RF PCB Toolbox™

Reference

MATLAB®

R2021b

MathWorks®

How to Contact MathWorks

Latest news:	`www.mathworks.com`
Sales and services:	`www.mathworks.com/sales_and_services`
User community:	`www.mathworks.com/matlabcentral`
Technical support:	`www.mathworks.com/support/contact_us`
Phone:	508-647-7000

The MathWorks, Inc.
1 Apple Hill Drive
Natick, MA 01760-2098

RF PCB Toolbox™ Reference

Revision History

September 2021	Online only	New for Version 1.0 (R2021b)

Contents

1

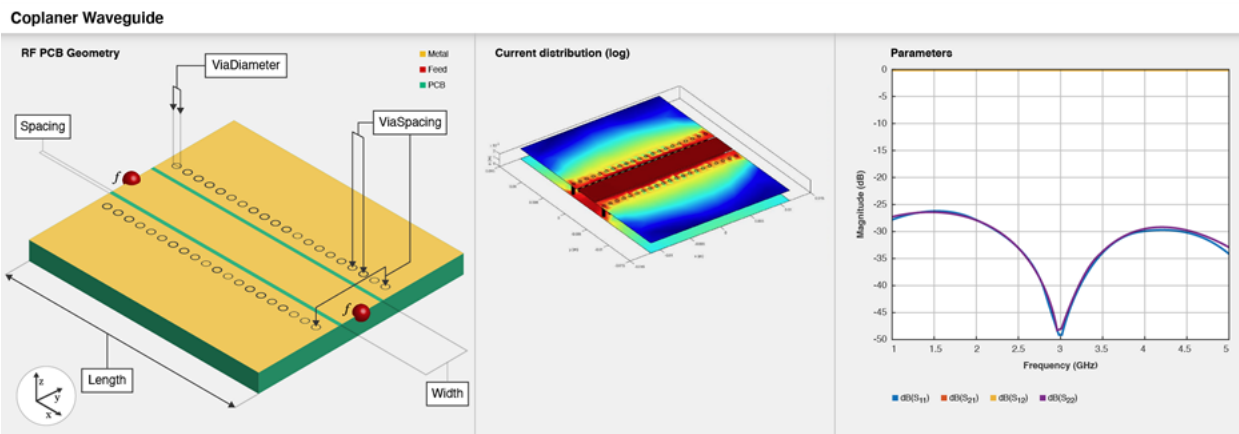
Objects

coplanarWaveguide

Create coplanar waveguide transmission line

Description

Use the `coplanarWaveguide` object to create a coplanar waveguide transmission line. Coplanar waveguide is a common type of transmission line used in any PCB implementation of RF and microwave components. A coplanar waveguide transmission line has a center conductor strip and two ground planes. One ground plane is the layer that acts as the conductor strip and the other ground plane is the bottom layer.



To analyze the behavioral model of a coplanar waveguide transmission line, set the `Behavioral` property in the `sparameters` object function to `true or 1`.

Creation

Syntax

```
cpgw = coplanarWaveguide
cpgw = coplanarWaveguide(Name=Value)
```

Description

`cpgw = coplanarWaveguide` creates a default coplanar waveguide transmission line with a Teflon substrate. The default property values are for a 50 ohm transmission line.

`cpgw = coplanarWaveguide(Name=Value)` sets "Properties" on page 1-3 using one or more name-value arguments. For example, `coplanarWaveguide(Width=0.0047)` creates a coplanar waveguide transmission line of width 0.0047 meters. Properties not specified retain their default values.

Properties

`Length` — Length of coplanar waveguide transmission line
`0.0231` (default) | positive scalar

Length of the coplanar waveguide transmission line in meters, specified as a positive scalar.

Example: `cpgw = coplanarWaveguide(Length=0.0300)`

Data Types: `double`

`Width` — Width of coplanar waveguide transmission line
`0.0039` (default) | positive scalar

Width of the coplanar waveguide transmission line in meters, specified as a positive scalar.

Example: `cpgw = coplanarWaveguide(Width=0.0047)`

Data Types: `double`

`Spacing` — Distance between transmission line and adjacent ground plane
`2.0000e-04` (default) | positive scalar

Distance between the transmission line and the adjacent top layer metal of the ground plane, specified as a positive scalar in meters.

Example: `cpgw = coplanarWaveguide(Spacing=3.0000e-04)`

Data Types: `double`

`ViaSpacing` — Distance between vias
`[0.0011 0.0070]` (default) | two-element vector

Distance between the vias, specified as a two-element vector of positive elements.

Example: `cpgw = coplanarWaveguide(ViaSpacing=[0.0021 0.0060])`

Data Types: `double`

`ViaDiameter` — Diameter of via
`5.0000e-04` (default) | positive scalar

Diameter of the via in meters, specified as a positive scalar.

Example: `cpgw = coplanarWaveguide(ViaDiameter=7.0000e-04)`

Data Types: `double`

`Height` — Height of coplanar waveguide transmission line
`0.0016` (default) | positive scalar

Height of the coplanar waveguide transmission line from the ground plane, specified as a positive scalar in meters.

Example: `cpgw = coplanarWaveguide(Height=0.0020)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane
`0.0300` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `cpgw = coplanarWaveguide(GroundPlaneWidth=0.0350)`

Data Types: `double`

`Substrate` — Type of dielectric material
`'Teflon'` (default) | `dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object.

Example: `d = dielectric("FR4"); cpgw = coplanarWaveguide(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'PEC'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("PEC"); cpgw =coplanarWaveguide(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design coplanar waveguide transmission line around particular frequency
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

Create Default Coplanar Waveguide

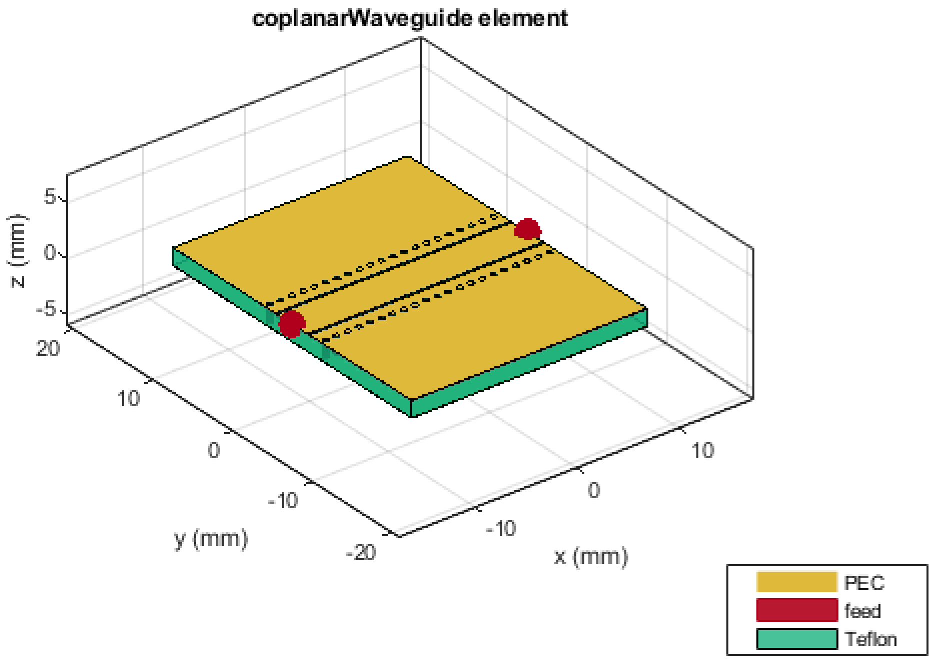
Create a coplanar waveguide transmission line.

```
waveguide = coplanarWaveguide
```

```
waveguide = 
  coplanarWaveguide with properties:

             Length: 0.0231
              Width: 0.0039
            Spacing: 2.0000e-04
         ViaSpacing: [0.0011 0.0070]
        ViaDiameter: 5.0000e-04
             Height: 0.0016
    GroundPlaneWidth: 0.0300
          Substrate: [1x1 dielectric]
          Conductor: [1x1 metal]
```

View the coplanar waveguide transmission line.

```
show(waveguide)
```



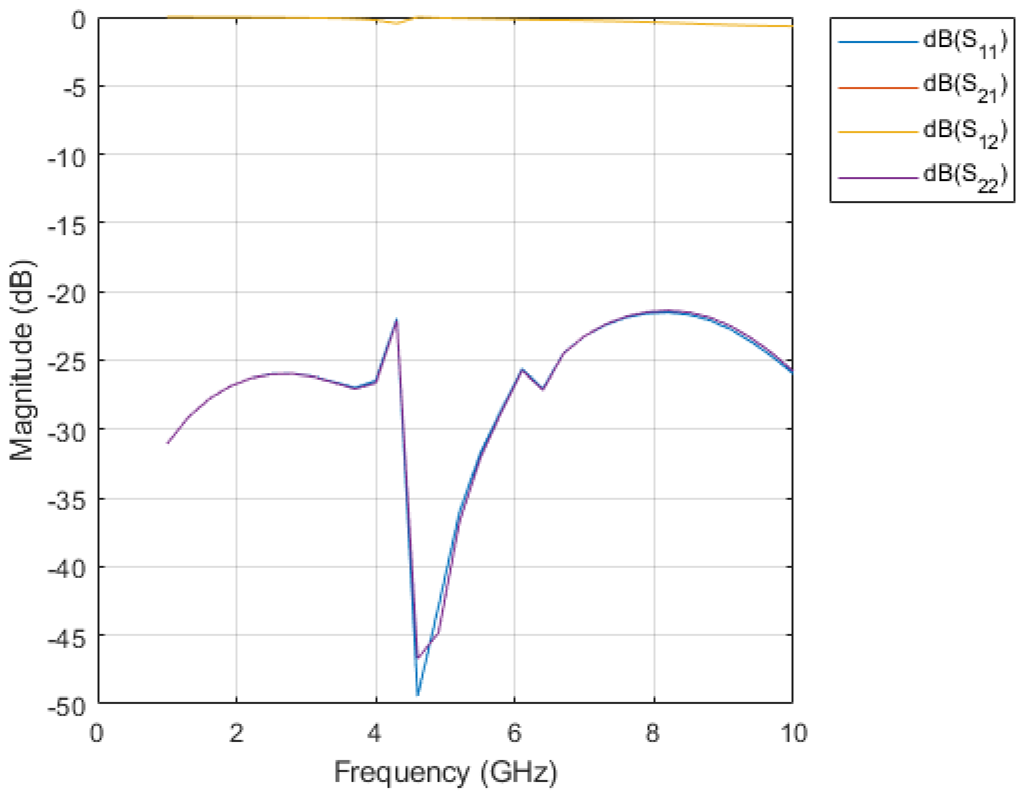
Calculate the S-parameters of the waveguide from 1-10 GHz.

```
sparam = sparameters(waveguide,1e9:0.3e9:10e9)
```

```
sparam = 
  sparameters: S-parameters object

       NumPorts: 2
    Frequencies: [31x1 double]
     Parameters: [2x2x31 double]
      Impedance: 50

  rfparam(obj,i,j) returns S-parameter Sij
```

Plot the S-parameters.

```
rfplot(sparam)
```



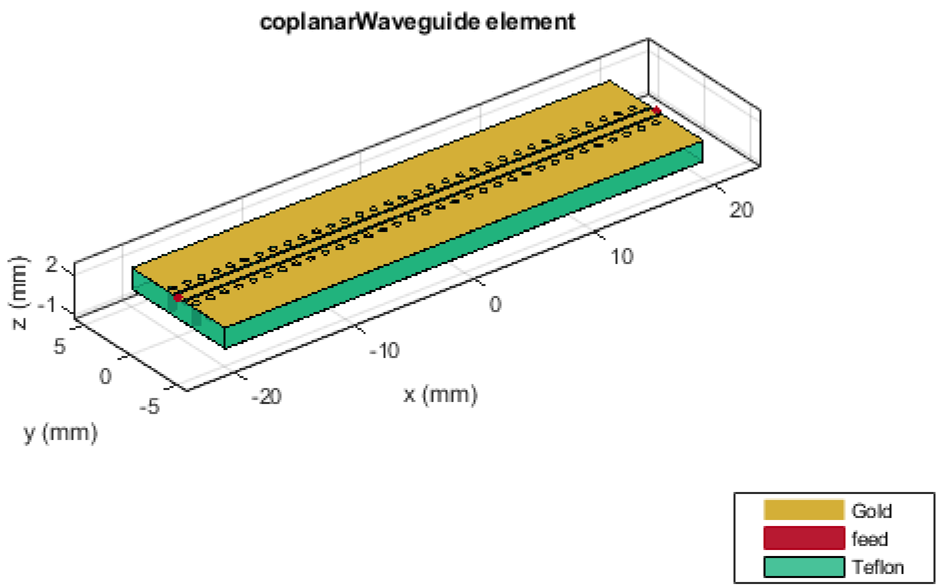
Behavioral S-parameters of Coplanar Waveguide Transmission Line

Create a coplanar waveguide transmission line using a gold substrate as the dielectric.

```
txem = coplanarWaveguide;
txem.Conductor = metal("Gold");
```

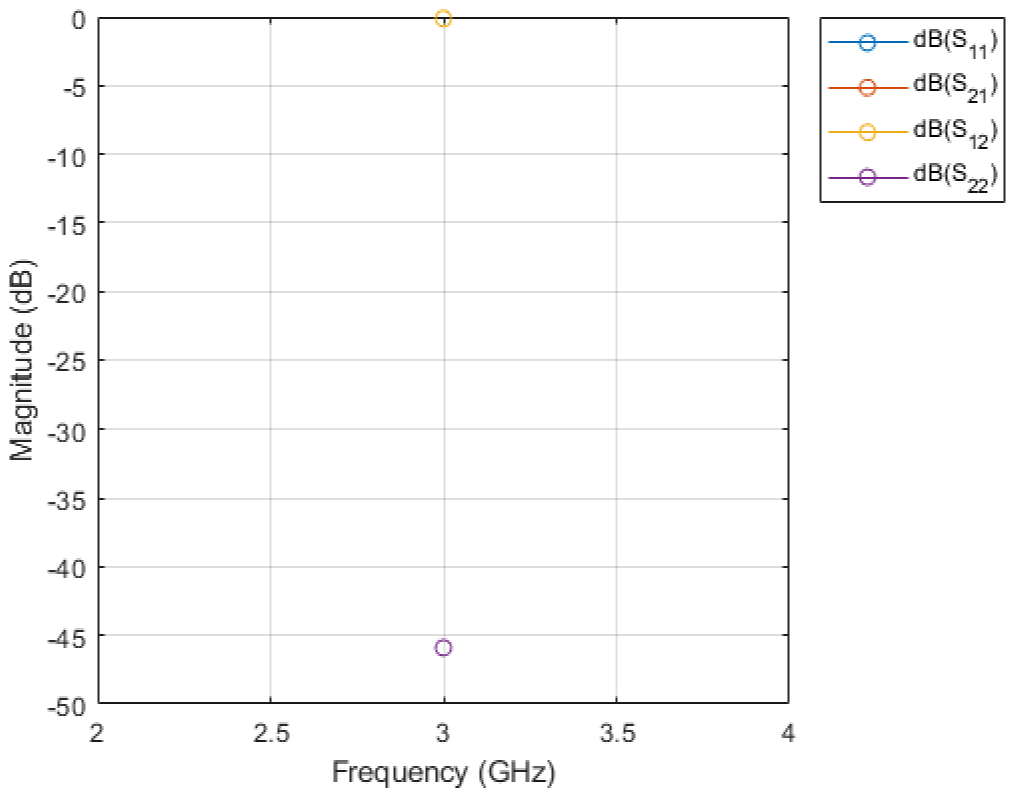
Design the coplanar waveguide at a frequency of 3 GHz, line length of 0.5 meters, and impedence of 75 ohms.

```
txem = design(txem,3e9,LineLength=0.5,Z0=75);
show(txem)
```



Compute and plot the behavioral S-parameters of the waveguide.

```
spar = sparameters(txem,3e9,Behavioral=true);
rfplot(spar)
```



References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

See Also

`microstripLine` | `coupledMicrostripLine` | `stripLine` | `coupledStripLine`

Introduced in R2021b

microstripLine

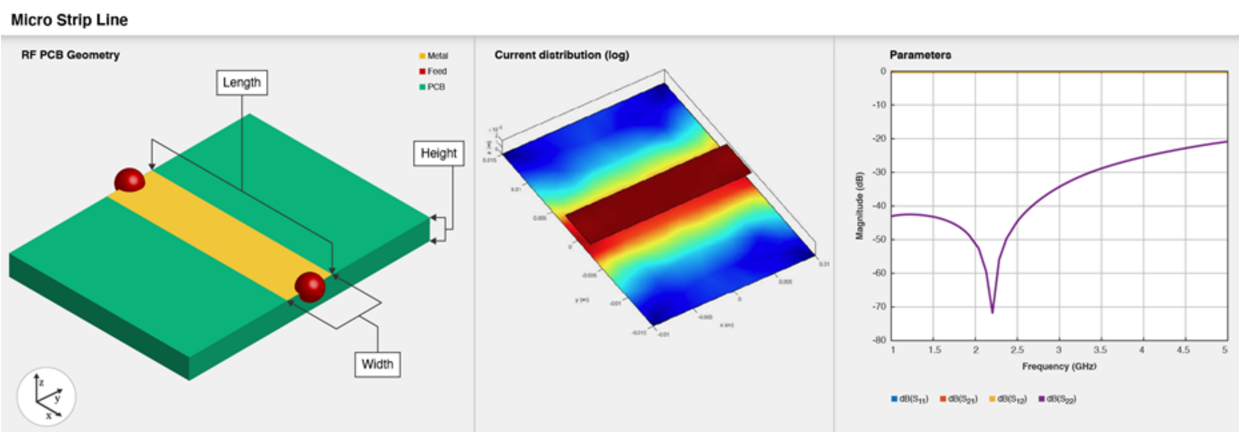
Create transmission line in microstrip form

Description

Use the `microstripLine` object to create a microstrip transmission line. Microstrip line is a transmission line that is a basic building block for most RF planar microwave devices. You can use this transmission line to connect between two PCB components or to create components such as filters, couplers, and feeding elements of various types of antennas.

A few applications of microstrip transmission lines are:

- Creating matching feed and coupling networks
- Transmitting power from one component to another
- Feeding planar antennas and coupling structures
- Creating varying inductances or capacitances using open- or short ended- transmission lines



To analyze the behavioral model for a microstrip transmission line, set the `Behavioral` property in the `sparameters` function to `true or 1`.

Creation

Syntax

```
microstrip = microstripLine
microstrip = microstripLine(Name,Value)
microstrip = microstripLine(txlineobj)
```

Description

`microstrip = microstripLine` creates a default microstrip transmission line using a Teflon substrate.

`microstrip = microstripLine(Name,Value)` sets properties using one or more name value pair arguments. For example, `microstrip = microstripLine('Length',0.0300)` creates a microstrip line of length 0.0300 meters. Properties not specified retain their default values.

`microstrip = microstripLine(txlineobj)` creates a microstrip transmission line from the behavioral model of a `txlineMicrostrip` object in RF Toolbox™.

Properties

`Length` — Length of microstrip line in meters

`0.0200` (default) | positive scalar

Length of the microstrip line in meters, specified as a positive scalar.

Example: `microstrip = microstripLine('Length',0.0300)`

Data Types: `double`

`Width` — Width of microstrip line in meters

`0.0050` (default) | positive scalar

Width of the microstrip line in meters, specified as a positive scalar.

Example: `microstrip = microstripLine('Width',0.00630)`

Data Types: `double`

`Height` — Height of microstrip line

`0.0016` (default) | positive scalar

Height of the microstrip line from the ground plane in meters, specified as a positive scalar.

In case of a multilayer substrate, you can use the height property to create a microstrip line at the interface of the two dielectrics.

Example: `microstrip = microstripLine('Height',0.0015)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane in meters

`0.0300` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `microstrip = microstripLine('GroundPlaneWidth',0.0400)`

Data Types: `double`

`Substrate` — Type of dielectric material

`'Teflon'` (default) | `dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. For more information, see `dielectric`. The thickness of the default dielectric material Teflon is `0.0016` m or the same as the height property.

Example: `d = dielectric('FR4'); microstrip = microstripLine('Substrate',d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers

`'PEC'` (default) | `metal` object

Type of metal used in conducting layers, specified as a `metal` object. For more information see `metal`.

