Created in COMSOL Multiphysics 6.1

Optical Yagi–Uda Antenna

Introduction

A Yagi–Uda antenna is a well-known antenna type in RF and microwave communication. The most common Yagi–Uda antenna consists of a feed element, a reflector, and an array of directors. The lengths and spacings of the elements are specifically designed to achieve high directivity.

By utilizing the strong plasmonic response of metallic nanostructures, the concept of antenna can be generalized to the optical regime. Optical antennas offer important functionalities such as the manipulation of light at the nanoscale.

This model demonstrates the modeling of an optical Yagi–Uda antenna made of aluminum nanorods using the boundary element method (BEM) in the Electromagnetic Waves, Boundary Elements interface of the Wave Optics Module.

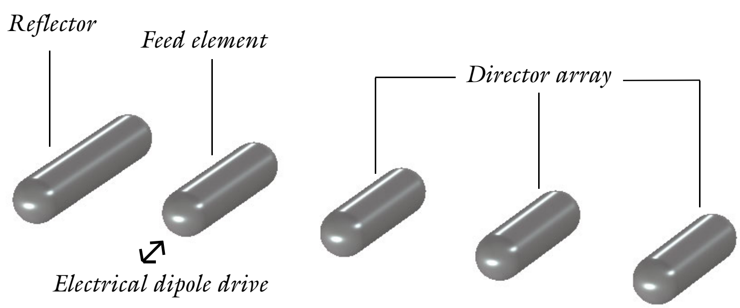


Figure 1: An optical Yagi–Uda antenna made of aluminum nanorods.

Model Definition

The optical Yagi–Uda antenna is constituted of aluminum nanorods of radius 20 nm. The nanorods have hemispherical caps on both sides. The antenna is designed to operate at 570 nm wavelength, where aluminum has permittivity of –38 – 10.9j. Due to the finite loss of metal at optical frequency, the lengths and spacings of the elements are different from an RF Yagi–Uda antenna for which the electromagnetic loss is negligible. In this particular design as shown in Ref. 1, the length of the feed element is 160 nm. The lengths of the reflector and director are 200 nm and 144 nm, respectively. The spacing between the reflector and the feed is about 130 nm. The spacing between the feed and the first director, as well as the director array spacing, is about 140 nm.

An optical Yagi–Uda antenna usually is driven by a quantum emitter such as a quantum dot near the end of the feed element. In simulation, the quantum emitter can be approximated as a classical electrical point dipole. To implement the electrical point dipole drive, the scattered field formulation was used with the background field set as the dipole field. Assuming the dipole is located at (x_p, y_p, z_p), the electric field at a point (x, y, z) in space is then given by the classical dipole field

$$\mathbf{E}(\mathbf{R}) = \frac{1}{4\pi\varepsilon_0}\left\{\frac{\omega^2}{c^2 R}(\hat{\mathbf{R}} \times \mathbf{p}) \times \hat{\mathbf{R}} + \left(\frac{1}{R^3} + \frac{j\omega}{cR^2}\right)(3\hat{\mathbf{R}}[\hat{\mathbf{R}} \cdot \mathbf{p}] - \mathbf{p})\right\} e^{-j\frac{\omega}{c}r} e^{j\omega t}, \quad (1)$$

where $\mathbf{p} = (p_x, p_y, p_z)$ is the dipole moment, c is the speed of light, ε_0 is the vacuum permittivity, ω is the angular frequency, $\mathbf{R} = (x - x_p, y - y_p, z - z_p)$ is the distance vector from the dipole position to the field position in space, $\hat{\mathbf{R}}$ is the unit vector of $\mathbf{R}$, and R is the norm of $\mathbf{R}$. In this model, we only simulate an electric dipole located at (0,–100 nm, 0) and oriented in the y direction, that is, $\mathbf{p} = (0, p_y, 0)$. To avoid entering long expressions for each component of the background field, we define variables $K = 1/(4\pi\varepsilon_0)$, $F_1 = \omega^2/(c^2 R)$, $F_2 = 1/R^3 + j\omega/(cR^2)$, and $F_3 = \exp(-j\omega r/c)$ in the model.

