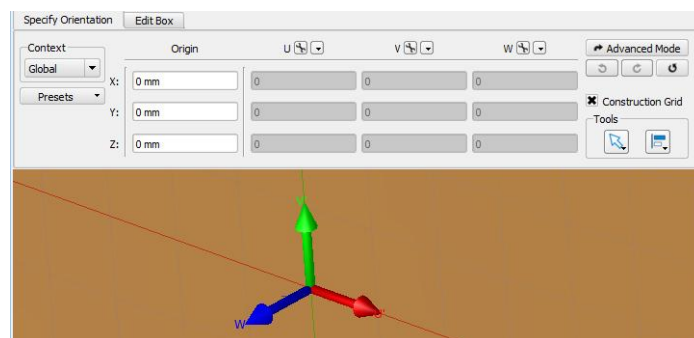
Materials must be added to the project by right-clicking on the **Definitions > Materials** section of the Project Tree. Normally, all the materials needed for this course can be found in the Standard Material Library, but custom materials can be defined (New Material Definition). This latter option may become necessary when the characteristics of a standard material have to be tuned to match the technology being used.

2.2. Coordinate systems

Every project will have a Global Coordinate System - GCS (x, y, z) displayed in the lower left corner of the Geometry workspace window, in report to which all items will be described. It's important therefore to examine the structure before starting to draw it and to choose the axes in such a way that they correspond to important directions in the structure: for example an axis of symmetry is a good candidate for one of the versors x, y, z or a direction to which most of the items in the structure have parallel sides. If GCS is properly chosen, most of the elements will be described by reference to this coordinate system. As a result, the x, y, z versors will be displayed twice, as the GCS in the lower left corner of the structure, and as the local coordinate system in the center of the item.



Some items might not be described easily in the GCS (usually this happens for a particular object inclined or rotated by rapport to all other items). In this case, a Local Coordinate System - LCS (u, v, w) for that item appears in a window similar to the one that follows, where we can **Specify Orientation** of the LCS indicating vector projections of the u, v, w versors in rapport to the GCS versors x, y, z.



2.3. Items

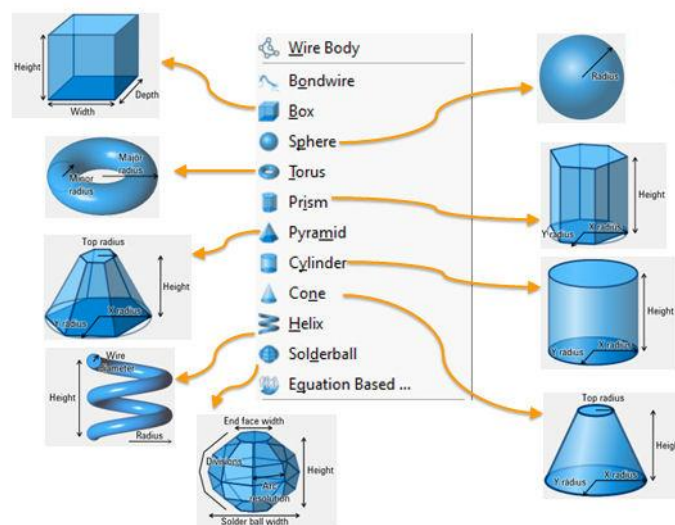
Items are inserted by right-clicking **Parts** in the Project Tree, then **Create New** An item must be given a suggestive name, useful to quickly find it in the drawn structure (if you have 100 boxes in you structure and all are named Box1, Box2, ..., Box100 you will have a hard time trying to find a particular item). In most cases, an item is created by drawing an object (section 2.4 below), but in the case of more complex items in the structure, it may be the result of a Boolean operation (unification, extraction, etc.) on several primary geometric shapes. An item, even if it does not have a geometric regular shape after the operations it has undergone, will have only one characteristic material. For example, a simple model of the BNC connector is made of one (metallic) material.

Elements can be (but is not necessarily to) grouped by creating assemblies: **Parts > Create New > Assembly**. An assembly must be given a name (recommended to be suggestive too) and allows the unitary manipulation of several items at the same time. The assembly may be composed of elements with different materials but physically manipulated together. For example, a coaxial cable can be defined as an assembly (inner conductor, dielectric, outer conductor).