Example: `m = metal('PEC'); microstrip = microstripLine('Conductor',m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design microstrip transmission line around specified frequency
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

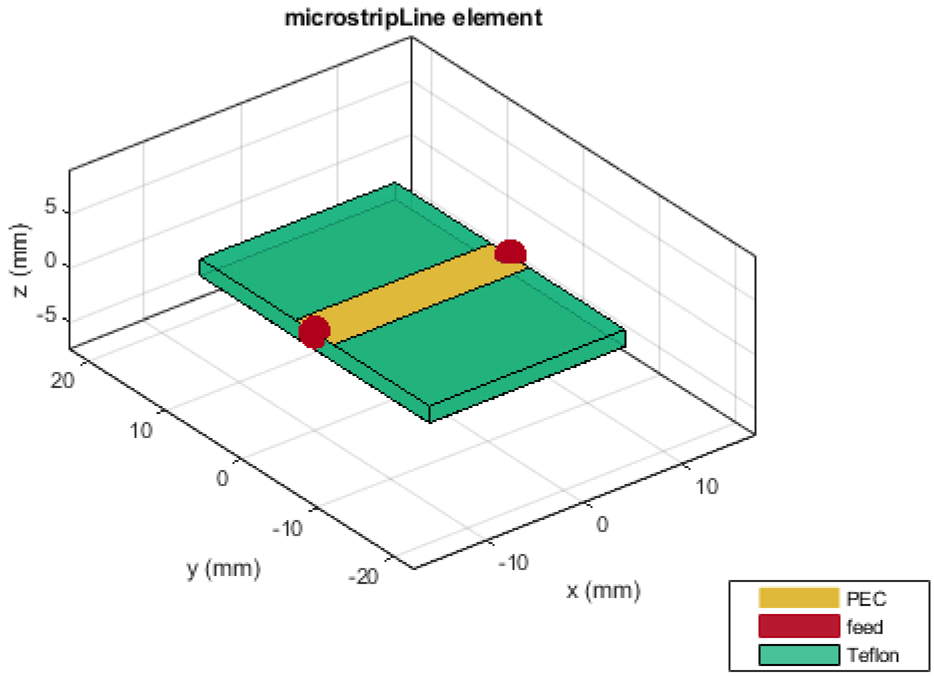
Default Microstrip Line

Create and view a default microstrip transmission line.

```
microstrip = microstripLine
```

```
microstrip = 
  microstripLine with properties:

             Length: 0.0200
              Width: 0.0050
             Height: 0.0016
    GroundPlaneWidth: 0.0300
           Substrate: [1x1 dielectric]
           Conductor: [1x1 metal]
```

```
show(microstrip)
```



Microstrip Transmission Line at 3 GHz

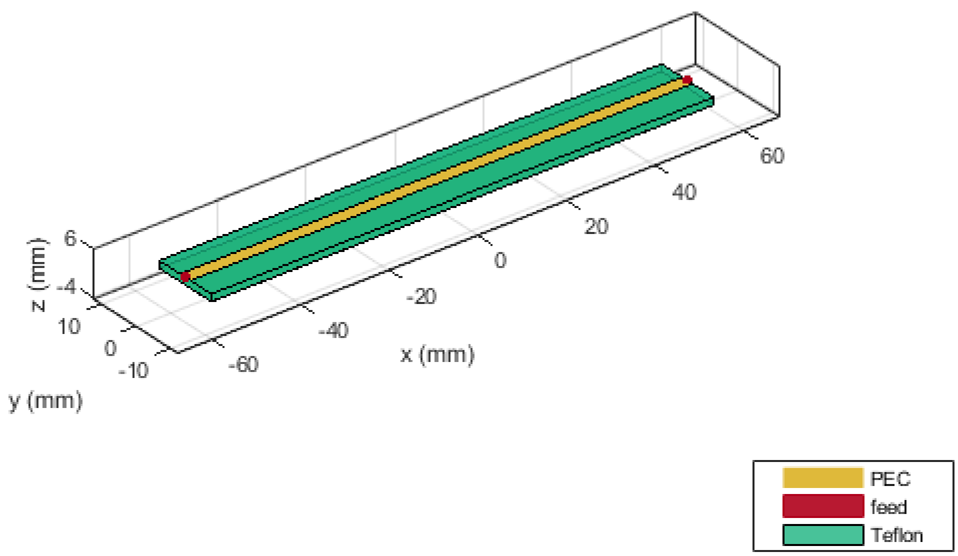
Design a microstrip transmission line at 3 GHz, with a characteristic impedance of 70 ohms and a line length 1.5 times the wavelength.

```
microstrip = design(microstripLine,3e9,'Z0',70,'LineLength',1.5)
```

```
microstrip = 
  microstripLine with properties:

             Length: 0.1132
              Width: 0.0030
             Height: 0.0016
    GroundPlaneWidth: 0.0150
          Substrate: [1×1 dielectric]
          Conductor: [1×1 metal]
```

View the microstrip transmission line.

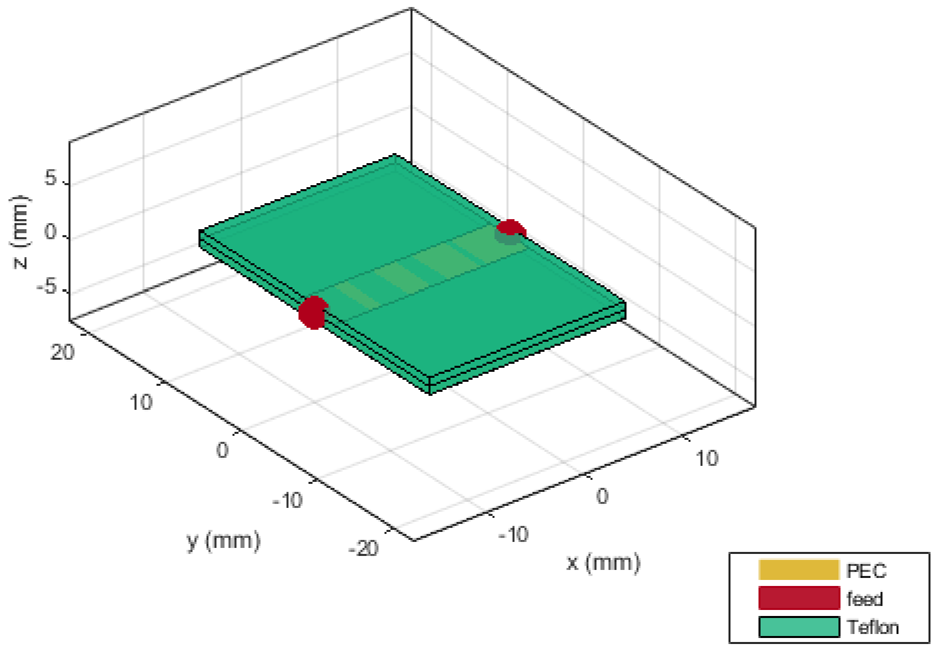
```
show(microstrip)
```



Multilayer Dielectric Microstrip Transmission Line

Create and view a multilayer dielectric microstrip transmission line.

```
microstrip = microstripLine;
microstrip.Substrate = dielectric('Name',{'Teflon','Teflon'},'EpsilonR', ...
    [2.1 2.1],'LossTangent',[0 0],'Thickness',[0.8e-3 0.8e-3]);
    microstrip.Height = 0.8e-3;
show(microstrip);
```



Use Behavioral Model to Calculate S-Parameters of Microstrip Cross

Design a microstrip transmission line at 3 GHz for FR4 substrate.

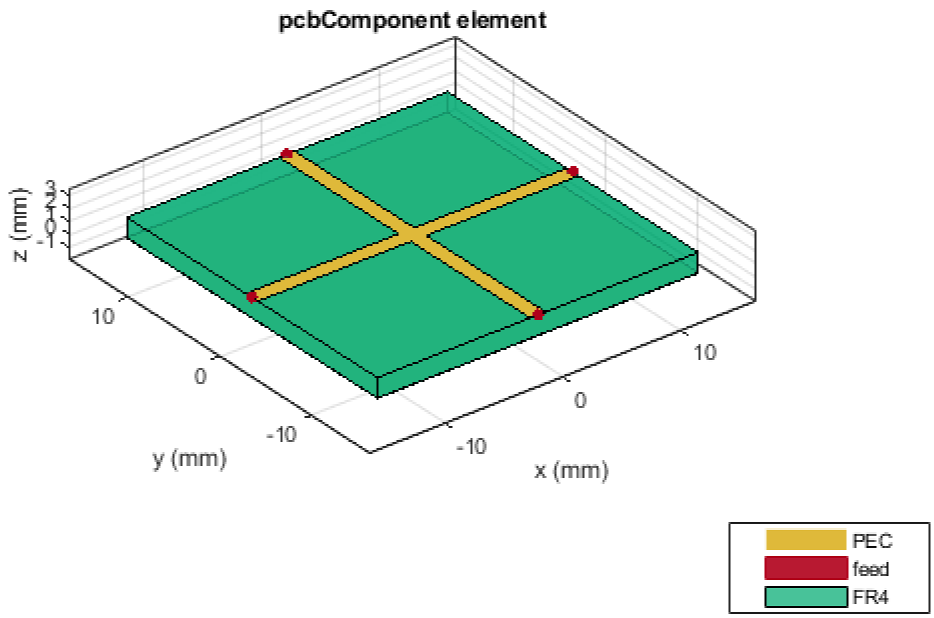
```
d = dielectric('FR4');
d.LossTangent = 0;
m = design(microstripLine('Substrate',d),3e9,'Z0',75,...
    'LineLength',0.5);
```

Create a microstrip cross.

```
layer2d = traceCross('Length',[m.Length m.Length], ...
    'Width',[m.Width m.Width]);
```

Convert the cross trace to a PCB component.

```
robj = pcbComponent(layer2d);
robj.BoardThickness = m.Substrate.Thickness;
robj.Layers{2} = m.Substrate;
show(robj)
```



Define frequency points to calculate the s-parameters.

```
freq = (1:3:40)*100e6;
```

Calculate the s-parameters of the cross trace using the behavioral model.

```
Sckt = sparameters(robj,freq,'Behavioral',true);
```

```
Warning: Behavioral model is valid only when EpsilonR is 9.9.
```

Calculate the s-parameters of the cross trace using the electromagnetic solver.

```
Sem = sparameters(robj,freq);
```

References:

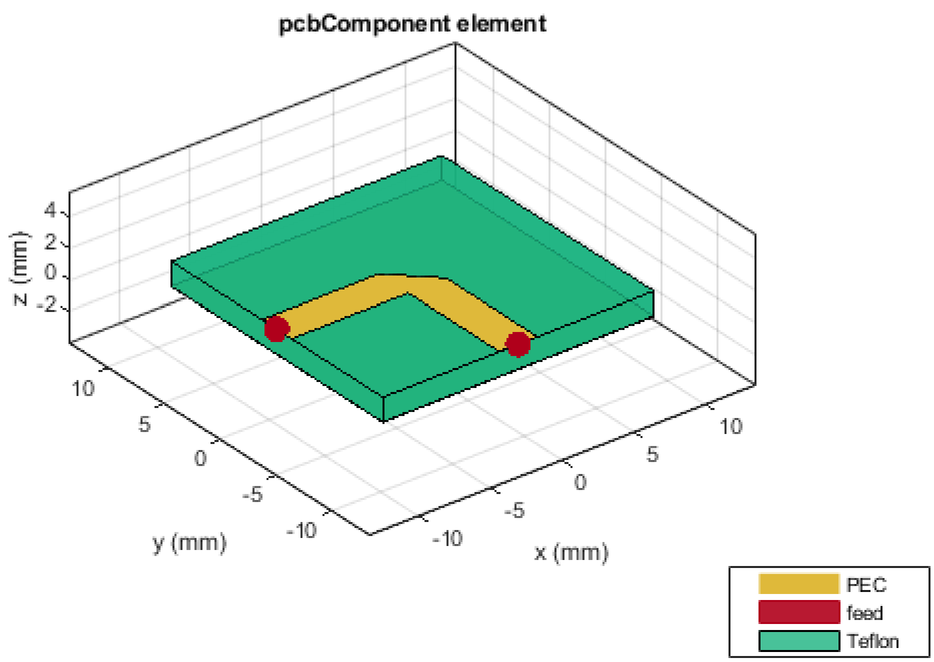
1 Ramesh Garg & I. J. Bahl (1978) Microstrip discontinuities, International Journal of Electronics, 45:1, 81-87, DOI: 10.1080/00207217808900883

2 Wadell, Brian C. *Transmission Line Design Handbook*. The Artech House Microwave Library. Boston: Artech House, 1991.

Use Behavioral Model to Calculate S-Parameters of Mitered Bend Microstrip

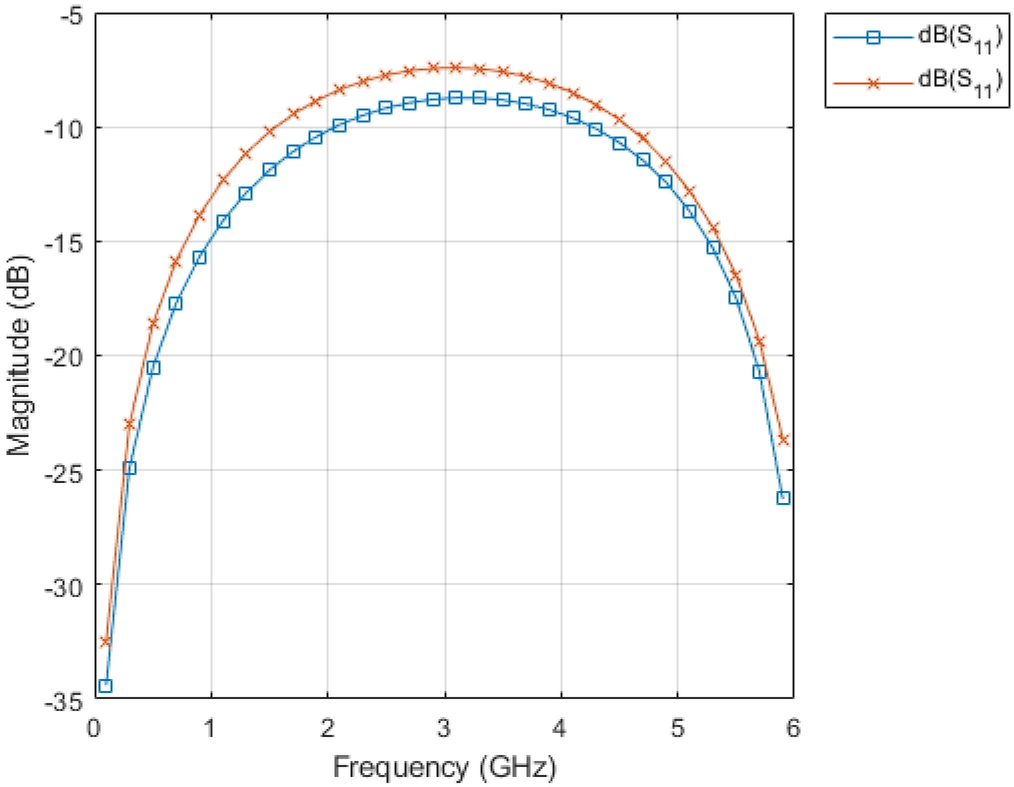
Create mitered bend microstrip.

```
m = design(microstripLine,6e9,"Z0",75);
layer2d = bendMitered('Length',[m.Length/2 m.Length/2],...
"Width",[m.Width m.Width],'MiterDiagonal',sqrt(2)*m.Width);
robj = pcbComponent(layer2d);
robj.BoardThickness = m.Substrate.Thickness;
robj.Layers{2} = m.Substrate;
show(robj)
```



Compute and plot s-parameters.

```
freq = (1:2:60)*100e6;
Sckt = sparameters(robj,freq,'Behavioral',true);
Sem = sparameters(robj,freq);
rfplot(Sckt,1,1,'db','-s')
hold on
rfplot(Sem,1,1,'db','-x')
```



Reference:

M. Kirschning, R. H. Jansen and N. H. L. Koster, "Measurement and Computer-Aided Modeling of Microstrip Discontinuities by an Improved Resonator Method," 1983 IEEE MTT-S International Microwave Symposium Digest, Boston, MA, USA, 1983, pp. 495-497, doi: 10.1109/MWSYM.1983.1130959.

References

[1] Pozar, David M. *Microwave Engineering / David M. Pozar*, University of Massachusetts at Amherst, 2012. https://public.ebookcentral.proquest.com/choice/publicfullrecord.aspx?p=2064708.

See Also

`coplanarWaveguide` | `coupledMicrostripLine`

Introduced in R2021b

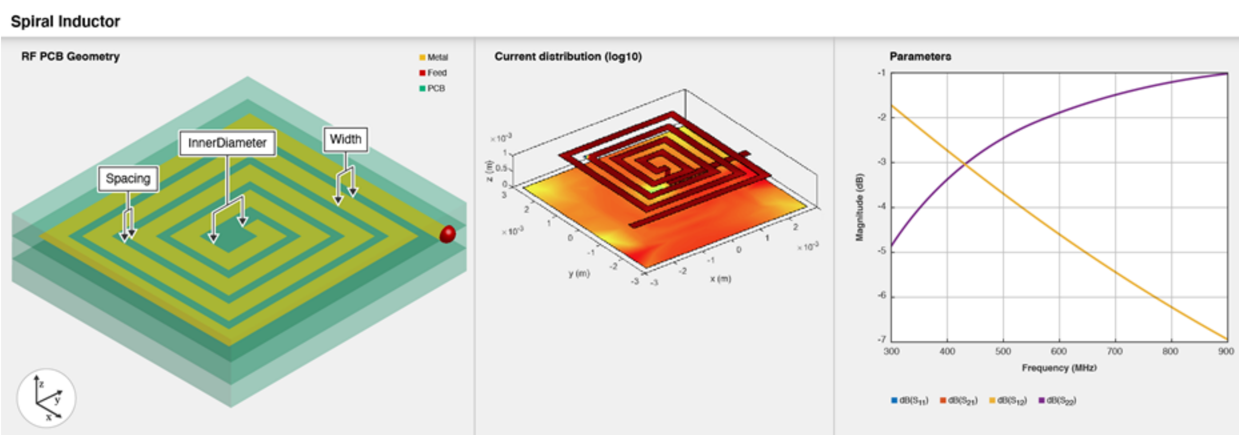
spiralInductor

Create spiral inductor in four different shapes

Description

Use the `spiralInductor` object to create a spiral inductor in one of four different shapes: square, circle, hexagon, or octagon. The spiral inductor is a two-port planar inductor with a single or multiple dielectric layers. A turn in a spiral inductor is the length of a complete 360-degree revolution. Spiral inductor filaments have uniform spacing and width throughout the structure. Spiral inductors are an integral part of many radio-frequency and microwave circuits, acting as resonant elements or chokes. The inductor feed can be configured in one of the following two ways:

- The input and output ports are punched through at the same layer.
- The input port is routed out from the layer below the inductor by a via hole. The output port is extended to the end of the dielectric in the same layer.



To analyze the behavioral model of a spiral inductor, set the `Behavioral` property in the `sparameters` function to `true or 1`.

Creation

Syntax

```
inductor = spiralInductor
inductor = spiralInductor(Name=Value)
```

Description

`inductor = spiralInductor` creates a square spiral planar inductor. The default properties are for a resonant frequency of 600 MHz.

`inductor = spiralInductor(Name=Value)` sets “Properties” on page 1-19 using one or more name-value arguments. For example, `spiralInductor(SpiralShape="Octagon")` creates an octagonal spiral inductor. Properties not specified retain their default values.

Properties

`SpiralShape` — Shape of spiral inductor
`"Square"` (default) | `"Circle"` | `"Hexagon"` | `"Octagon"`

Shape of the spiral inductor, specified as either `"Square"`, `"Circle"`, `"Hexagon"`, or `"Octagon"`.

Example: `inductor = spiralInductor(SpiralShape="Circle")`

Data Types: `string` | `char`

`InnerDiameter` — Inner diameter of polygon along edge
`5.0000e-04` (default) | positive scalar

Inner diameter of the polygon along the edge in meters, specified as a positive scalar.

Example: `inductor = spiralInductor(InnerDiameter=8.0000e-04)`

Data Types: `double`

`Width` — Strip width
`2.5000e-04` (default) | positive scalar

Strip width in meters, specified as a positive scalar.

Example: `inductor = spiralInductor(Width=3.8000e-04)`

Data Types: `double`

`Spacing` — Distance between strips
`2.5000e-04` (default) | positive scalar

Distance between the strips in meters, specified as a positive scalar.

Example: `inductor = spiralInductor(Spacing=3.8000e-04)`

Data Types: `double`

`NumTurns` — Number of turns in spiral inductor
`4` (default) | positive scalar

Number of turns in the spiral inductor, specified as a positive scalar. You can specify a minimum of `1` turn and a maximum of `12` turns. One turn length is the length of a complete 360-degree revolution.

Example: `inductor = spiralInductor(NumTurns=6)`

Data Types: `double`

`Height` — Height from ground plane to inductor
`0.0010` (default) | positive scalar

Height from the ground plane to the inductor in meters, specified as a positive scalar.

Example: `inductor = spiralInductor(Height=0.0056)`

Data Types: `double`

`GroundPlaneLength` — Length of ground plane
`0.0056` (default) | positive scalar

Length of the ground plane in meters, specified as a positive scalar. This object does not support infinite ground plane length.

Example: `inductor = spiralInductor(GroundPlaneLength=0.046)`

Example: double

`GroundPlaneWidth` — Width of ground plane
`0.0056` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar. This object does not support infinite ground plane width.