Results and Discussion

In the finite element method (FEM) used in the Electromagnetic Waves, Frequency Domain interface, we can only evaluate field quantities within the finite simulation domain. The advantage of BEM is that field quantities can be evaluated everywhere, even if only boundaries are meshed. This is well suited for radiation problems. For example, in this model, the air domain surrounding the antenna will not be meshed. Nevertheless, the field evaluation can be done by using a Grid 3D dataset. We set the dataset range in the x, y, and z directions to –5000 nm to 5000 nm. The resolutions are set to 200 to ensure we can resolve the details of the field distribution. The plotted norm of the electric field is shown in Figure 2. This dataset covers a volume of ~20λ-by-~20λ-by-~20λ. It is generally not feasible to simulate such a large domain in FEM. Besides the Grid 3D dataset, other datasets such as Parameterized Surface and Parameterized Curve 3D can also be used to evaluate quantities of interest.

Furthermore, in order to obtain the far-field radiation characteristics of the antenna, a Far-Field Calculation node is applied on the boundaries of all antenna elements. This enables the evaluation of the far field quantities in postprocessing. The far-field radiation pattern shows a high directivity, which verifies the validity of the design.

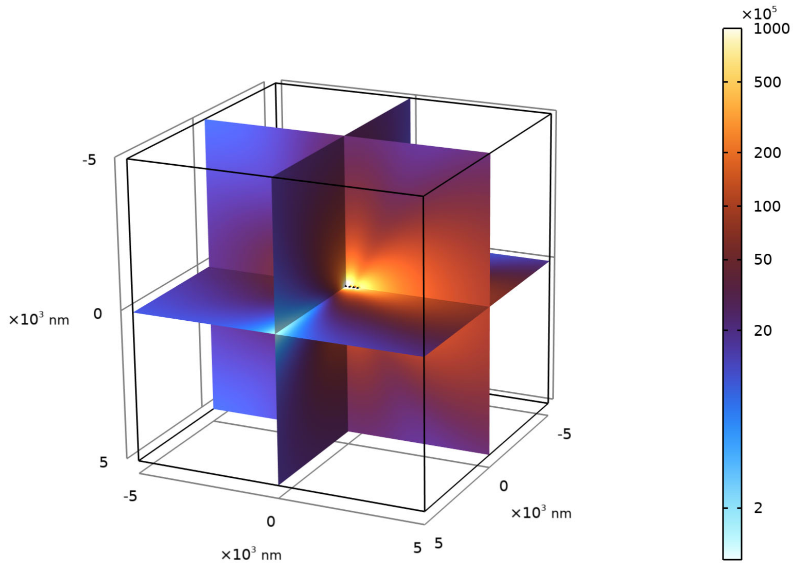


Figure 2: Norm of the electric field around the optical Yagi–Uda antenna. Using the Grid 3D dataset, we can evaluate the field quantities at arbitrary locations.

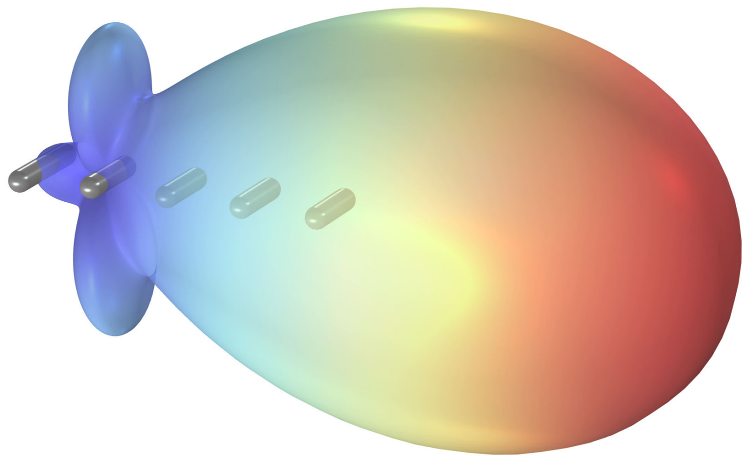


Figure 3: Far-field radiation pattern of the optical Yagi–Uda antenna, showing a high directivity.