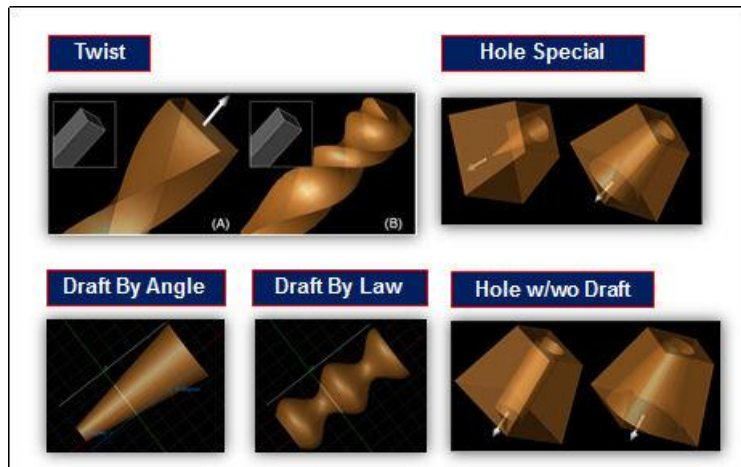
2.4. Drawing the structure

Because we cannot draw a 3D object using devices that allow only 2D control (mouse) and 2D plot (display), drawing the structure must begin by the decomposition of any object into simpler constituent items that can be described geometrically by equations.

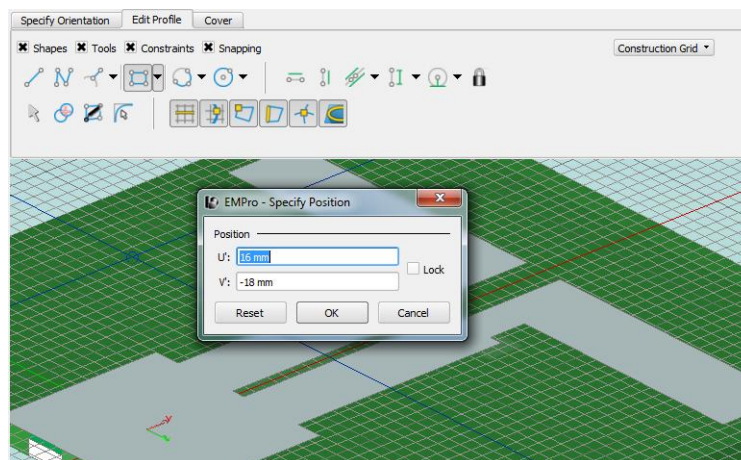
A first method consists in using and possibly overlapping multiple predefined 3D blocks **Parts > Create New > Box/Cylinder/ ...** and introducing their geometric definition.



A second method involves drawing a 2D surface (**Parts > Create New > Sheet Body**) and moving or rotating this two-dimensional figure to create a 3D object, **Parts > Create New > Extrude/Revolve**.



2D drawing calls for the introduction of common 2D geometric shapes available in the interface: rectangle, regular polygon, circle, ellipse. Their description is also achieved by equations (analytical geometry). Whenever a point is required, it can be indicated with the mouse on the screen or the coordinates of that point can be entered using the keyboard (if the point coordinates are known - press **Tab** instead of clicking on the screen with the mouse).

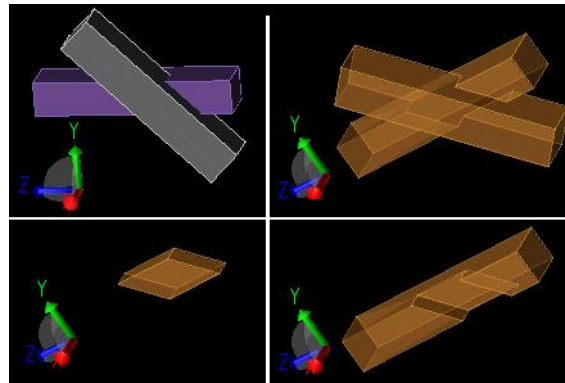


Some objects can be introduced as easily by both methods, for some objects instead the choice of the right method greatly reduce the difficulty of the describing the object.