Example: `inductor = spiralInductor(GroundPlaneWidth=0.046)`

Example: double

`Substrate` — Type of dielectric material
`dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The default value is a `dielectric` object with these properties:

- `Name—{'RTDuroid','RTDuroid','RTDuroid'}`
- `EpsilonR—[3.66,3.66,3.66]`
- `LossTangent—[0.0013,0.0013,0.0013]`
- `Thickness—[0.508e-3,0.508e-3,0.508e-3]`

Example: `d = dielectric("FR4"); inductor = spiralInductor(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'Copper'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("PEC"); inductor = spiralInductor(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
inductance	Calculate inductance
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

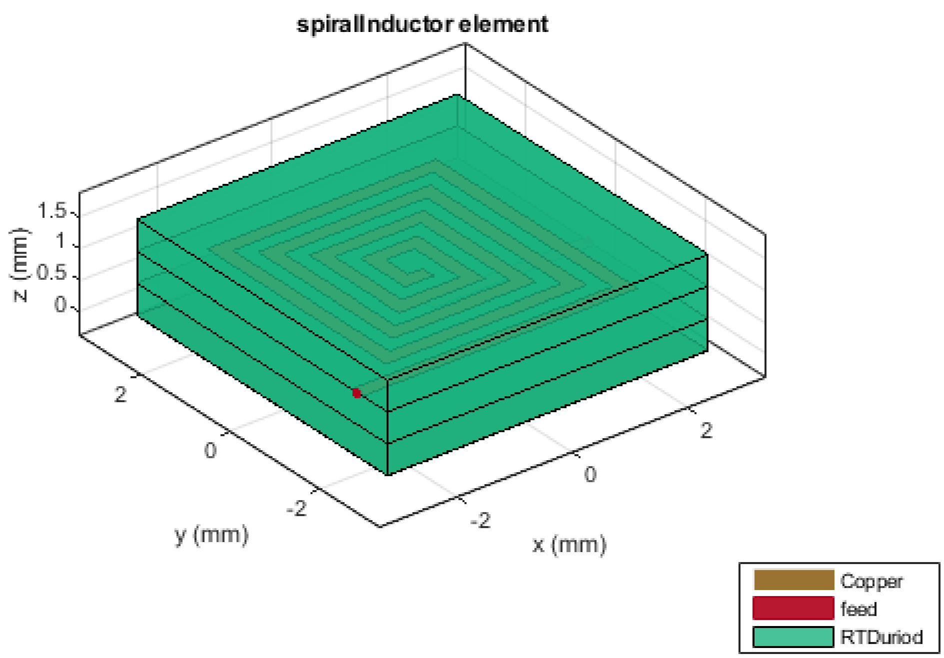
Create Default Spiral Inductor

Create and view a default spiral inductor.

```
inductor = spiralInductor
```

```
inductor = 
  spiralInductor with properties:

          SpiralShape: 'Square'
        InnerDiameter: 5.0000e-04
                Width: 2.5000e-04
              Spacing: 2.5000e-04
             NumTurns: 4
               Height: 0.0010
    GroundPlaneLength: 0.0056
     GroundPlaneWidth: 0.0056
            Substrate: [1x1 dielectric]
            Conductor: [1x1 metal]
```

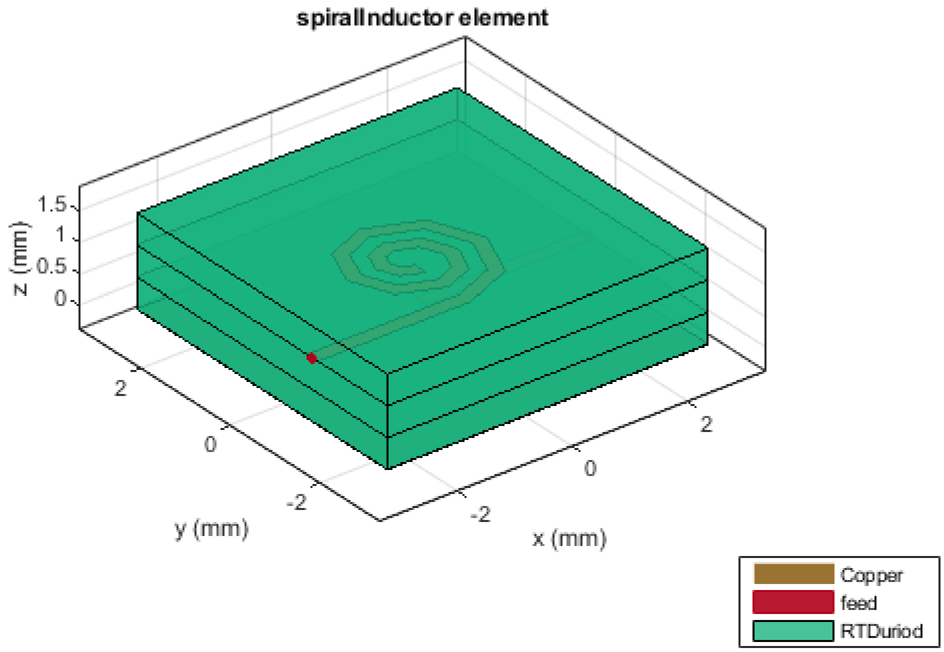
```
show(inductor)
```



Octagonal Spiral Inductor

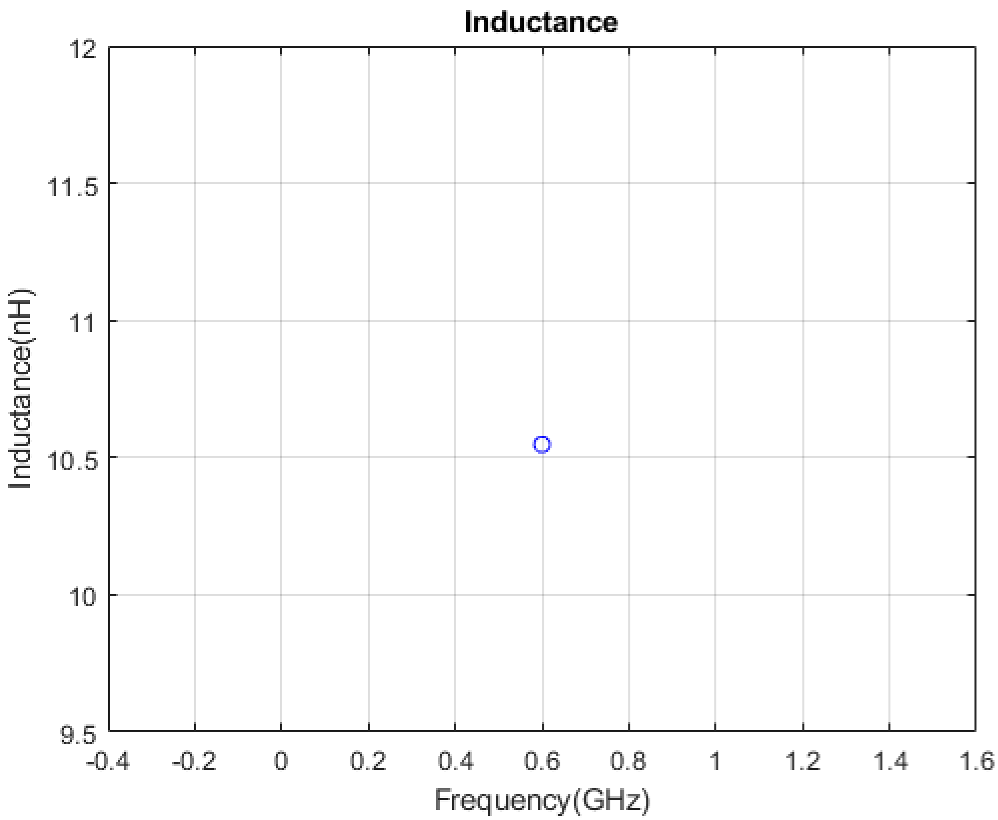
Create and view a two-turn octagonal spiral inductor. Enable edge feed for this inductor.

```
inductor = spiralInductor(SpiralShape="Octagon",NumTurns=2);
show(inductor)
```



Measure the inductance of the inductor.

```
figure
inductance(inductor,600e6)
```



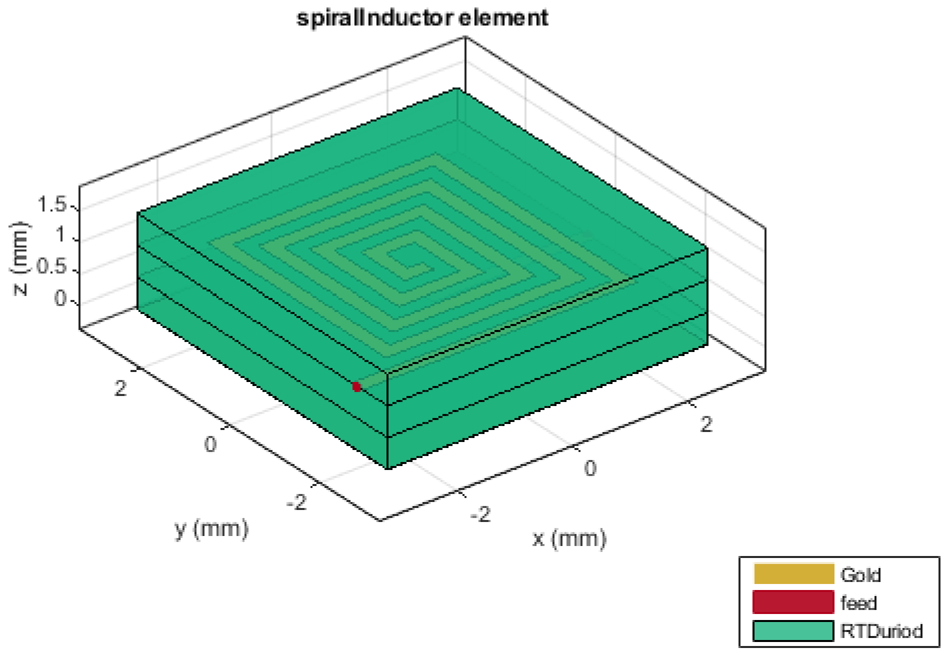

```
ind = inductance(inductor,600e6)
```

```
ind = 1.0548e-08
```

Analyze Spiral Inductor Using Behavioral S-parameters

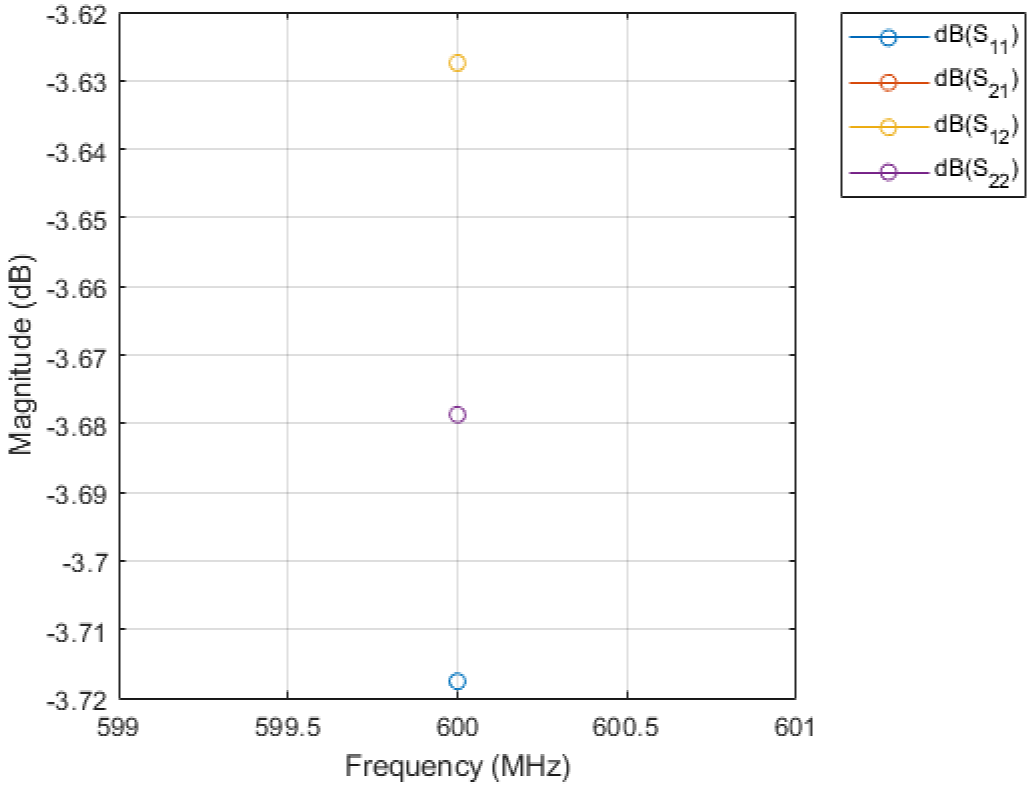
Create a spiral inductor using gold as the conductor.

```
inductor = spiralInductor;
inductor.Conductor = metal("Gold");
show(inductor)
```



Compute and plot the behavioral S-parameters of the inductor at 600 MHz.

```
spar = sparameters(inductor,600e6,Behavioral=true);
rfplot(spar)
```



More About

Parametric Analysis Guidelines

- The inductance of a spiral inductor is directly proportional to the physical parameters such as NumTurns, Spacing, and Width.
- The decrease in the width reduces the capacitance between the turns of the inductor.

Inductor Length and Area

Inductor length equation:

$$inductorlength = NumTurns * Davg * N * \tan\left(\frac{\pi}{N}\right)$$

where,

- N = Number of sides of the polygon
- $D_{avg} = \frac{(d_{out} - d_{in})}{(d_{out} + d_{in})}$

References

[1] Beeresha, R.S., A.M. Khan, and H.V. Manjunatha Reddy. “The Design and EM-Simulation of Square Spiral Inductor Using Simple Equations.” *Materials Today: Proceedings* 5, no. 4 (2018): 10875–82. https://doi.org/10.1016/j.matpr.2018.05.074.

[2] Mohan, S.S., M. del Mar Hershenson, S.P. Boyd, and T.H. Lee. “Simple Accurate Expressions for Planar Spiral Inductances.” *IEEE Journal of Solid-State Circuits* 34, no. 10 (October 1999): 1419–24. https://doi.org/10.1109/4.792620.

See Also

`interdigitalCapacitor`

Introduced in R2021b

stripLine

Create transmission line in strip form

Description

Use the `stripLine` object to create a transmission line in the strip form. Striplines are high speed transmission lines used for routing various RF components in the inner layers of a multi-layer printer circuit board (PCB). The structure consists of a transmission line trace surrounded by dielectric material suspended between two ground planes.

For the electromagnetic (EM) interactions between the ground planes, the feed is located in the middle of the strip and a via connects the top of the ground plane to the strip line.

Types and applications of stripline routing techniques:

- `Symmetric or Plain` — Transmission lines routed on the internal layers
- `Asymmetric` — Transmission lines not centrally embedded in the ground plane. Asymmetric striplines are placed closed to one of the ground planes. When routing signals the closer ground plane is used for reference for the stripline to ensure stronger return signal..
- `Edge-Coupled` — Routing technique used for differential pairs. It has the same structure as the symmetric stripline routing but with trace spacing for the differential pair.
- `Broadside-Coupled` — Routing technique used for differential pairs but stacked one on top of the other.
- `Suspended` — Printed stripline with a strip conductor centered between two parallel ground planes, placed on a dielectric surface. The substrate is suspended in a metal closer. The major portion of the EM field is confined to the air gaps between dielectric substrates and ground planes

Creation

Syntax

```
strip = stripLine
strip = stripLine('propety1',value1,'property2',value2...)
```

Description

`strip = stripLine` creates a default strip transmission line. The default dimensions are for a frequency range of 1 GHz to 5 GHz. The default transmission line lies in the X-Y plane.

`strip = stripLine('propety1',value1,'property2',value2...)` sets properties using one or more name value pair arguments. For example, `strip = stripLine('Length',0.0300)` creates a strip transmission line of length 0.0300 meters. Properties not specified retain their default values.

Properties

`Length` — Length of strip line
`0.0202` (default) | positive scalar

Length of the strip line in meters, specified as a positive scalar.

Example: `strip = stripLine('Length',0.0300)`

Data Types: `double`

`Width` — Width of strip line
`0.0027` (default) | positive scalar

Width of the strip line in meters, specified as a positive scalar.

Example: `strip = stripLine('Width',0.0037)`

Data Types: `double`

`Height` — Height of strip line
`0.0016` (default) | positive scalar

Height of the strip line in meters, specified as a positive scalar.

Example: `strip = stripLine('Height',0.0026)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane
`0.0150` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `strip = stripLine('GroundPlaneWidth',0.0350)`

Data Types: `double`

`Substrate` — Type of dielectric material
`'RT-Duriod'` (default) | `dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The values of the default dielectric material are: `('Name',{'RT-Duriod'},'EpsilonR',2.2, 'LossTangent',0.001,'Thickness',0.0032)` For more information see `dielectric`.

Example: `d = dielectric('FR4'); strip = stripLine('Substrate',d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used for conducting layers
`'PEC'` (default) | `metal` object

Type of metal used for conducting layers, specified as a `metal` object. For more information see `metal`.

Example: `m = metal('PEC'); strip = stripLine('Conductor',m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design symmetric strip transmission line around given frequency
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
mesh	Change and view mesh properties of metal or dielectric in PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

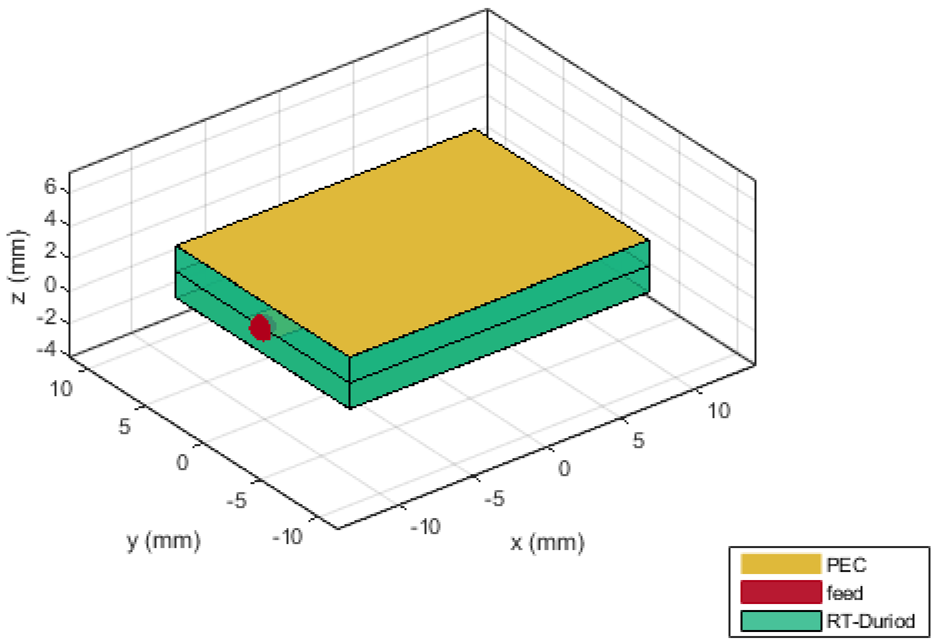
Default Strip Line Transmission Line

Create and view a transmission line in the form of a strip.

```
txstrip = stripLine
```

```
txstrip = 
  stripLine with properties:

              Length: 0.0202
               Width: 0.0027
              Height: 0.0016
    GroundPlaneWidth: 0.0150
           Substrate: [1×1 dielectric]
           Conductor: [1×1 metal]
```

```
show(txstrip)
```



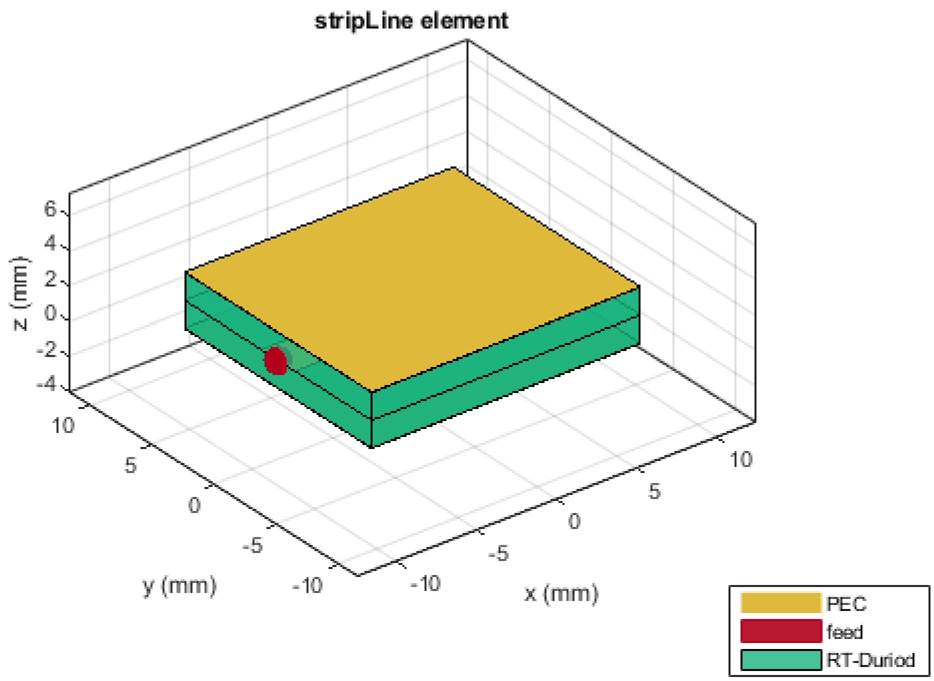
Plot S-parameters of Stripline Transmission Line