Notes About the COMSOL Implementation

This model uses the Electromagnetic Waves, Boundary Elements interface to demonstrate its functionalities. In principle, the Electromagnetic Waves, Frequency Domain interface can be used to model the same structure and achieve the same results. Both BEM and FEM solve the full Maxwell equations but FEM requires a finite simulation domain with volumetric meshing while BEM can model infinite domain and only require boundary meshing. Although the degrees of freedom in a BEM model are generally fewer compared to FEM, the memory and computation time requirements are not necessarily smaller. Therefore, one method could be more efficient than the other, depending on the type of problem to be solved.

When using the Electromagnetic Waves, Boundary Elements interface to model dielectrics scatterers, additional Wave Equation, Electric nodes must be added and applied to all finite domains representing the scatterers. The first Wave Equation, Electric node should only contain the finite void when using the scattered field formulation. The best practice is to apply one separate Wave Equation, Electric node to each enclosed domain individually. In this model, however, we only apply one Wave Equation, Electric node to all the dielectric domains since there is only one material involved.

Reference

1. T.H. Taminiau, F.D. Stefani, and N.F. van Hulst, "Enhanced directional excitation and emission of single emitters by a nano-optical Yagi–Uda antenna," *Optics Express*, vol. 165, no. 14, pp. 10858–10866, 2008.

Application Library path: `Wave_Optics_Module/Optical_Scattering/optical_yagi_uda_antenna`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Optics>Wave Optics>Electromagnetic Waves, Boundary Elements (ebem)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Wavelength Domain**.

6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **nm**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
py	4e-24[C*m]	4E-24 C·m	Dipole moment, y-component
xp	0[nm]	0 m	Dipole position, x-component
yp	-100[nm]	-1E-7 m	Dipole position, y-component
zp	0[nm]	0 m	Dipole position, z-component
lda0	570[nm]	5.7E-7 m	Wavelength
epsilon	-38-10.9j	-38-10.9i	Dielectric constant of aluminum
L_f	160[nm]	1.6E-7 m	Feed length
L_d	0.9*L_f	1.44E-7 m	Director length
L_r	1.25*L_f	2E-7 m	Reflector length
a_d	lda0/4	1.425E-7 m	Director spacing
a_r	lda0/4.4	1.2955E-7 m	Reflector spacing
r	20[nm]	2E-8 m	Antenna radius

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
R	sqrt((x-xp)^2+(y-yp)^2+(z-zp)^2)	m	Distance from the point-dipole to a point in space
Rx	(x-xp)/R		Unit vector from the point-dipole to a point in space, x-component
Ry	(y-yp)/R		Unit vector from the point-dipole to a point in space, y-component
Rz	(z-zp)/R		Unit vector from the point-dipole to a point in space, z-component
K	1/(4*pi*epsilon0_const)	m/F	Variable to simplify expressions in the background field
F1	ebem.omega^2/(c_const^2*R)	$1/m^3$	Variable to simplify expressions in the background field
F2	1/R^3+j*ebem.omega/(c_const*R^2)	$1/m^3$	Variable to simplify expressions in the background field
F3	exp(-j*ebem.omega*R/c_const)		Variable to simplify expressions in the background field

Here we define the distance from the point dipole position to a point in space at (x,y,z) as well as the components of the unit vector connecting them. Variabls K, F1, F2, and F3 are defined for convenience to avoid typing long and repeated expressions in the background field.

GEOMETRY 1

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type r.

4 In the **Height** text field, type L_r-2*r.

5 Locate the **Position** section. In the **x** text field, type -a_r.

6 In the **y** text field, type -(L_r-2*r)/2.

7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Cylinder 2 (cyl2)

1 Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Height** text field, type L_f-2*r.