2.5. Boolean operation with items

The Boolean operations: Subtract, Union, Intersect, Chop. Select the item(s) to which the operation is to be applied in the Project Tree (**Parts > ...**). Some operations can be applied to a single object (together with Extrude/Revolve) but most require two or more objects. After selecting the items, right-click **Parts** in Project Tree, **Boolean > ...**

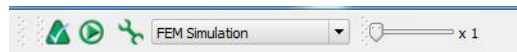
In the following figure, Union, Intersection, Subtraction operations are exemplified for two objects. For all operations, the original objects can remain in the structure (by selecting "Keep original") otherwise they disappear leaving only the result of the operation.



It is essential that no overlapping objects remain in the drawn structure (for example, an dielectric object and a metallic object cannot coexist in the same point in space). As such, Boolean operations between objects are either optional (as a method of drawing complex objects that cannot be described with elementary 3D blocks) or mandatory to remove illegal overlaps between objects.

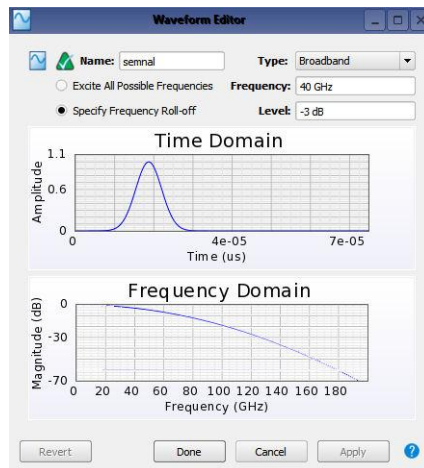
2.6. Choosing the solver

EmPro provides two general-purpose 3D electromagnetic simulators: FEM (Finite Element Method) and FDTD (Finite Difference Time Domain). Choosing a specific simulation method is done from the top side of the interface, next to the main menu.



The Finite Element Method will analyze the structure at a single frequency, and therefore obtaining a broadband response requires repetition of the simulation for a number of frequency points (fixed or determined by algorithms). The FEM method tends to require more computation time but instead each point is calculated independently and as a result is more accurate.

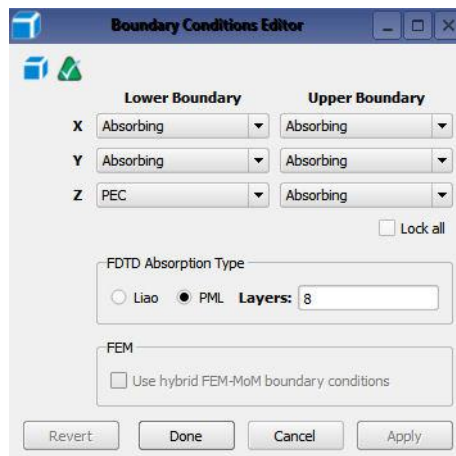
The Finite Difference Time Domain method works in time domain, applying to the structure a time-varying signal and obtaining the response to this excitation. By calculating the Fourier transforms for response and excitation, the system's transfer function can be determined, thus obtaining the broadband response in a single pass. The FDTD method tends to be faster, but it is dependent on the input signal (not all frequencies are excited to the same extent, generally the response is more accurately calculated at the center frequencies of the simulation bandwidth).



There is also an Eigenmode solver module, useful for finding the natural resonant frequencies of a particular structure. We will not use this module at Antennas and Propagation course.

2.7. Defining the boundary conditions

Regardless of the simulation method we use, computation in infinite space cannot be done. We must define the boundary conditions (**Simulation Domain > Boundary Conditions**), the conditions encountered on sides of the parallelepiped that delimits the computing volume.



The most common values are Absorbing (equivalent to free space, all energy exits, nothing returns), PEC (perfect electrical conductor - equivalent to the ideal metallic wall), PMC (perfect magnetic conductor - mathematical idealization, usable for odd symmetry of electromagnetic fields), Periodic (means the infinite repetition of the structure, usable for periodic structures - instead of simulating thousands of identical cells we can simulate only one).

2.8. Defining the computing mesh

Both common methods require discretization of the physical structure by defining a grid on whose specific points we will define and compute the fields. The meshing method is dependent on the simulation method (options are found in **Simulation Domain > ...**, only some of them are active depending on which of the two solvers is chosen for simulation).