Create a stripline transmission line of length 16.84 mm.

```
sline = stripLine('Length',16.84e-3);
```

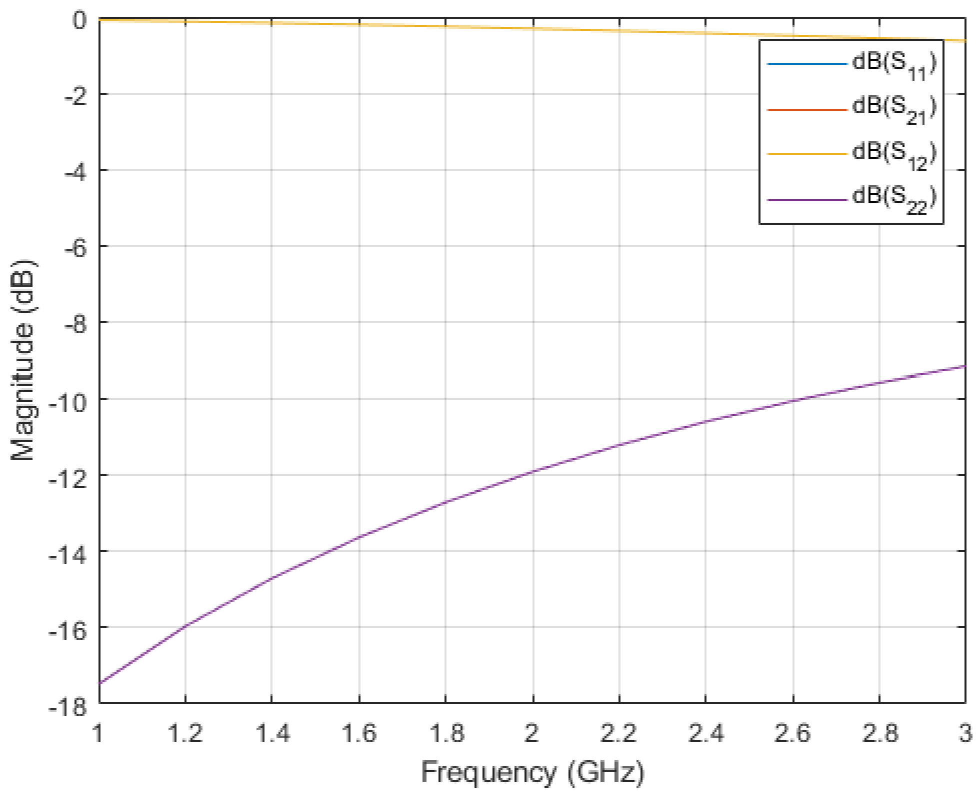
View the transmission line.

```
show(sline)
```



Plot the s-parameters of the transmission line.

```
spar = sparameters(sline,linspace(1e9,3e9,11));
rfplot(spar);
```



More About

Parametric Analysis

- Decrease the dimensions of the line such as length and width to increase the operating frequency.
- Vary the width of the line to control the characteristic impedance of the line.
- `FeedDiameter` cannot be increased more than the width of the strip.

See Also

`coplanarWaveguide` | `coupledMicrostripLine` | `coupledStripLine` | `microstripLine`

Introduced in R2021b

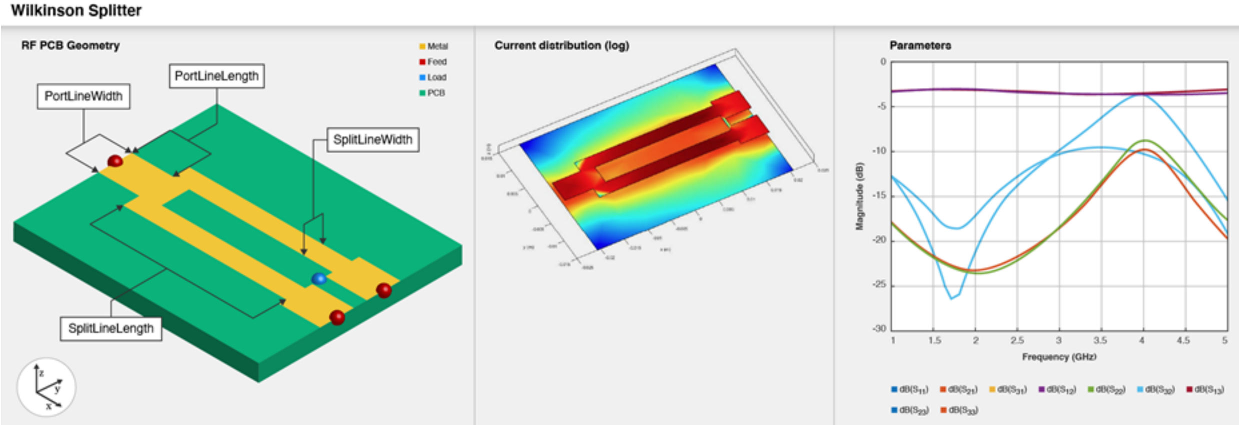
wilkinsonSplitter

Create Wilkinson splitter in microstrip form on X-Y plane

Description

Use the `wilkinsonSplitter` object to create a Wilkinson power splitter in microstrip form on the X-Y plane. The Wilkinson power splitter is the most common type of power divider. It is a lossless power divider and provides matching at all ports. The isolation between the output ports is achieved using a `2*Z0` resistor connected between the output ports. The Wilkinson splitter is used in transmitters, receivers, power combining applications, and in devices measuring the power of a test signal.

Wilkinson Splitter



Creation

Syntax

```
splitter = wilkinsonSplitter
splitter = wilkinsonSplitter(Name=Value)
```

Description

`splitter = wilkinsonSplitter` creates a Wilkinson splitter with a Teflon substrate. The default property values are for an operating frequency of 1.8 GHz.

`splitter = wilkinsonSplitter(Name=Value)` sets “Properties” on page 1-33 using one or more name-value arguments. For example, `wilkinsonSplitter(PortLineLength=0.0300)` creates a Wilkinson splitter with an input and output line length of 0.0300 meters. Properties not specified retain their default values.

Properties

Shape — Shape of Wilkinson splitter
`"Rectangular"` (default) | `"Circular"`

Shape of the Wilkinson splitter, specified as `"Rectangular"` or `"Circular"`.

Example: `splitter = wilkinsonSplitter(Shape="Circular")`

Data Types: `char` | `string`

`PortLineLength` — Length of input and output line
`0.0060` (default) | positive scalar

Length of the input and the output line in meters, specified as a positive scalar.

Example: `splitter = wilkinsonSplitter(PortLineLength=0.0070)`

Data Types: `double`

`PortLineWidth` — Width of input and output line
`0.0049` (default) | positive scalar

Width of the input and the output line in meters, specified as a positive scalar.

Example: `splitter = wilkinsonSplitter(PortLineWidth=0.0070)`

Data Types: `double`

`SplitLineLength` — Length of 70-ohm line
`0.0300` (default) | positive scalar

Length of the 70-ohm line in meters, specified as a positive scalar. The typical length of a Wilkinson splitter is $\lambda/4$.

Example: `splitter = wilkinsonSplitter(SplitLineLength=0.0570)`

Data Types: `double`

`SplitLineWidth` — Width of 70-ohm line
`0.0028` (default) | positive scalar

Width of the 70-ohm line in meters, specified as a positive scalar.

Example: `splitter = wilkinsonSplitter(SplitLineWidth=0.00780)`

Data Types: `double`

`ResistorLength` — Length of resistor
`0.0020` (default) | positive scalar

Length of the resistor in meters, specified as a positive scalar. The resistor length determines the distance between the output ports.

Example: `splitter = wilkinsonSplitter(ResistorLength=0.0050)`

Data Types: `double`

`Resistance` — Resistance value
`100` (default) | positive scalar

Resistance value in ohms, specified as a positive scalar. The default value is for an equal-split Wilkinson splitter.

Example: `splitter = wilkinsonSplitter(Resistance=50)`

Data Types: `double`

`Height` — Height of Wilkinson splitter from ground plane
`0.0016` (default) | positive scalar

Height of the Wilkinson splitter from the ground plane in meters, specified as a positive scalar. In the case of a multilayer substrate, you can use the `Height` property to create a Wilkinson splitter where the two dielectrics interface.

Example: `splitter = wilkinsonSplitter(Height=0.0076)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane in meters
`0.0300` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `splitter = wilkinsonSplitter(GroundPlaneWidth=0.046)`

Example: double

`Substrate` — Type of dielectric material
`'Teflon'` (default) | `dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The thickness of the default dielectric material Teflon is `0.0016` m or the same value as the `Height` property.

Example: `d = dielectric("FR4")`; `splitter = wilkinsonSplitter(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'Copper'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("PEC")`; `splitter = wilkinsonSplitter(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design Wilkinson splitter around specified frequency
feedCurrent	Calculate current at feed port
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

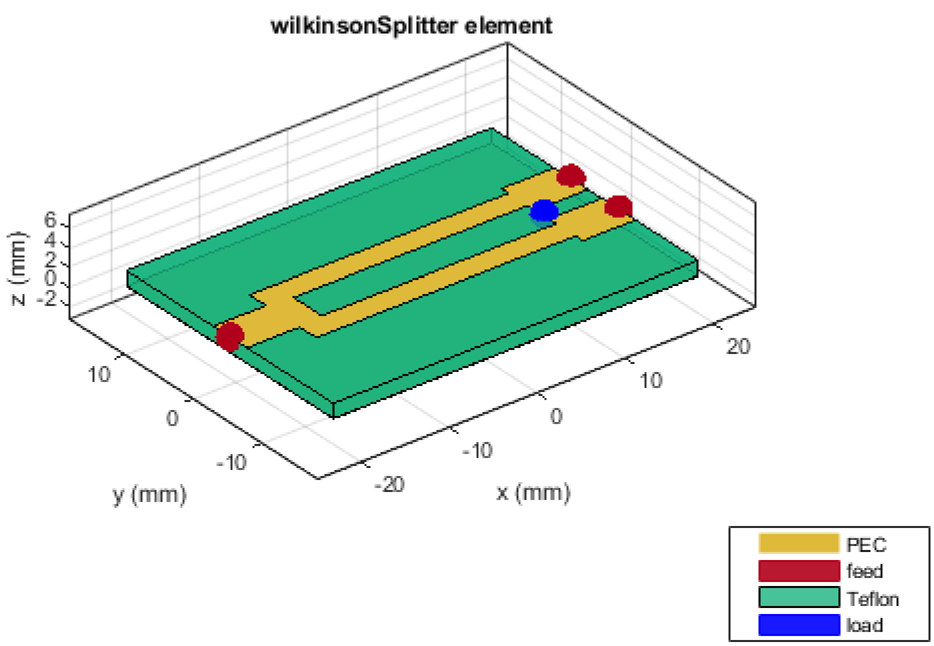
Create Default Wilkinson Splitter

Create and view a default Wilkinson splitter on the X-Y plane.

```
splitter = wilkinsonSplitter
```

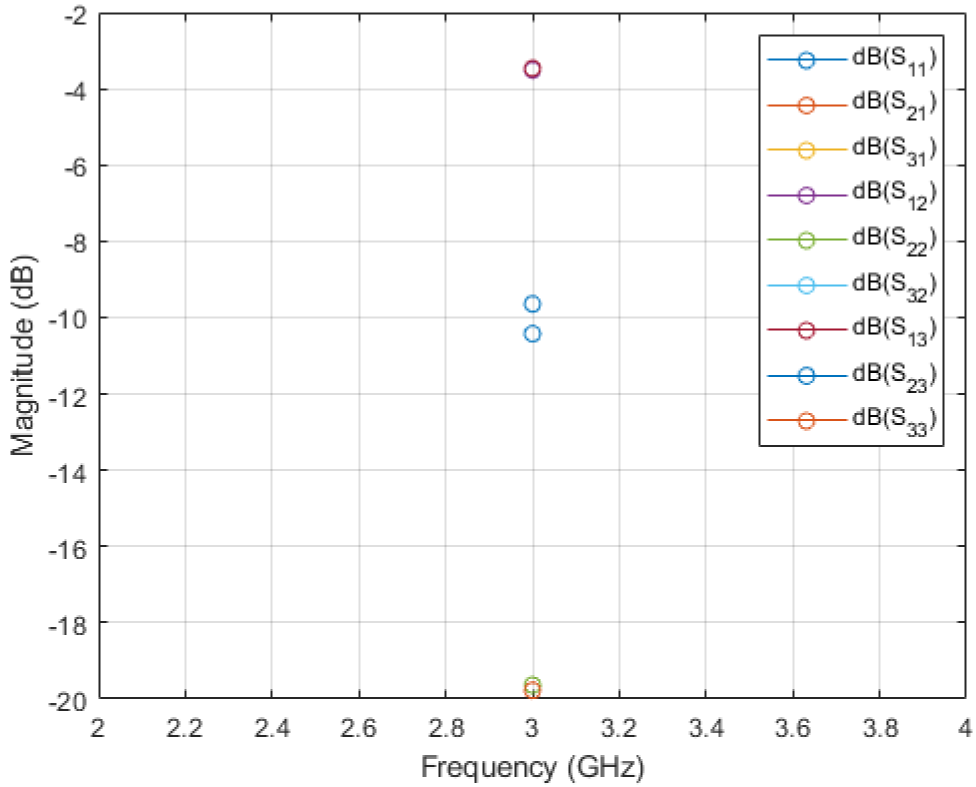
```
splitter = 
  wilkinsonSplitter with properties:

              Shape: 'Rectangular'
     PortLineLength: 0.0060
      PortLineWidth: 0.0049
    SplitLineLength: 0.0300
     SplitLineWidth: 0.0028
     ResistorLength: 0.0020
         Resistance: 100
             Height: 0.0016
   GroundPlaneWidth: 0.0300
          Substrate: [1x1 dielectric]
          Conductor: [1x1 metal]
```

```
show(splitter)
```



Calculate and plot the S-parameters of the splitter at 3 GHz.

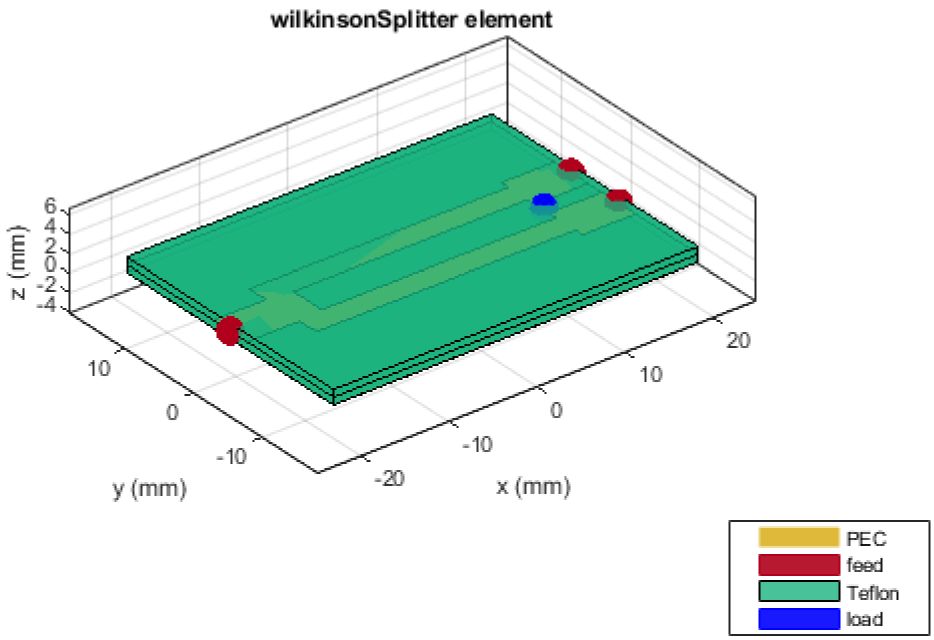
```
spar=sparameters(splitter,3e9);
figure
rfplot(spar);
```



Create Multilayer Wilkinson Splitter

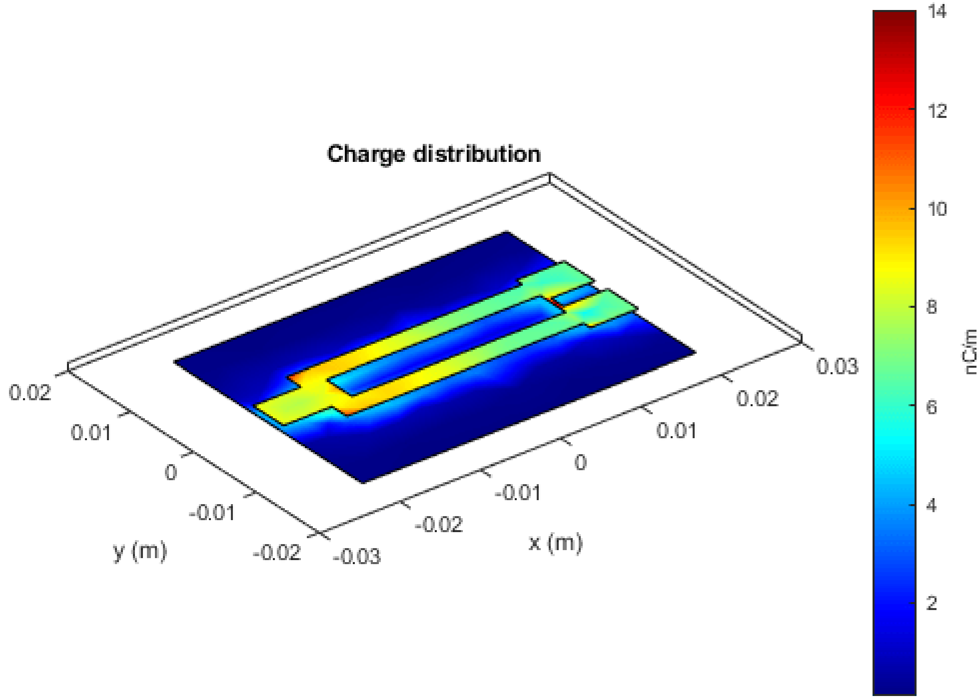
Create and view a multilayer Wilkinson splitter.

```
sub =  dielectric(Name=["Teflon","Teflon"],EpsilonR=[2.1 2.1], ...
    LossTangent=[0 0],Thickness=[0.8e-3 0.8e-3]);
splitter = wilkinsonSplitter(Height=0.8e-3,Substrate=sub);
show(splitter)
```

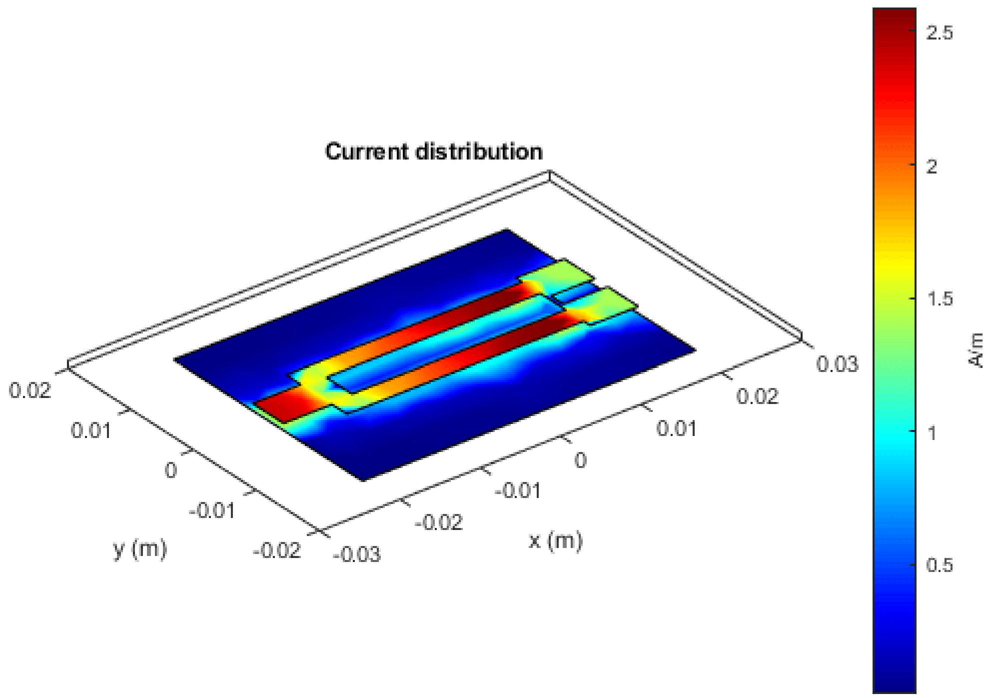


Plot the charge and current on this splitter at 3 GHz.

```
figure
charge(splitter,3e9)
```




```
figure
current(splitter,3e9)
```



References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

See Also

`wilkinsonSplitterUnequal`

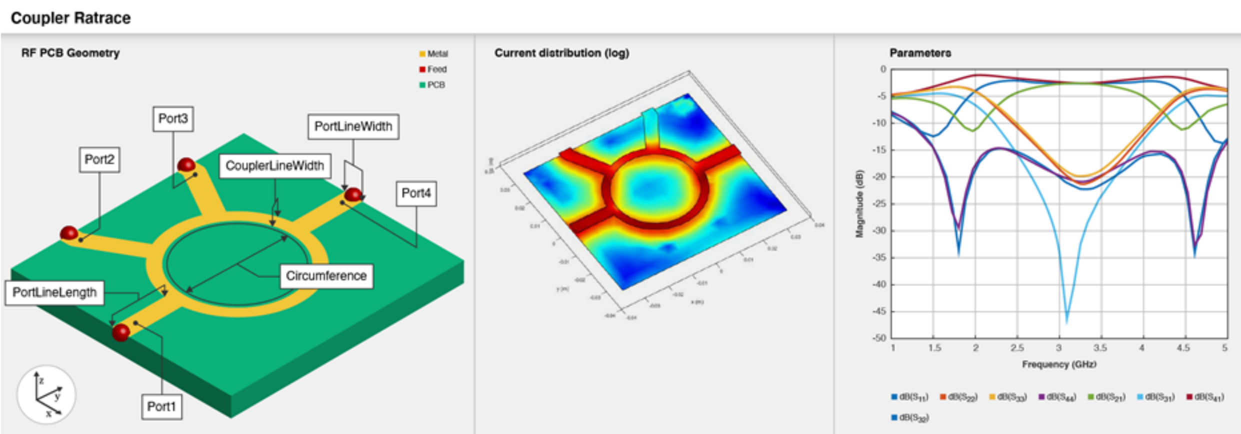
Introduced in R2021b

couplerRatrace

Create equal-split rat-race coupler or 180-degree-ring hybrid

Description

Use the `couplerRatrace` object to create an equal-split rat-race coupler or a 180-degree-ring hybrid.