4 Locate the **Position** section. In the **x** text field, type 0.

5 In the **y** text field, type -(L_f-2*r)/2.

Cylinder 3 (cyl3)

1 Right-click **Cylinder 2 (cyl2)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Height** text field, type L_d-2*r.

4 Locate the **Position** section. In the **y** text field, type -(L_d-2*r)/2.

5 In the **x** text field, type a_d.

Sphere 1 (sph1)

1 In the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type r.

4 Locate the **Position** section. In the **y** text field, type L_f/2-r.

Sphere 2 (sph2)

1 Right-click **Sphere 1 (sph1)** and choose **Duplicate**.

2 In the **Settings** window for **Sphere**, locate the **Position** section.

3 In the **y** text field, type -L_f/2+r.

Sphere 3 (sph3)

1 Right-click **Sphere 2 (sph2)** and choose **Duplicate**.

2 In the **Settings** window for **Sphere**, locate the **Position** section.

3 In the **x** text field, type -a_r.

4 In the **y** text field, type -L_r/2+r.

Sphere 4 (sph4)

1 Right-click **Sphere 3 (sph3)** and choose **Duplicate**.

2 In the **Settings** window for **Sphere**, locate the **Position** section.

3 In the **y** text field, type `L_r/2-r`.

Sphere 5 (sph5)

1 Right-click **Sphere 4 (sph4)** and choose **Duplicate**.

2 In the **Settings** window for **Sphere**, locate the **Position** section.

3 In the **x** text field, type `a_d`.

4 In the **y** text field, type `L_d/2-r`.

Sphere 6 (sph6)

1 Right-click **Sphere 5 (sph5)** and choose **Duplicate**.

2 In the **Settings** window for **Sphere**, locate the **Position** section.

3 In the **y** text field, type `-L_d/2+r`.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the objects **cyl3**, **sph5**, and **sph6** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `3`.

5 Locate the **Displacement** section. In the **x** text field, type `a_d`.

MATERIALS

Air

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Air` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **All voids**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

Al

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Al` in the **Label** text field.

3 Click the **Select All** button in the **Graphics** toolbar.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	`epsilon`	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

Next, we change the **Formulation** to **Scattered field** and set up the background field, which is the field created by an electrical point dipole as given by equation (1).

ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS (EBEM)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Boundary Elements (ebem)**.

2 In the **Settings** window for **Electromagnetic Waves, Boundary Elements**, locate the **Formulation** section.

3 From the list, choose **Scattered field**.

4 Specify the $\mathbf{E}_b$ vector as

`K*(F1*(-Rx*Ry)+F2*3*Rx*Ry)*F3*py`	x
`K*(F1*(-(Rz^2+Rx^2))+F2*(3*Ry^2-1))*F3*py`	y
`K*(F1*(-Ry*Rz)+F2*3*Ry*Rz)*F3*py`	z

Wave Equation, Electric 2

1 In the **Physics** toolbar, click **Domains** and choose **Wave Equation, Electric**.

A **Wave Equation, Electric** node is added and applied to the antenna domains. This node includes all the finite domains of the same material. The first **Wave Equation, Electric** node should only contains the infinite void when using the scattered field formulation.

2 Click the **Select All** button in the **Graphics** toolbar.

Far-Field Calculation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 Click the **Select All** button in the **Graphics** toolbar.

Far-Field Calculation is added so later in the postprocessing we can evaluate the far field radiation pattern.

STUDY 1

Step 1: Wavelength Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Wavelength Domain**.

2 In the **Settings** window for **Wavelength Domain**, locate the **Study Settings** section.

3 In the **Wavelengths** text field, type `lda0`.

4 In the **Home** toolbar, click **Compute**.

RESULTS

Grid 3D 1

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Grid 3D 1**.