FEM divides the volume into tetrahedra and has an automatic meshing algorithm, the meshing being finer around the drawn objects and with larger tetrahedra to the edge of the structure.

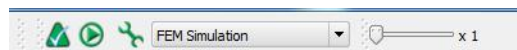
FDTD is based on a rectangular grid that largely requires user control to define grid dimensions, possibly finer grid areas (also around objects).

2.9. Defining the energy sources

Sources definition is also dependent on the chosen solver. Energy sources or access points (input/output) are called ports and are defined in the Project Tree by right clicking on **Circuit Components/Ports**. The option **New Circuit Component with > New Feed Definition** is valid for both solvers. For FDTD, the form of the excitation signal must be defined (see the previous Waveform Editor figure), for FEM there is also the possibility of defining waveguide sources sources (a whole boundary surface has the fields imposed acting as a source) **Circuit Components/Ports > New Waveguide Port**.

2.10. Solver options

The solver options are dependent on the simulation method and are accessible by clicking the green wrench button in the Simulation Control section. They are not treated in this laboratory because they will be the subject of a future laboratory.



2.11. Simulare și vizualizare rezultate

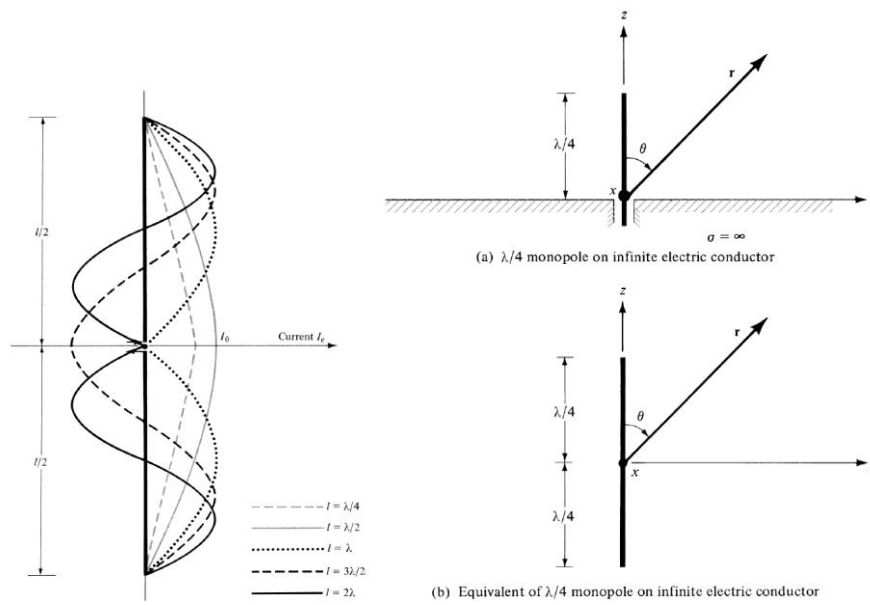
The simulation starts clicking the green "Play" button next to the above-mentioned green wrench button. The way the results are visualized is dependent on the solver and other signal characteristics. What is calculated and eventually displayed can be controlled in the **Sensors** section in the Project Tree (except the S parameters that are always calculated). This subject will also be covered in a future laboratory.

3. Activities in the laboratory

We practice drawing of typical antennas.

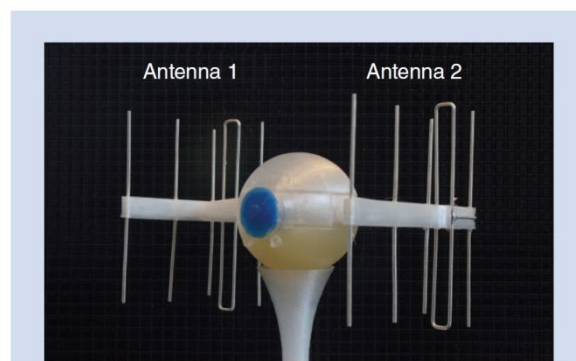
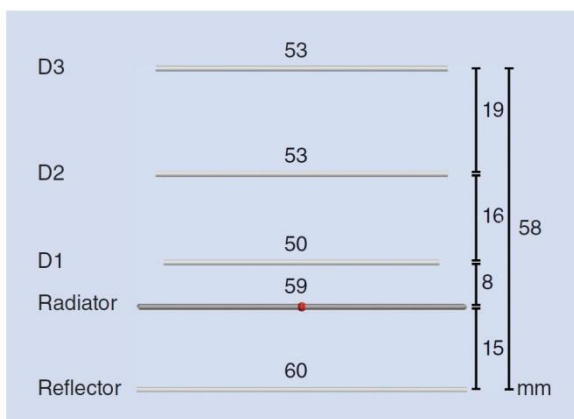
3.1. Half-Wavelength Dipole