The rat-race coupler is used as a splitter with a phase shift. When given two inputs, the coupler can create sum and difference ports for added and subtracted power. It also acts as an interface between transmitters and receivers for integrating with an antenna and in building circuits with complex functionality like a comparator with sum and difference ports.

There are four ports and the circumference is 1.5 λ. The phase shift between the output ports is 180 degrees. When you apply an input at port 1, port 2 and port 4 are coupled ports, where the output has a phase difference of 180 degrees, and port 3 is the isolated port. When you apply an input at port 3, the output is split equally with same phase at port 2 and port 4.

Creation

Syntax

```
coupler = couplerRatrace
coupler = couplerRatrace(Name=Value)
```

Description

`coupler = couplerRatrace` creates a rat-race coupler. The default property values are for a frequency of 3 GHz.

`coupler = couplerRatrace(Name=Value)` sets “Properties” on page 1-42 using one or more name-value arguments. For example, `couplerRatrace(PortLineLength=0.0286)` creates a rat-race coupler with a port line length of 0.0286 meters. Properties not specified retain their default values.

Properties

`PortLineLength` — Length of input and output line
`0.0186` (default) | positive scalar

Length of the input and the output line in meters, specified as a positive scalar.

Example: `coupler = couplerRatrace(PortLineLength=0.0286)`

Data Types: `double`

`PortLineWidth` — Width of input and output line
`0.0050` (default) | positive scalar

Width of the input and the output line in meters, specified as a positive scalar.

Example: `coupler = couplerRatrace(PortLineWidth=0.0070)`

Data Types: `double`

`CouplerLineWidth` — Width of coupler line
`0.0030` (default) | positive scalar

Width of the coupler line in meters, specified as a positive scalar. The default value is for a `λ/4` line with an impedance of `Zo/sqrt(2)` ohms.

Example: `coupler = couplerRatrace(CouplerLineWidth=0.0070)`

Data Types: `double`

`Circumference` — Length of coupler line
`0.1110` (default) | positive scalar

Length of the coupler line in meters, specified as a positive scalar. The default value is for a 1.5 λ line with an impedance of `Zo/sqrt(2)` ohms.

Example: `coupler = couplerRatrace(Circumference=0.2303)`

Data Types: `double`

`Height` — Height of rat-race coupler from ground plane
`0.0016` (default) | positive scalar

Height of the rat-race coupler from the ground plane in meters, specified as a positive scalar.

In the case of a multilayer substrate, you can use the `Height` property to create a rat-race coupler where the two dielectrics interface.

Example: `coupler = couplerRatrace(Height=0.0015)`

Data Types: `double`

`Substrate` — Type of dielectric material
`'Teflon'` (default) | `dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The thickness of the default dielectric material Teflon is `0.0016` m or the same as the `Height` property.

Example: `d = dielectric("FR4"); coupler = couplerRatrace(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'Copper'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("PEC"); coupler = couplerRatrace(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design rat-race coupler around specified frequency
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

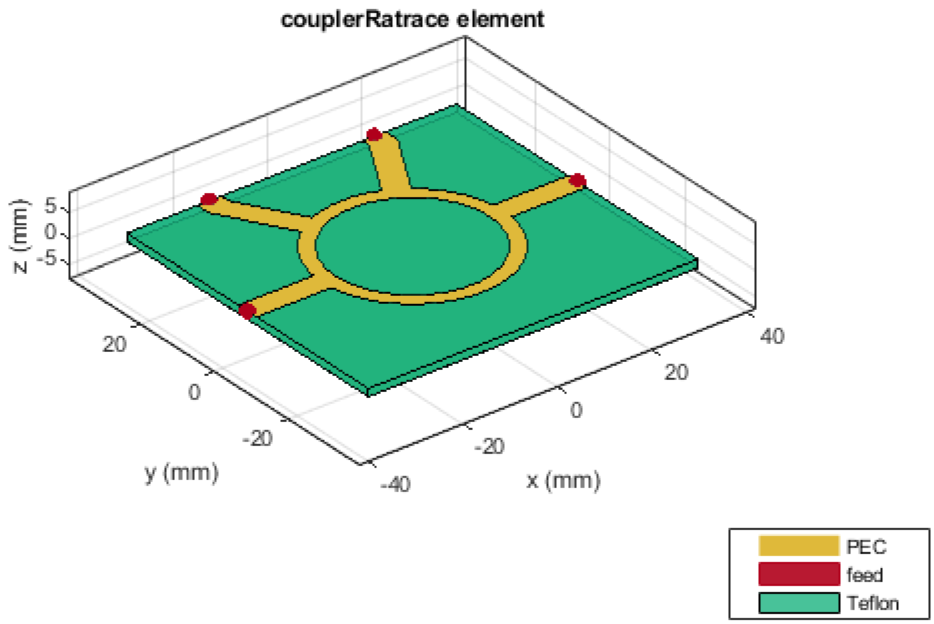
Create Default Rat-Race Coupler

Create and view a default rat-race coupler.

```
ratrace = couplerRatrace
```

```
ratrace = 
  couplerRatrace with properties:

      PortLineLength: 0.0186
       PortLineWidth: 0.0050
    CouplerLineWidth: 0.0030
       Circumference: 0.1110
              Height: 0.0016
           Substrate: [1x1 dielectric]
           Conductor: [1x1 metal]
```

```
show(ratrace)
```



Calculate Current Distribution on Rat-Race Coupler

Create a rat-race coupler with default properties.

```
coupler = couplerRatrace;
```

Set the excitation voltage and the phase angle at the ports of the coupler.

```
v = voltagePort(4)
```

```
v = 
  voltagePort with properties:

         NumPorts: 4
      FeedVoltage: [1 0 0 0]
        FeedPhase: [0 0 0 0]
    PortImpedance: 50
```

```
v.FeedVoltage = [1 0 1 0]
```

```
v = 
  voltagePort with properties:

         NumPorts: 4
```

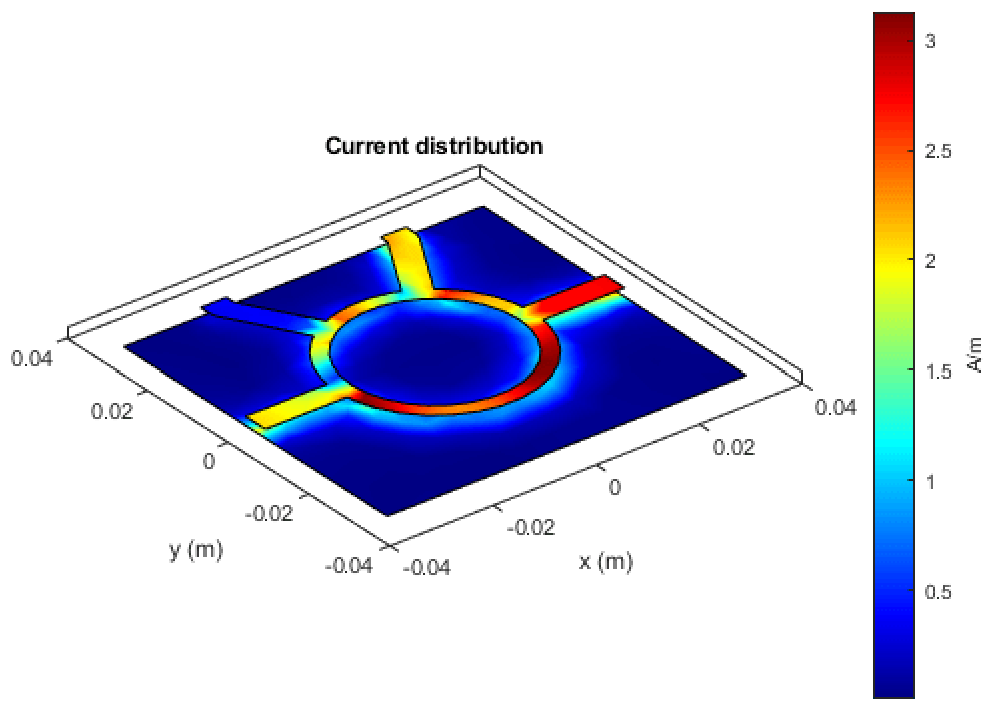
```
      FeedVoltage: [1 0 1 0]
        FeedPhase: [0 0 0 0]
    PortImpedance: 50
```

```
v.FeedPhase = [90 0 270 0]
```

```
v = 
  voltagePort with properties:

         NumPorts: 4
      FeedVoltage: [1 0 1 0]
        FeedPhase: [90 0 270 0]
    PortImpedance: 50
```

Calculate and plot the current on the coupler at 3 GHz.

```
figure
current(coupler,3e9,Excitation=v)
```



References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

See Also
`couplerBranchline`

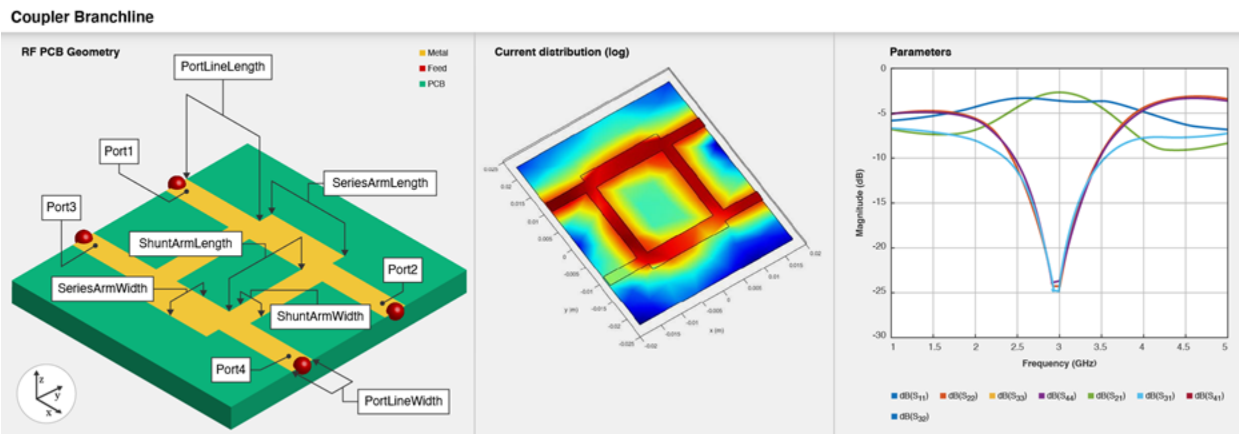
Introduced in R2021b

couplerBranchline

Create branch line coupler or quadrature hybrid

Description

Use the `couplerBranchline` object to create a branch line coupler or a quadrature hybrid. A branch line coupler or a quadrature hybrid divides the power between two ports with a phase difference of 90 degrees. This PCB component has four ports. By default, port 1 is the input port, port 2 is the through port, port 4 is the coupled port, and port 3 is the isolated port.



Creation

Syntax

```
coupler = couplerBranchline
coupler = couplerBranchline(Name=Value)
```

Description

`coupler = couplerBranchline` creates a branch line coupler. The default property values are for the frequency of 3 GHz.

`coupler = couplerBranchline(Name=Value)` sets “Properties” on page 1-47 using one or more name-value arguments. For example, `couplerBranchline(PortLineLength=0.0286)` creates a branch line coupler of length 0.0286 meters. Properties not specified retain their default values.

Properties

`PortLineLength` — Length of input and output line

`0.0186` (default) | positive scalar

Length of the input and the output line in meters, specified as a positive scalar.

Example: `coupler = couplerBranchline(PortLineLength=0.0286)`

Data Types: `double`

`PortLineWidth` — Width of input and output line
`0.0051` (default) | positive scalar

Width of the input and the output line in meters, specified as a positive scalar.

Example: `coupler = couplerBranchline(PortLineWidth=0.0070)`

Data Types: `double`

`SeriesArmLength` — Length of series arm
`0.0184` (default) | positive scalar

Length of the series arm in meters, specified as a positive scalar.

Example: `coupler = couplerBranchline(SeriesArmLength=0.0286)`

Data Types: `double`

`SeriesArmWidth` — Width of series arm
`0.0083` (default) | positive scalar

Width of the series arm in meters, specified as a positive scalar.

Example: `coupler = couplerBranchline(SeriesArmWidth=0.0096)`

Data Types: `double`

`ShuntArmLength` — Length of shunt arm
`0.0186` (default) | positive scalar

Length of the shunt arm in meters, specified as a positive scalar.

Example: `coupler = couplerBranchline(ShuntArmLength=0.0286)`

Data Types: `double`

`ShuntArmWidth` — Width of shunt arm
`0.0051` (default) | positive scalar

Width of the shunt arm in meters, specified as a positive scalar.

Example: `coupler = couplerBranchline(ShuntArmWidth=0.0096)`

Data Types: `double`

`Height` — Height of branch line coupler from ground plane
`0.0016` (default) | positive scalar

Height of the branch line coupler from the ground plane in meters, specified as a positive scalar.

In the case of a multilayer substrate, you can use the `Height` property to create a branch line coupler line where the two dielectrics interface.

Example: `coupler = couplerBranchline(Height=0.0076)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane
`0.0600` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `coupler = couplerBranchline(GroundPlaneWidth=0.046)`

Example: double

`Substrate` — Type of dielectric material
`'RTDuriod'` (default) | `dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object.

Example: `d = dielectric("FR4"); coupler = couplerBranchline(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'Copper'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("PEC"); coupler = couplerBranchline(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design branchline coupler around particular frequency
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

Create Default Branchline Coupler

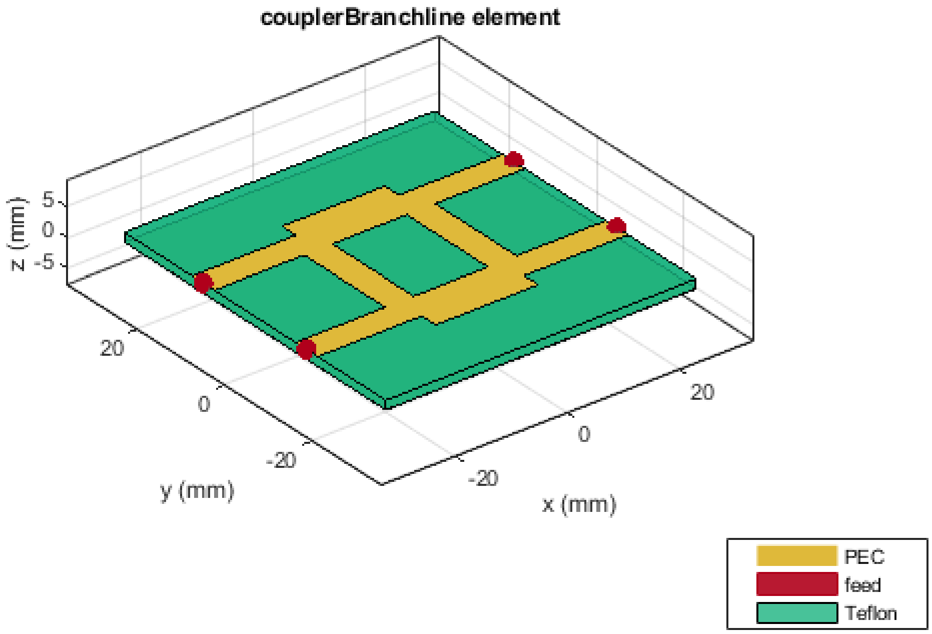
Create and view a default branchline coupler.

```
coupler = couplerBranchline
```

```
coupler = 
  couplerBranchline with properties:

      PortLineLength: 0.0186
       PortLineWidth: 0.0051
     SeriesArmLength: 0.0184
      SeriesArmWidth: 0.0083
      ShuntArmLength: 0.0186
       ShuntArmWidth: 0.0051
```

```
            Height: 0.0016
  GroundPlaneWidth: 0.0600
         Substrate: [1x1 dielectric]
         Conductor: [1x1 metal]
```

```
show(coupler)
```



References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

See Also

`couplerRatrace`

Introduced in R2021b

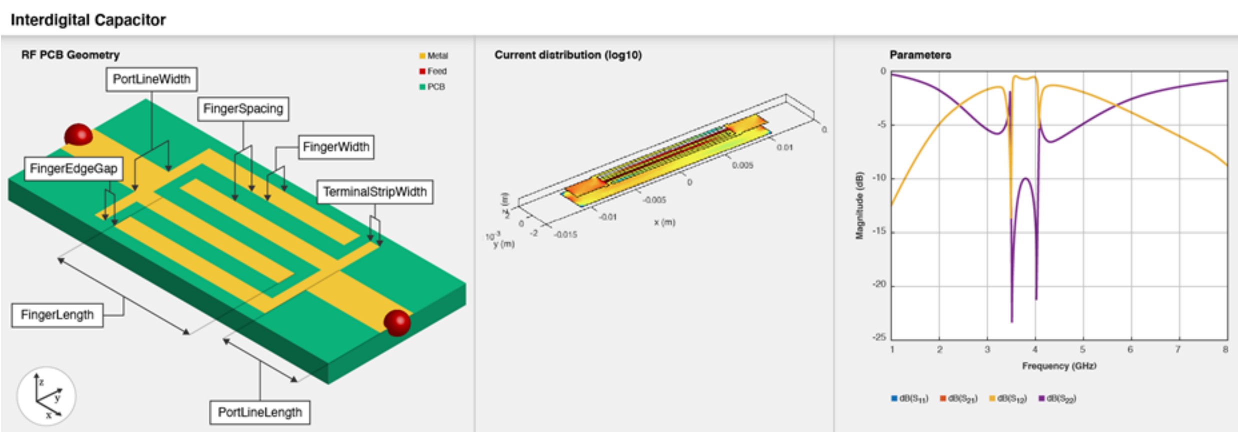
interdigitalCapacitor

Create basic interdigital capacitor

Description

Use the `interdigitalCapacitor` object to create an interdigital planar capacitor (IDC). IDCs are used in high frequency applications such as:

- Receiver circuits where antenna radiators are connected to RF
- Wireless data communications with RFID
- Humidity and solution concentration measurements
- Lab-on-chip devices (LOCs)



A two-port series IDC with microstrip form feeder lines supports single and multiple dielectrics. It is a coplanar structure consisting of multiple comb electrodes or intersecting fingers with spaces between the fingers. An IDC can have identical port line lengths and widths on either sides.

To analyze the behavioral model for an interdigital capacitor, set the `Behavioral` property in the `sparameters` function to `true or 1`.

Creation

Syntax

```
capacitor = interdigitalCapacitor
capacitor = interdigitalCapacitor(Name=Value)
```

Description

`capacitor = interdigitalCapacitor` creates a basic interdigital capacitor. The default property values are for an operating bandwidth of 3.6-4 GHz.

`capacitor = interdigitalCapacitor(Name=Value)` sets “Properties” on page 1-52 using one or more name-value arguments. For example, `interdigitalCapacitor(NumFingers=10)` creates an interdigital capacitor with 10 fingers. Properties not specified retain their default values.

Properties

`NumFingers` — Number of fingers on capacitor
`4` (default) | positive scalar

Number of fingers on the capacitor, specified as a positive scalar.

Example: `capacitor = interdigitalCapacitor(NumFingers=10)`

Data Types: `double`

`FingerLength` — Length of overlapping fingers
`0.0137` (default) | positive scalar

Length of the overlapping fingers in meters, specified as a positive scalar.