2 In the **Settings** window for **Grid 3D**, locate the **Parameter Bounds** section.

3 Find the **First parameter** subsection. In the **Minimum** text field, type `-5000`.

4 In the **Maximum** text field, type `5000`.

5 Find the **Second parameter** subsection. In the **Minimum** text field, type `-5000`.

6 In the **Maximum** text field, type `5000`.

7 Find the **Third parameter** subsection. In the **Minimum** text field, type `-5000`.

8 In the **Maximum** text field, type `5000`.

9 Click to expand the **Grid** section. In the **x resolution** text field, type `200`.

10 In the **y resolution** text field, type `200`.

11 In the **z resolution** text field, type `200`.

In FEM, we can only visualize the physical quantities such as the electric field within the simulation domain. However, in BEM, we are free to visualize them everywhere by using the Grid 3D dataset. This dataset is added by default.

Electric Field, Domains (ebem)

1 In the **Model Builder** window, under **Results** click **Electric Field, Domains (ebem)**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

Multislice 1

1 In the **Model Builder** window, expand the **Electric Field, Domains (ebem)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, click to expand the **Range** section.

3 Select the **Manual color range** check box.

4 In the **Minimum** text field, type `0`.

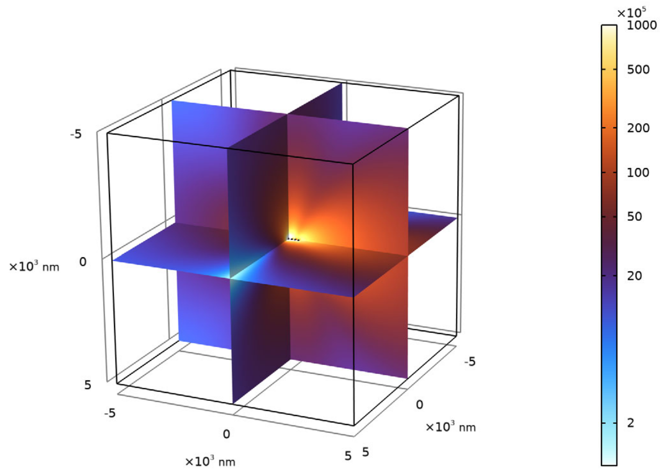
5 In the **Maximum** text field, type `1e8`.

6 Locate the **Coloring and Style** section. From the **Scale** list, choose **Logarithmic**.

7 Click **Change Color Table**.

8 In the **Color Table** dialog box, select **Thermal>ThermalWave** in the tree.

9 Click **OK**.



First plot the norm of the electric field around the Yagi-Uda antenna. To enhance the visualization, use a logarithmic scale. Next, plot the far-field radiation pattern.

3D Plot Group 4

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

Radiation Pattern 1

1 In the **3D Plot Group 4** toolbar, click **More Plots** and choose **Radiation Pattern**.

2 In the **Settings** window for **Radiation Pattern**, locate the **Expression** section.

3 In the **Expression** text field, type `ebem.normEfar^2`.

4 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of elevation angles** text field, type `100`.

5 In the **Number of azimuth angles** text field, type `100`.

6 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

Transparency 1

1 Right-click **Radiation Pattern 1** and choose **Transparency**.

2 In the **Settings** window for **Transparency**, locate the **Transparency** section.

3 Set the **Transparency** value to **0.35**.

Surface 1

In the **Model Builder** window, right-click **3D Plot Group 4** and choose **Surface**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Material Appearance**.

2 Click the **Show Axis Orientation** button in the **Graphics** toolbar.

3 Click the **Show Grid** button in the **Graphics** toolbar.

4 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

5 From the **Appearance** list, choose **Custom**.

6 From the **Material type** list, choose **Aluminum**.

7 Click **Customize**.

8 Set the **Reflectance at normal incidence** value to **0.45**.

9 Set the **Surface roughness** value to **0.4**.

10 Set the **Anisotropy** value to **0.35**.

The visualization is enhanced by adding **Material Appearance** on the antenna surface.