Draw/simulate the Half-Wavelength Dipole and the equivalent created by Quarter-Wavelength monopole on an infinite perfect electric conductor. Calculate the length so that the working frequency is around 1.9 GHz. The dipole/monopole is created from a copper wire with 1mm diameter, and the slot in the $\lambda/2 \times \lambda/2$ rectangular metallic wall has a 2mm diameter.



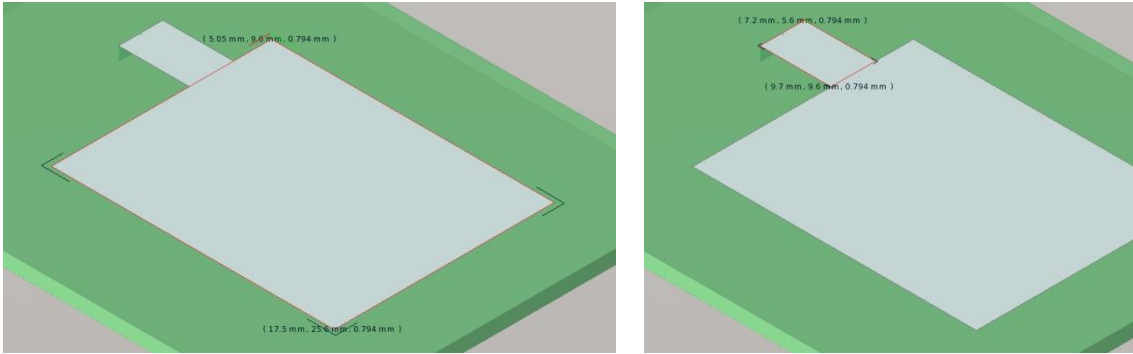
3.2. Yagi-Uda Antenna

Draw/simulate the five-element Yagi-Uda antenna in the following figure made of 1mm copper metallic wire. Working frequency $\sim 2.45\text{GHz}$. Choose for the Radiator element a rectangular shape with a square cross-section conductor (1mm edge). The Radiator element is perpendicular on the plane in which the Reflector and Director elements are placed.

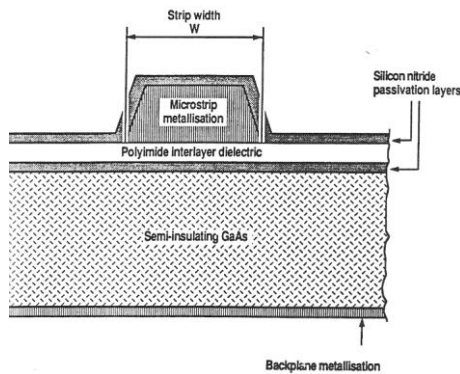


3.3. Microstrip Patch Antenna

Draw/simulate the microstrip patch antenna with the dimensions of $12.45 \times 16\text{mm}$, with a $2.5 \times 4\text{mm}$ feeding line, as in the following figure (working frequency $\sim 7.45\text{GHz}$).



This antenna is placed on a substrate with $\epsilon_r = 2.2$ and the height of $h = 0.794\text{mm}$ (RTDuroid). The substrate is a $40 \times 40\text{mm}$ square. Draw/simulate the antenna in the simple (plane) model with PEC **and** in the technologic accurate version with 0.1mm copper metallization on the same type of substrate and passivation with 0.01mm Si_3N_4 (materials from the Standard Material Library).



3.4. Optional

If you finished drawing the 5 previous antennas, try to draw/simulate the above five element Yagi-Uda antenna with a rounded corners Radiator made with circular cross-section conductor (1mm diameter).