Example: `capacitor = interdigitalCapacitor(FingerLength=0.0217)`

Data Types: `double`

`FingerWidth` — Width of overlapping fingers
`3.1600e-04` (default) | positive scalar

Width of the overlapping fingers in meters, specified as a positive scalar.

Example: `capacitor = interdigitalCapacitor(FingerWidth=4.8000e-04)`

Data Types: `double`

`FingerSpacing` — Distance between fingers
`3.0000e-04` (default) | positive scalar

Distance between the fingers in meters, specified as a positive scalar.

Example: `capacitor = interdigitalCapacitor(FingerSpacing=2.9000e-04)`

Data Types: `double`

`FingerEdgeGap` — Gap between edges of fingers
`3.4100e-04` (default) | positive scalar

Gap between the edges of the fingers in meters, specified as a positive scalar.

Example: `capacitor = interdigitalCapacitor(FingerEdgeGap=2.05000e-04)`

Data Types: `double`

`TerminalStripWidth` — Width of terminals
`5.0000e-04` (default) | positive scalar

Width of the terminals in meters, specified as a positive scalar.

Example: `capacitor = interdigitalCapacitor(TerminalStripWidth=4.9000e-04)`

Data Types: `double`

`PortLineWidth` — Width of ports
`0.0019` (default) | positive scalar

Width of the ports in meters, specified as a positive scalar.

Example: `capacitor = interdigitalCapacitor(PortLineWidth=0.0020)`

Data Types: `double`

`PortLineLength` — Length of ports
`0.0030` (default) | positive scalar

Length of the ports in meters, specified as a positive scalar.

Example: `capacitor = interdigitalCapacitor(PortLineLength=0.0040)`

Data Types: `double`

`Height` — Height from ground plane to capacitor
`7.8700e-04` (default) | positive scalar

Height from the capacitor to the ground plane in meters, specified as a positive scalar.

Example: `capacitor = interdigitalCapacitor(Height=6.9000e-04)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane
`0.0030` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `capacitor = interdigitalCapacitor(GroundPlaneWidth=0.0040)`

Example: double

`Substrate` — Type of dielectric material
`dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The default value is a `dielectric` object with these properties:

- `Name—{'Roger'}`
- `EpsilonR—3.2`
- `LossTangent—0.0002`
- `Thickness—0.000787`

Example: `d = dielectric("FR4"); capacitor = interdigitalCapacitor(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'Coppper'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("PEC"); capacitor =interdigitalCapacitor(Conductor=m)`

Data Types: `string` | `char`

Object Functions

capacitance	Calculate capacitance
charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

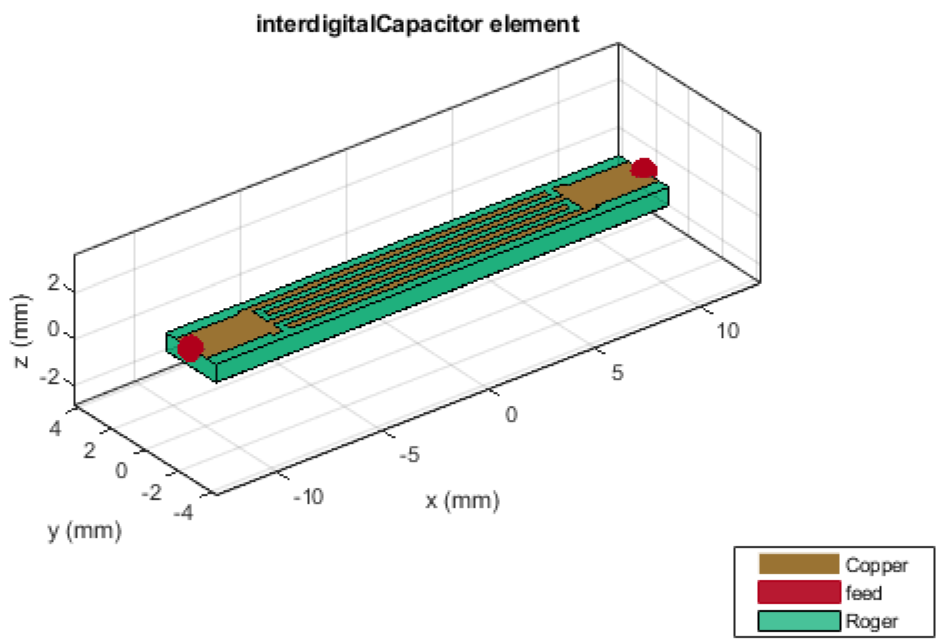
Create Default Interdigital Capacitor

Create and view a default interdigital capacitor.

```
idcapacitor = interdigitalCapacitor
```

```
idcapacitor = 
  interdigitalCapacitor with properties:

            NumFingers: 4
          FingerLength: 0.0137
           FingerWidth: 3.1600e-04
         FingerSpacing: 3.0000e-04
          FingerEdgeGap: 3.4100e-04
    TerminalStripWidth: 5.0000e-04
         PortLineWidth: 0.0019
        PortLineLength: 0.0030
                Height: 7.8700e-04
      GroundPlaneWidth: 0.0030
             Substrate: [1x1 dielectric]
             Conductor: [1x1 metal]
```

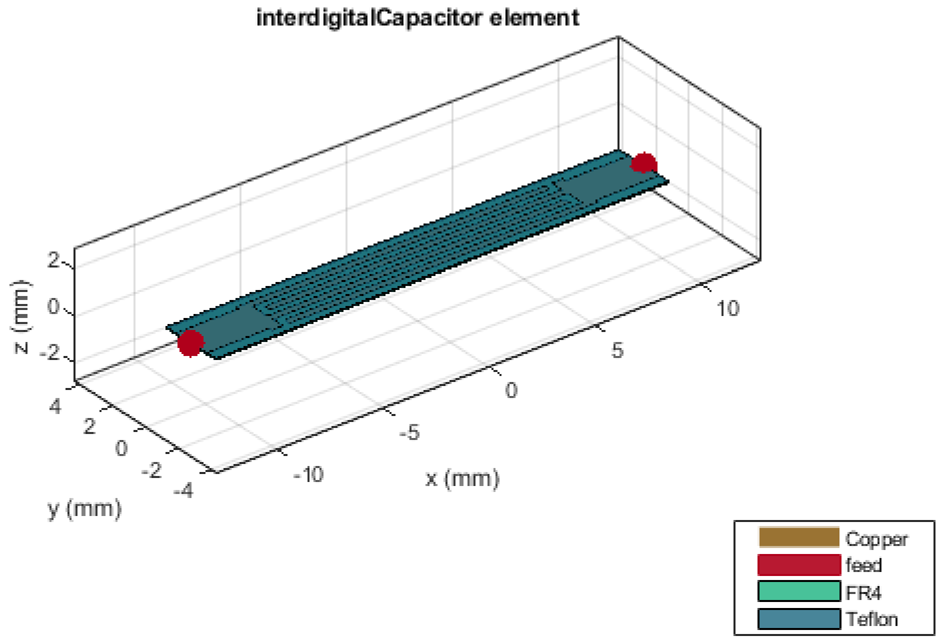
```
show(idcapacitor)
```



Multilayer Interdigital Capacitor

Create and view a multilayer interdigital capacitor with two different dielectrics.

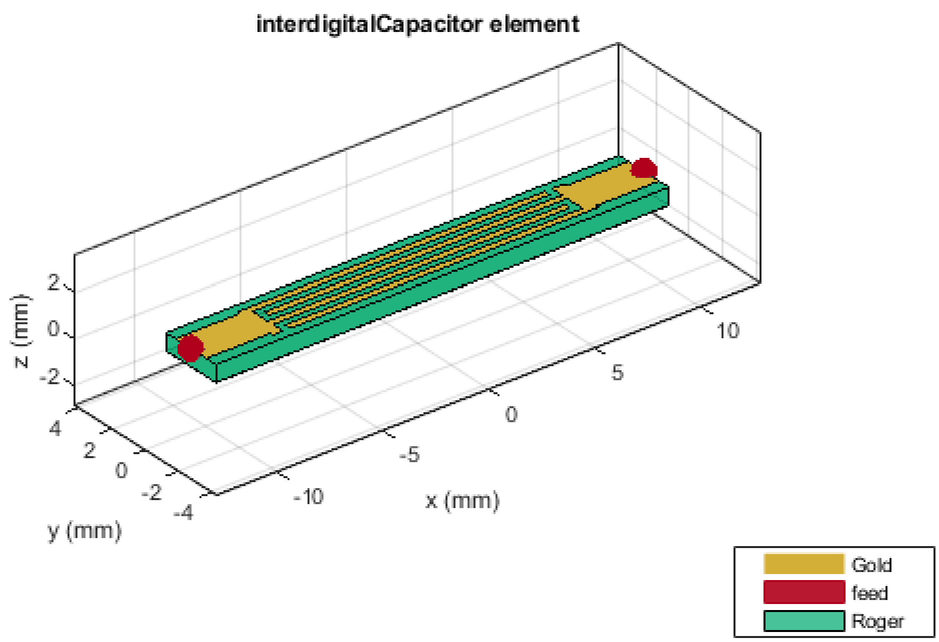
```
idcapacitor = interdigitalCapacitor;
sub = dielectric("FR4","Teflon");
sub.Thickness =[0.00003 0.00003];
idcapacitor.Substrate = sub;
idcapacitor.Height = 0.00003;
show(idcapacitor);
```



Analyze Interdigital Capacitor Using Behavioral S-parameters

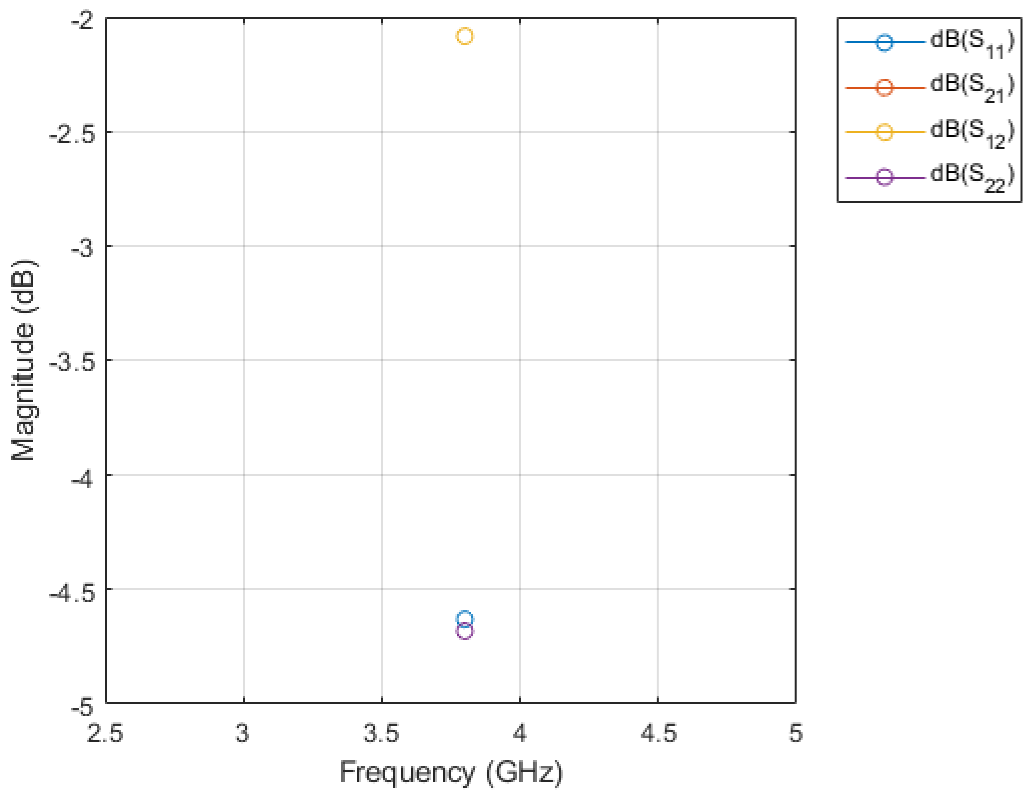
Create an interdigital capacitor using gold as the conductor.

```
capacitor = interdigitalCapacitor;
capacitor.Conductor = metal("Gold");
show(capacitor)
```



Compute and plot the behavioral S-parameters of the capacitor at 3.8 GHz.

```
spar = sparameters(capacitor,3.8e9,Behavioral=true);
rfplot(spar)
```



More About

Parametric Analysis Guidelines

- The capacitance of an interdigital capacitor is directly proportional to its physical parameters such as NumFingers, FingerLength, FingerWidth, and FingerSpacing.
- Increasing the height of the dielectric decreases the parasitic capacitance.

To design the capacitor at high frequency consider the following assumptions:

- Increasing the number of fingers creates a periodic and smooth structure.
- Capacitor dimensions should be much smaller than the quarter wavelength.

References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

[2] Jungreuthmayer, Christian, Gerald M. Birnbaumer, Peter Ertl, and Jürgen Zanghellini. “Improving the Measurement Sensitivity of Interdigital Dielectric Capacitors (IDC) by Optimizing the Dielectric Property of the Homogeneous Passivation Layer.” *Sensors and Actuators B: Chemical* 162, no. 1 (February 2012): 418–24. https://doi.org/10.1016/j.snb.2011.12.009.

[3] Ruppin, R. “Surface Polaritons of a Left-Handed Material Slab.” *Journal of Physics: Condensed Matter* 13, no. 9 (March 5, 2001): 1811–18. https://doi.org/10.1088/0953-8984/13/9/304.

[4] Caloz, Christophe, and Tatsuo Itoh. *Electromagnetic Metamaterials: Transmission Line Theory and Microwave Applications: The Engineering Approach*. Hoboken, NJ, USA: John Wiley & Sons, Inc., 2005. https://doi.org/10.1002/0471754323.

See Also

`spiralInductor`

Introduced in R2021b

bendCurved

Create curved bend shape on X-Y plane

Description

Use the `bendCurved` object to create a curved bend shape on the X-Y plane.

Creation

Syntax

```
bend = bendCurved
bend = bendCurved(Name=Value)
```

Description

`bend = bendCurved` creates a curved bend shape on the X-Y plane.

`bend = bendCurved(Name=Value)` sets "Properties" on page 1-60 using one or more name-value arguments. For example, `bendCurved(ReferencePoint=[1 1])` creates a curved bend shape with the reference point at `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of curved bend shape
`'myCurvedbend'` (default) | character vector | string scalar

Name of the curved bend shape, specified as a character vector or a string scalar.

Example: `bend = bendCurved(Name="bendcurve1")`

Data Types: `char`

`ReferencePoint` — Reference point
`[0 0]` (default) | two-element vector

Reference point for the curved bend shape in Cartesian coordinates, specified as a two-element vector.

Example: `bend = bendCurved(ReferencePoint=[1 1])`

Data Types: `double`

`Length` — Length of curved bend shape
`[0.0100 0.01000]` (default) | two-element vector

Length of the curved bend shape in meters, specified as a two-element vector.

Example: `bend = bendCurved(Length=[0.0500 0.0500])`

Data Types: `double`

Width — Width of curved bend shape

`[0.0500 0.0500]` (default) | two-element vector

Width of the curved bend shape in meters, specified as a two-element vector.

Example: `bend = bendCurved(Width=[0.0100 0.0100])`

Data Types: `double`

CurveRadius — Radius of corner

`0.0035` (default) | positive scalar

Radius of the corner in meters, specified as a positive scalar.

Example: `bend = bendCurved(CurveRadius=2)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Curved Bend Shape

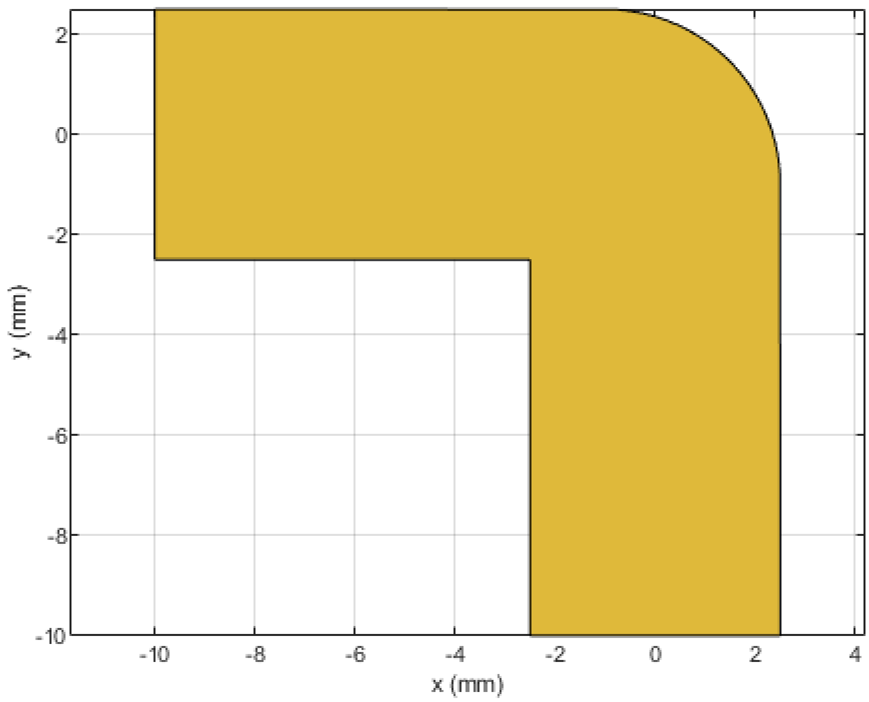
Create a curved bend shape with default properties.

```
bend = bendCurved
```

```
bend = 
  bendCurved with properties:

              Name: 'myCurvedbend'
    ReferencePoint: [0 0]
            Length: [0.0100 0.0100]
             Width: [0.0050 0.0050]
       CurveRadius: 0.0035
```

View the shape.

```
show(bend)
```



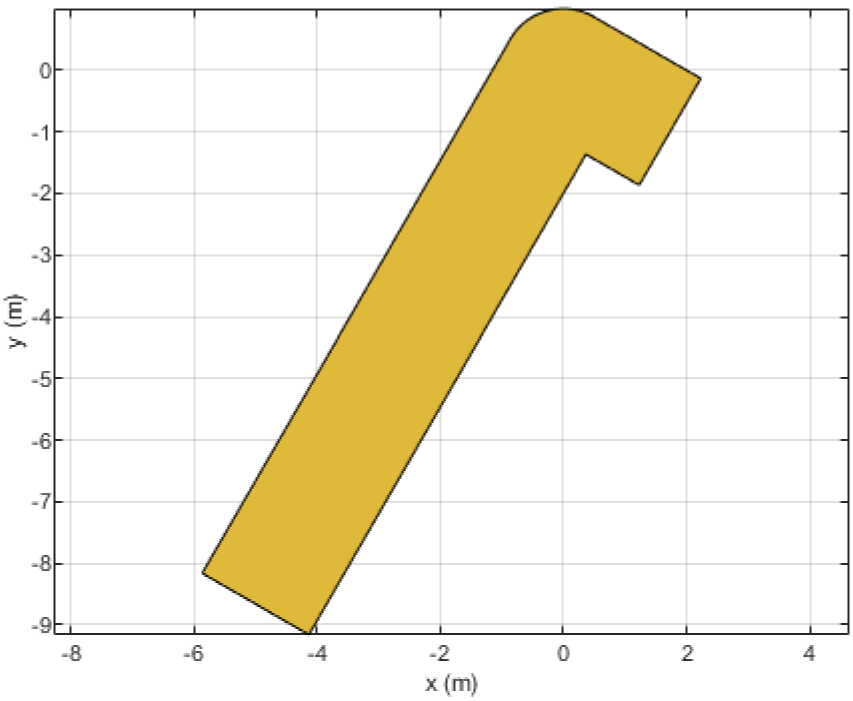
Mesh Rotated Curved Bend Shape

Create a curved bend shape of lengths of 10 m and 2 m, width of 2 m, and rotate it about the Z-axis by 60 degrees.

```
bend = bendCurved(Length=[10 2],Width=[2 2],CurveRadius=1)
```

```
bend = 
  bendCurved with properties:

              Name: 'myCurvedbend'
    ReferencePoint: [0 0]
            Length: [10 2]
             Width: [2 2]
       CurveRadius: 1
```

```
bend = rotateZ(bend,60);
show(bend)
```

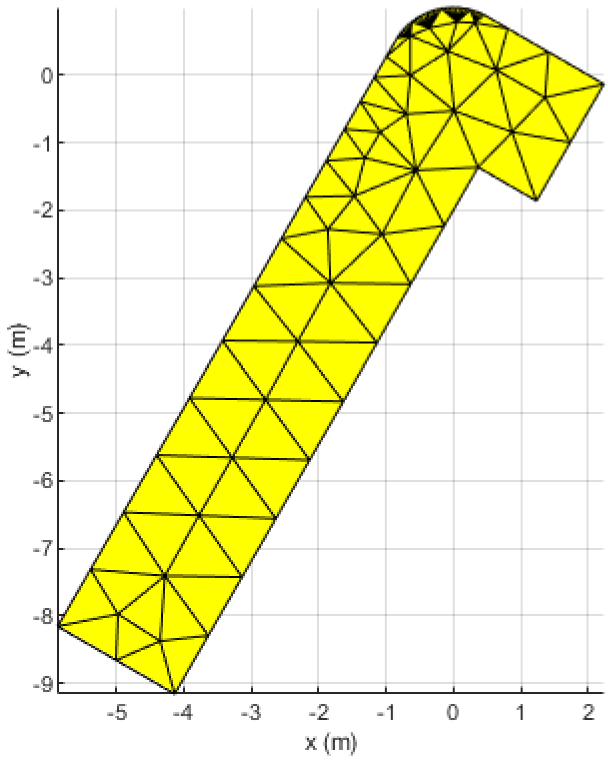


Mesh the curved bend shape at a maximum edge length of 1 m.

```
meshconfig(bend,"manual")
```

```
ans = struct with fields:
     NumTriangles: 0
    NumTetrahedra: 0
         NumBasis: []
    MaxEdgeLength: []
         MeshMode: 'manual'
```

```
mesh(bend,MaxEdgeLength=1)
```



See Also
`bendRightAngle` | `bendMitered`

Introduced in R2021b

bendMitered

Create mitered bend shape on X-Y plane

Description

Use the `bendMitered` object to create a mitered bend shape on the X-Y plane.

Creation

Syntax

```
bend = bendMitered
bend = bendMitered(Name=Value)
```

Description

`bend = bendMitered` creates a mitered bend shape on the X-Y plane.

`bend = bendMitered(Name=Value)` sets "Properties" on page 1-65 using one or more name-value arguments. For example, `bendMitered(ReferencePoint=[1 1])` creates a mitered bend shape with the reference point at `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of mitered bend shape
`'myMiteredbend'` (default) | character vector | string scalar

Name of the mitered bend shape, specified as a character vector or a string scalar.

Example: `bend = bendMitered(Name="bendmitered1")`

Data Types: `char`

`ReferencePoint` — Reference point
`[0 0]` (default) | two-element vector

Reference point for the mitered bend shape in Cartesian coordinates, specified as a two-element vector.

Example: `bend = bendMitered(ReferencePoint=[1 2])`

Data Types: `double`

`Length` — Length of mitered bend shape
`[0.0100 0.0100]` (default) | two-element vector

Length of the mitered bend shape in meters, specified as a two-element vector.

Example: `bend = bendMitered(Length=[0.005 0.005])`

Data Types: `double`

`Width` — Width of mitered bend shape
`[0.0050 0.0500]` (default) | two-element vector

Width of the mitered bend shape in meters, specified as a two-element vector.

Example: `bend = bendMitered(Width=[1 1])`

Data Types: `double`

`MiterDiagonal` — Length of miter diagonal
`0.0035` (default) | positive scalar

Length of the miter diagonal in meters, specified as a positive scalar.

Example: `bend = bendMitered(MiterDiagonal=2)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Mitered Bend

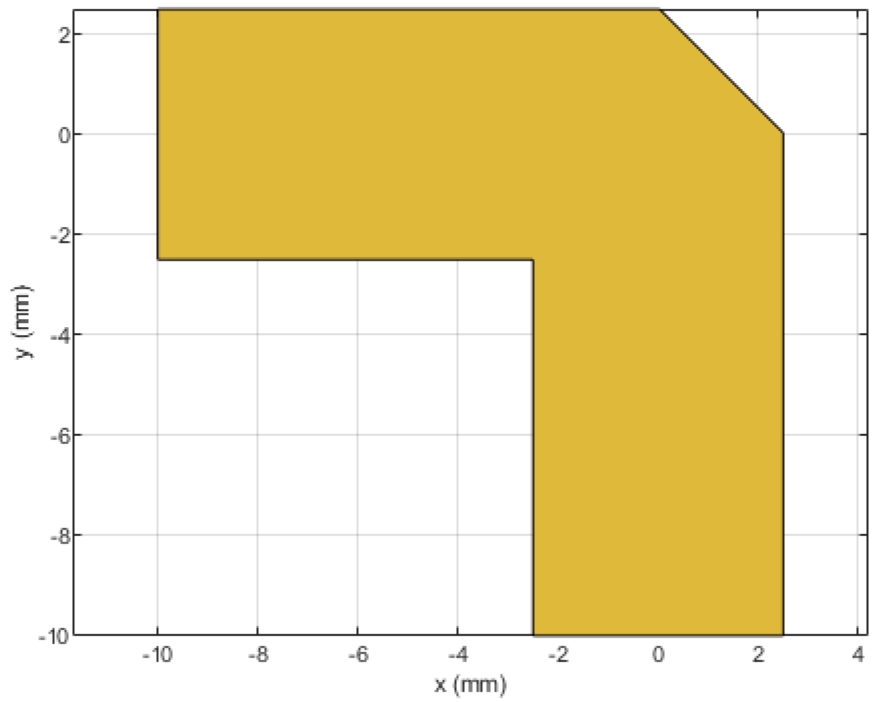
Create a mitered bend with default properties.

```
bend = bendMitered
```

```
bend = 
  bendMitered with properties:

             Name: 'myMiteredbend'
   ReferencePoint: [0 0]
           Length: [0.0100 0.0100]
            Width: [0.0050 0.0050]
    MiterDiagonal: 0.0035
```

View the shape.

```
show(bend)
```



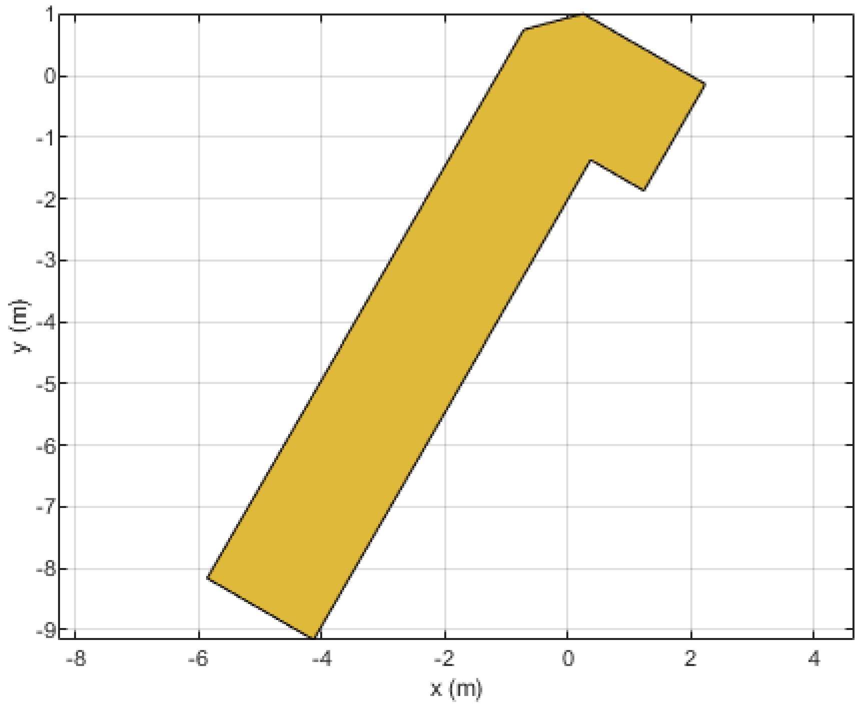
Mesh Rotated Mitered Bend Shape

Create a mitered bend shape of lengths of 10 m and 2 m, width of 2 m, and rotate it about the Z-axis by 60 degrees.

```
bend = bendMitered(Length=[10 2],Width=[2 2],MiterDiagonal=1);
bend = rotateZ(bend,60)
```

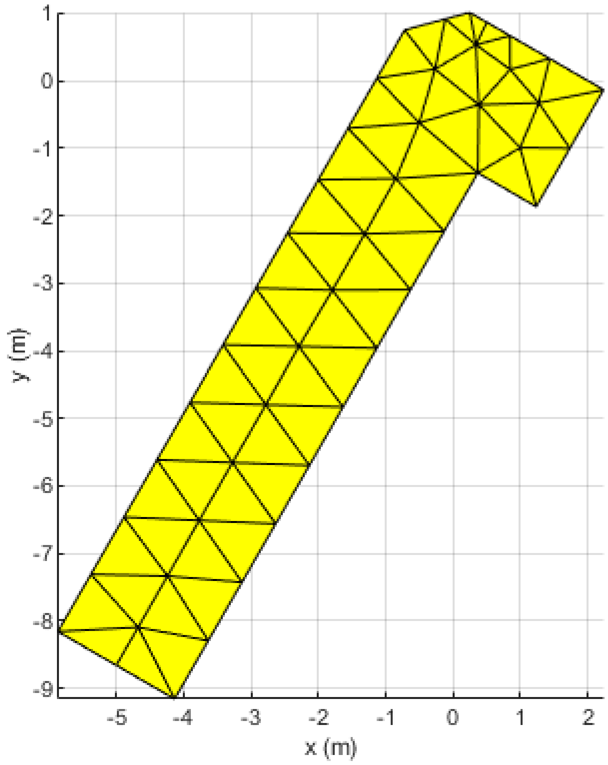
```
bend = 
  bendMitered with properties:

              Name: 'myMiteredbend'
    ReferencePoint: [0 0]
            Length: [10 2]
             Width: [2 2]
     MiterDiagonal: 1
```

```
show(bend)
```



Mesh the mitered bend shape at a maximum edge length of 1 m.

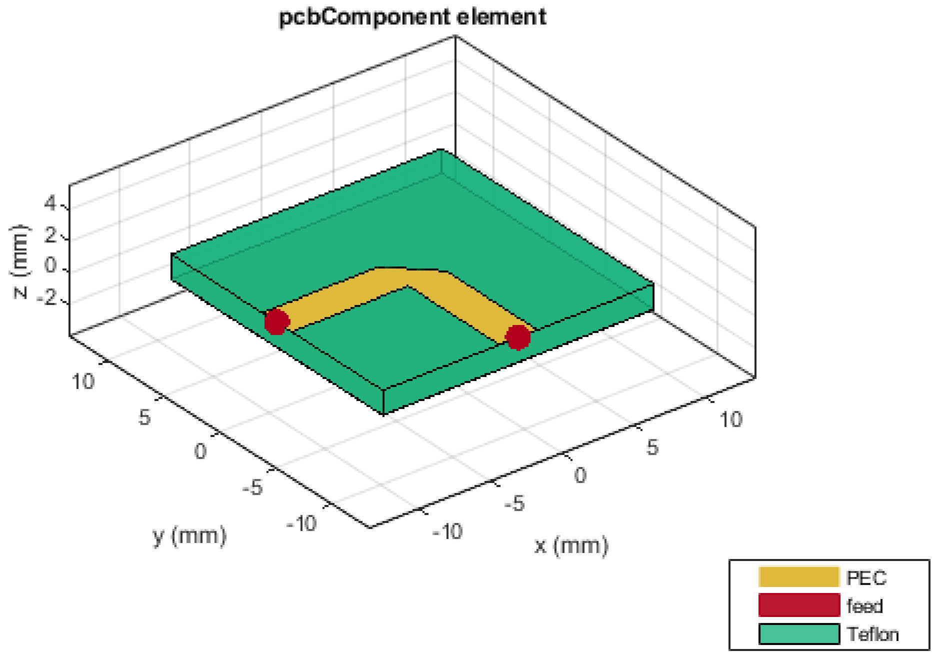
```
mesh(bend,MaxEdgeLength=1)
```



Use Behavioral Model to Calculate S-Parameters of Mitered Bend Microstrip

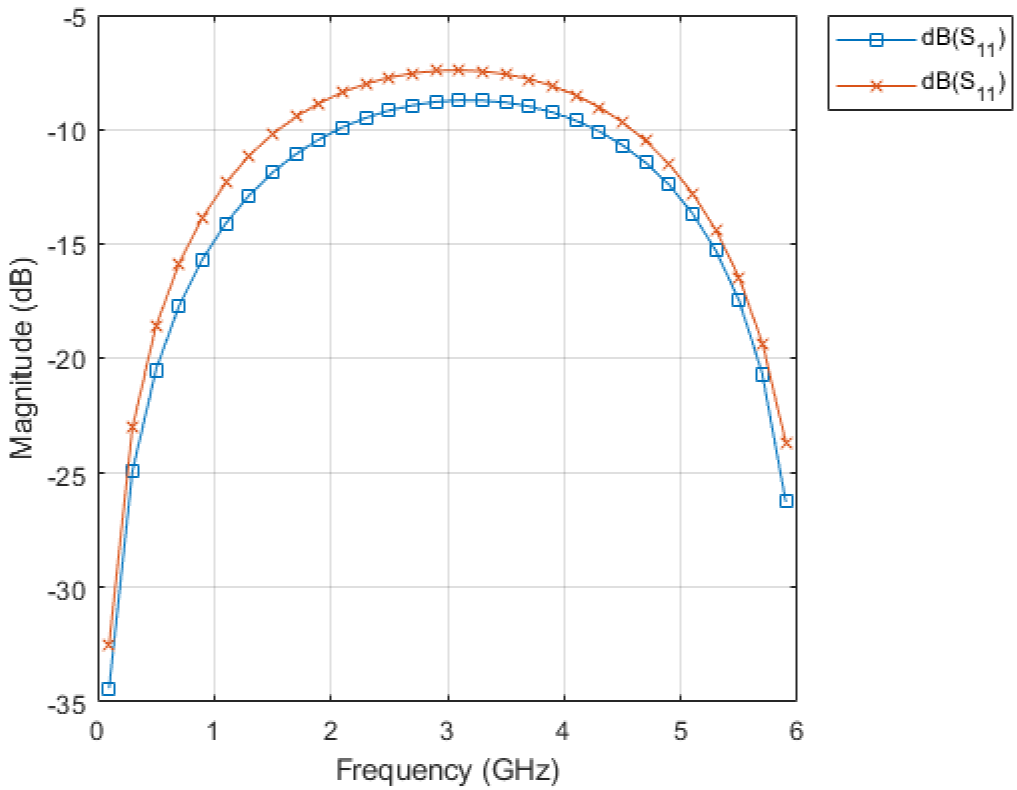
Create mitered bend microstrip.

```
m = design(microstripLine,6e9,"Z0",75);
layer2d = bendMitered('Length',[m.Length/2 m.Length/2],...
"Width",[m.Width m.Width],'MiterDiagonal',sqrt(2)*m.Width);
robj = pcbComponent(layer2d);
robj.BoardThickness = m.Substrate.Thickness;
robj.Layers{2} = m.Substrate;
show(robj)
```



Compute and plot s-parameters.

```
freq = (1:2:60)*100e6;
Sckt = sparameters(robj,freq,'Behavioral',true);
Sem = sparameters(robj,freq);
rfplot(Sckt,1,1,'db','-s')
hold on
rfplot(Sem,1,1,'db','-x')
```



Reference:

M. Kirschning, R. H. Jansen and N. H. L. Koster, "Measurement and Computer-Aided Modeling of Microstrip Discontinuities by an Improved Resonator Method," 1983 IEEE MTT-S International Microwave Symposium Digest, Boston, MA, USA, 1983, pp. 495-497, doi: 10.1109/MWSYM.1983.1130959.

See Also

`bendRightAngle` | `bendCurved`

Introduced in R2021b

bendRightAngle

Create right-angle bend shape on X-Y plane

Description

Use the `bendRightAngle` object to create a right-angle bend shape on the X-Y plane.

Creation

Syntax

```
bend = bendRightAngle
bend = bendRightAngle(Name=Value)
```

Description

`bend = bendRightAngle` creates a right-angle bend shape on the X-Y plane.

`bend = bendRightAngle(Name=Value)` sets "Properties" on page 1-72 using one or more name-value arguments. For example, `bendRightAngle(ReferencePoint=[1 1])` creates a right-angle bend shape with the reference point at `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of right-angle bend shape
`'myRightAnglebend'` (default) | character vector | string scalar

Name of the right-angle bend shape, specified as a character vector or a string scalar.

Example: `bend = bendRightAngle(Name="bendrightangle1")`

Data Types: `char`

`ReferencePoint` — Reference point
`[0 0]` (default) | two-element vector

Reference point for the right-angle bend shape in Cartesian coordinates, specified as a two-element vector.

Example: `bend = bendRightAngle(ReferencePoint=[1 2])`

Data Types: `double`

`Length` — Length of right-angle bend shape
`[0.0100 0.0100]` (default) | two-element vector

Length of the right-angle bend shape in meters, specified as a two-element vector.

Example: `bend = bendRightAngle(Length=[0.0500 0.0500])`

Data Types: `double`

`Width` — Width of right-angle bend shape
`[0.0050 0.0050]` (default) | two-element vector

Width of the right-angle bend shape in meters, specified as a two-element vector.

Example: `bend = bendRightAngle(Width=[0.0200 0.0200])`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Right-Angle Bend

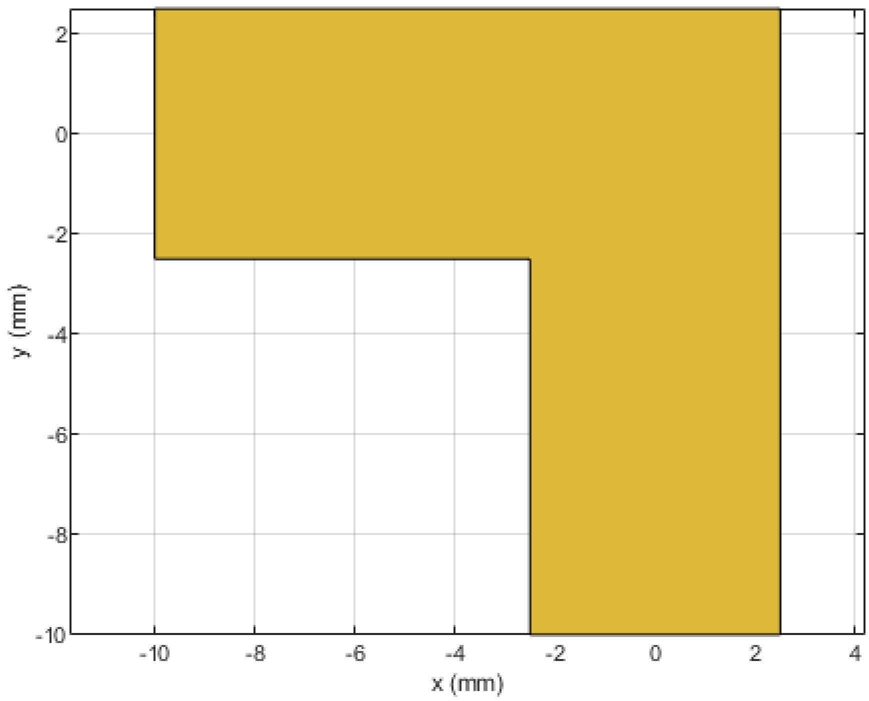
Create a right-angle bend with default properties.

```
bend = bendRightAngle
```

```
bend = 
  bendRightAngle with properties:

              Name: 'myRightAnglebend'
    ReferencePoint: [0 0]
            Length: [0.0100 0.0100]
             Width: [0.0050 0.0050]
```

View the shape.

```
show(bend)
```



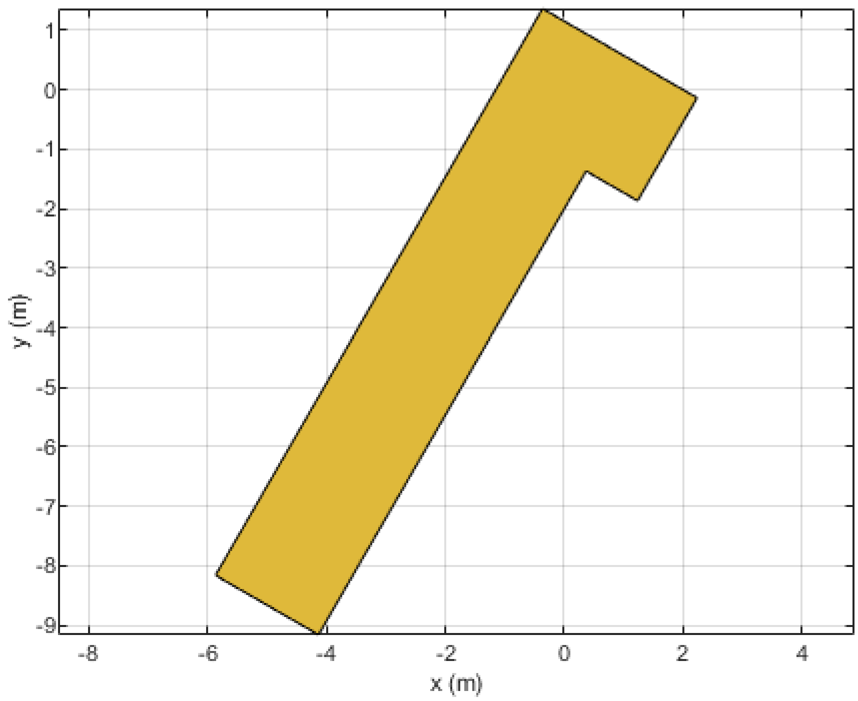
Mesh Rotated Right-Angle Bend Shape

Create a right-angle bend shape of lengths of 10 m and 2 m, width of 2m, and rotate it about the Z-axis by 60 degrees.

```
bend = bendRightAngle(Length=[10 2],Width=[2 2]);
bend = rotateZ(bend,60)
```

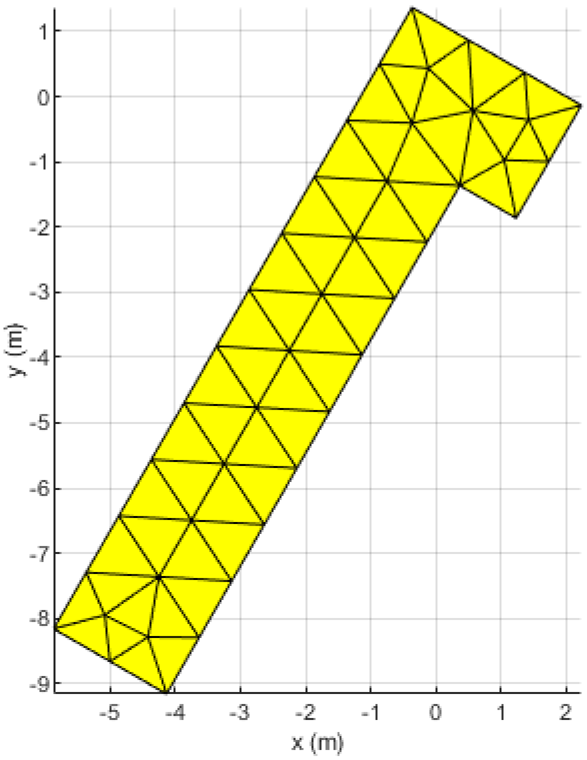
```
bend = 
  bendRightAngle with properties:

              Name: 'myRightAnglebend'
    ReferencePoint: [0 0]
            Length: [10 2]
             Width: [2 2]
```

```
show(bend)
```



Mesh the right-angle bend shape at a maximum edge length of 1 m.

```
mesh(bend,MaxEdgeLength=1)
```



See Also

bendMitered | bendCurved

Introduced in R2021b

curve

Create curved shape on X-Y plane

Description

Use the `curve` object to create a curved shape centered on the X-Y plane.

Creation

Syntax

```
curvedshape = curve
curvedshape = curve(Name=Value)
```

Description

`curvedshape = curve` creates a curved shape centered on the X-Y plane.

`curvedshape = curve(Name=Value)` sets “Properties” on page 1-77 using one or more name-value arguments. For example, `curve(ReferencePoint=[1 1])` creates a curved shape at the reference point `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of curved shape
`'myCurve'` (default) | character vector | string scalar

Name of the curved shape, specified as a character vector or string scalar.

Example: `curvedshape = curve(Name='curvedshape1')`

Data Types: `char` | `string`

`ReferencePoint` — Reference point
`[0 0]` (default) | two-element vector

Reference point of the curved shape in Cartesian coordinates, specified as a two-element vector.

Example: `curvedshape = curve(ReferencePoint=[1 1])`

Data Types: `double`

`Radius` — Radius of curved shape
`0.0100` (default) | positive scalar

Radius of the curved shape, specified as a positive scalar in meters.

Example: `curvedshape = curve(Radius=0.0300)`

Data Types: `double`

`Width` — Width of curved shape
`0.0020` (default) | positive scalar

Width of the curved shape, specified as a positive scalar in meters.

Example: `curvedshape = curve(Width=0.0030)`

Data Types: `double`

`ArcAngle` — Start angle and stop angle
`[0 180]` (default) | two-element vector

Start angle and stop angle in degrees, specified as a two-element vector.

Example: `curvedshape = curve(ArcAngle=[90 140)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Curved Shape

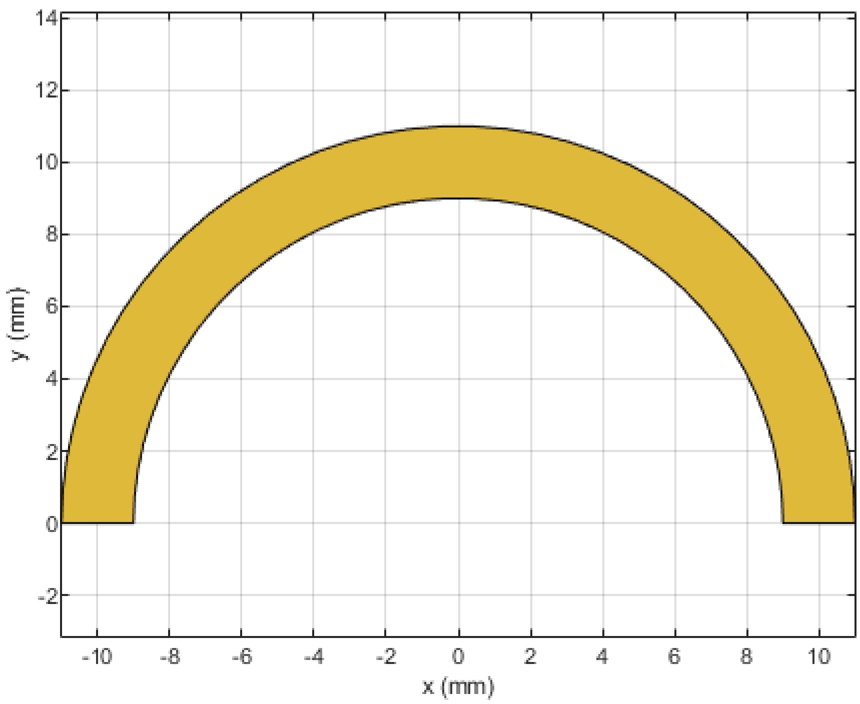
Create a curved shape with default properties.

```
curved = curve
```

```
curved = 
  curve with properties:

              Name: 'myCurve'
    ReferencePoint: [0 0]
            Radius: 0.0100
             Width: 0.0020
          ArcAngle: [0 180]
```

View the curved shape.

```
show(curved)
```



Mesh Rotated Curved Shape

Create a curved shape with the of 8 m and width of 2 m.

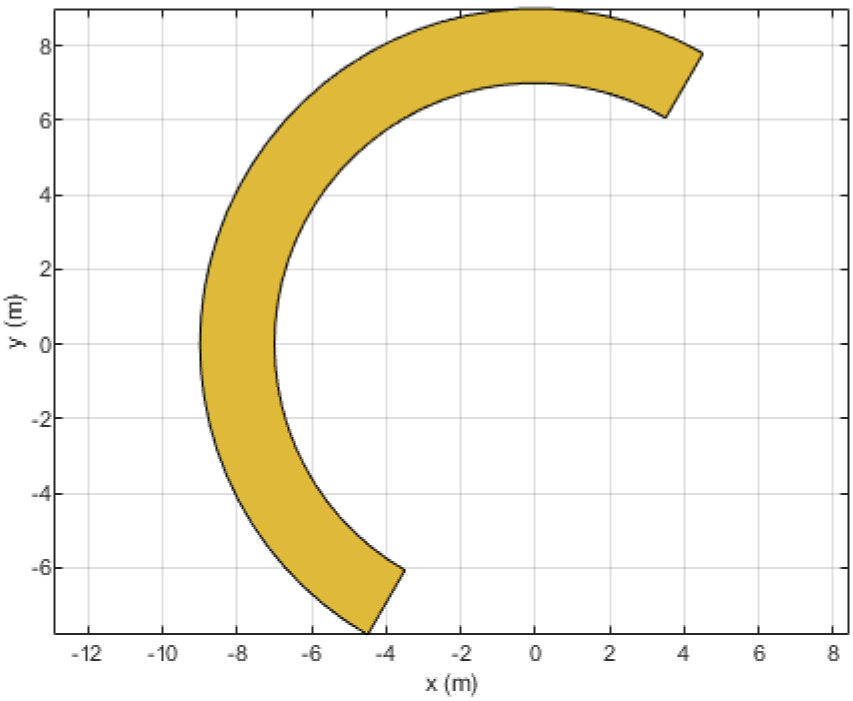
```
curved = curve(Radius=8,Width=2);
```

Rotate the shape by 60 degrees.

```
curved = rotateZ(curved,60)
```

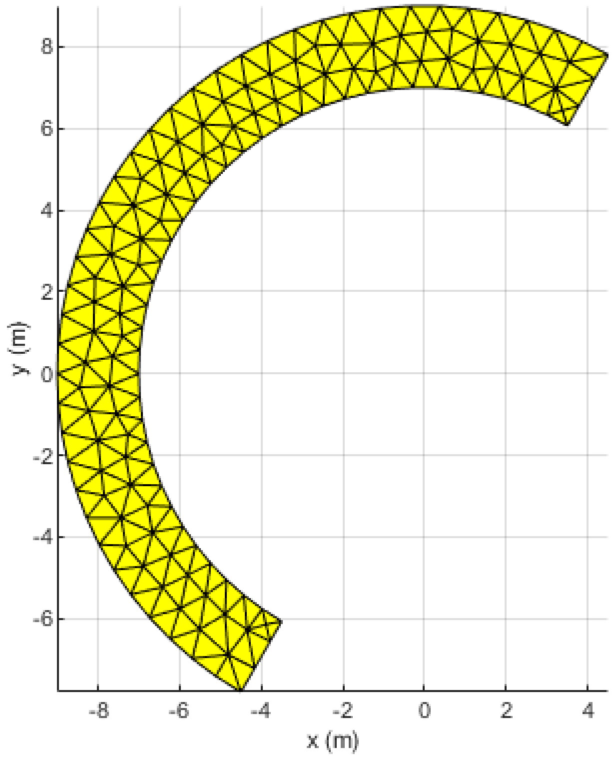
```
curved = 
  curve with properties:

              Name: 'myCurve'
    ReferencePoint: [0 0]
            Radius: 8
             Width: 2
          ArcAngle: [0 180]
```

```
show(curved)
```



Mesh the curved shape at a maximum edge length of 1 m.

```
mesh(curved,MaxEdgeLength=1)
```



See Also

`ringAnnular` | `ringSquare` | `radial` | `delta`

Introduced in R2021b

ringAnnular

Create annular ring on X-Y plane

Description

Use the `ringAnnular` object to create an annular ring on the X-Y plane.

Creation

Syntax

```
ring = ringAnnular
ring = ringAnnular(Name=Value)
```

Description

`ring = ringAnnular` creates an annular ring on the X-Y plane.

`ring = ringAnnular(Name=Value)` sets "Properties" on page 1-82 using one or more name-value arguments. For example, `ringAnnular(Center=[1 1])` creates an annular ring shape centered at `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of annular ring
`'myringAnnular'` (default) | character vector | string scalar

Name of the annular ring, specified as a character vector or string scalar.

Example: `ring = ringAnnular(Name='ringannular1')`

Data Types: `char` | `string`

`Center` — Center of annular ring
`[0 0]` (default) | two-element vector

Center of the annular ring in Cartesian coordinates, specified as a two-element vector.

Example: `ring = ringAnnular(Center=[1 1])`

Data Types: `double`

`InnerRadius` — Inner radius of annular ring
`0.0050` (default) | positive scalar

Inner radius of the annular ring, specified as a positive scalar in meters.

Example: `ring = ringAnnular(InnerRadius=0.006)`

Data Types: `double`

`Width` — Width of annular ring

`0.0020` (default) | positive scalar

Width of the annular ring, specified as a positive scalar in meters.

Example: `ring = ringAnnular(Width=3)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Annular Ring Shape

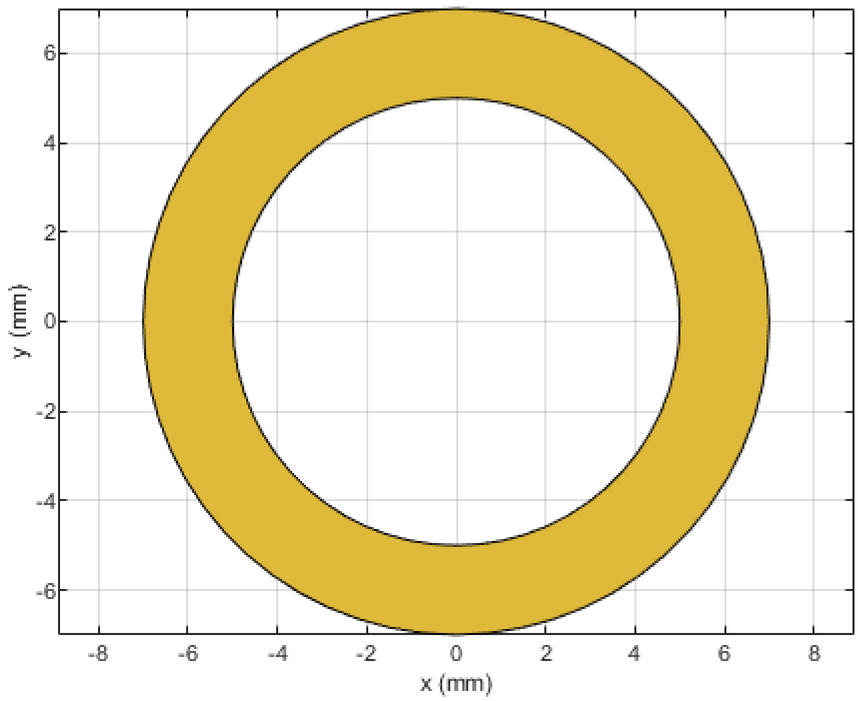
Create an annular ring shape with default properties.

```
ring = ringAnnular
```

```
ring = 
  ringAnnular with properties:

           Name: 'myringAnnular'
         Center: [0 0]
          Width: 0.0020
    InnerRadius: 0.0050
```

View the shape.

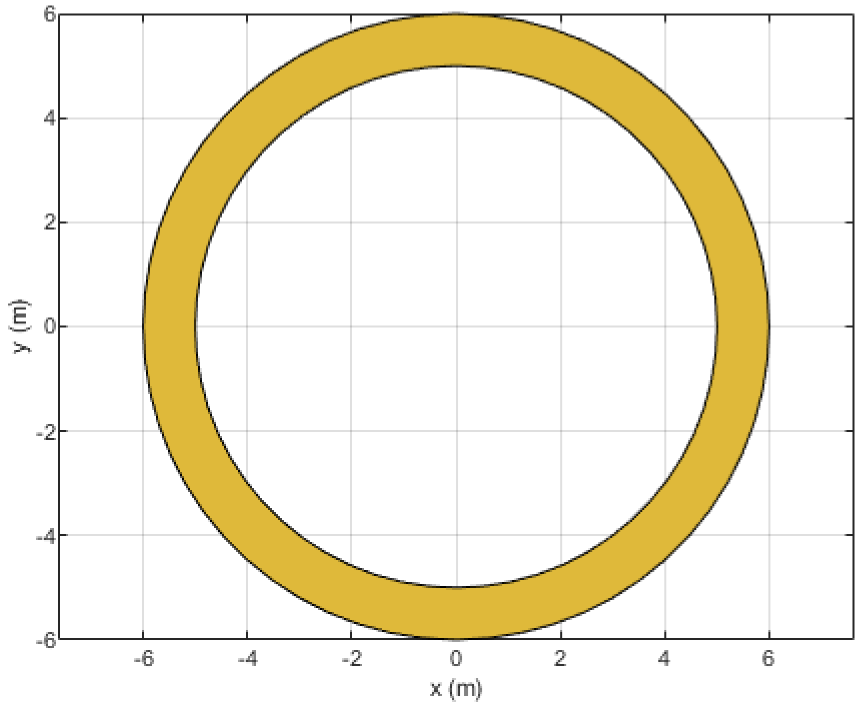
```
show(ring)
```



Mesh Annular Ring Shape

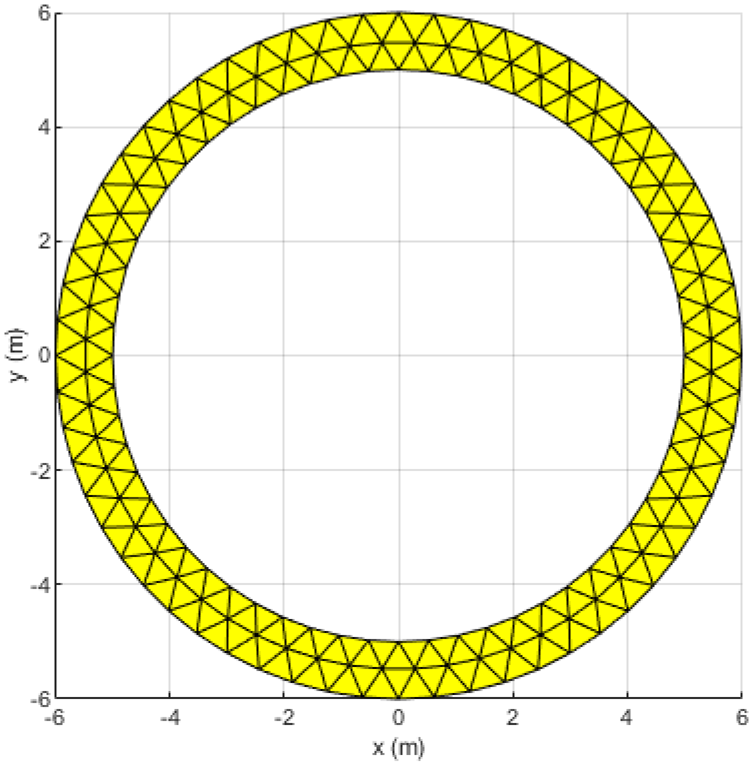
Create an annular ring shape of inner radius 5 m and width 1 m.

```
ring = ringAnnular(InnerRadius=5,Width=1);
show(ring)
```



Mesh the ring at a maximum edge length of 1 m.

```
mesh(ring,MaxEdgeLength=1)
```



See Also

`ringAnnular` | `ringSquare`

Introduced in R2021b

ringSquare

Create square ring on X-Y plane

Description

Use the `ringSquare` object to create a square ring on the X-Y plane.

Creation

Syntax

```
ring = ringSquare
ring = ringSquare(Name=Value)
```

Description

`ring = ringSquare` creates a square ring on the X-Y plane.

`ring = ringSquare(Name=Value)` sets "Properties" on page 1-87 using one or more name-value arguments. For example, `ringSquare(Center=[1 1])` creates an square ring shape centered at `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of square ring
`'myringSquare'` (default) | character vector | string scalar

Name of square ring, specified as a character vector or string scalar.

Example: `ring = ringSquare(Name='ringsquare1')`

Data Types: `char` | `string`

`Center` — Center of square ring
`[0 0]` (default) | two-element vector

Center of the square ring in Cartesian coordinates, specified as a two-element vector.

Example: `ring = ringAnnular(Center=[1 1])`

Data Types: `double`

`InnerSide` — Length of inner side
`0.0050` (default) | positive scalar

Length of the inner side, specified as a positive scalar in meters.

Example: `ring = ringSquare(InnerSide=0.0060)`

Data Types: `double`

`Width` — Width of square ring

`0.0020` (default) | positive scalar

Width of the square ring, specified as a positive scalar in meters.

Example: `ring = ringSquare(Width=0.0030)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Square Ring Shape

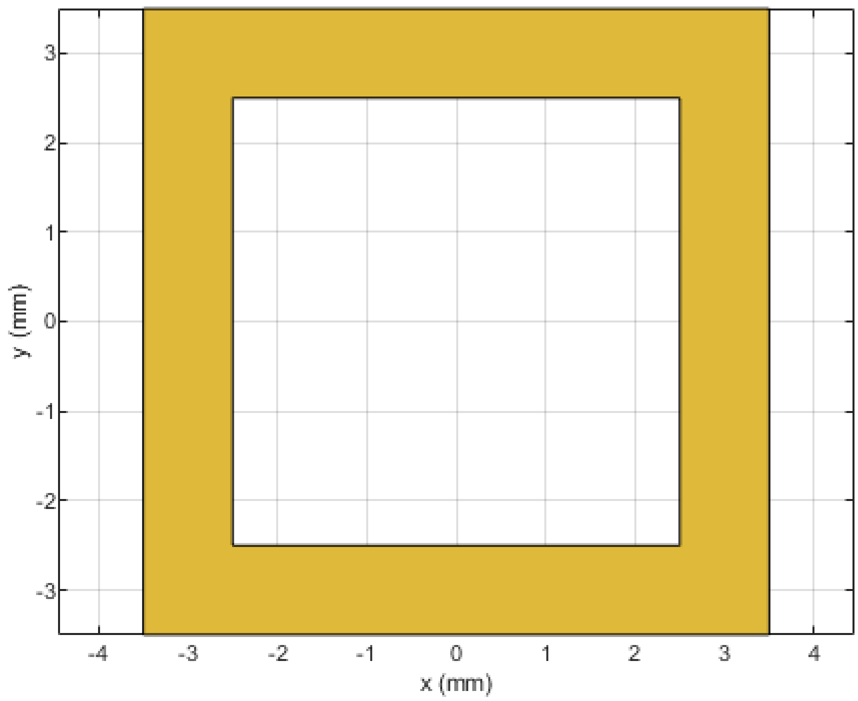
Create a square ring with default properties.

```
ring = ringSquare
```

```
ring = 
  ringSquare with properties:

         Name: 'myringSquare'
       Center: [0 0]
        Width: 0.0020
    InnerSide: 0.0050
```

View the shape.

```
show(ring)
```



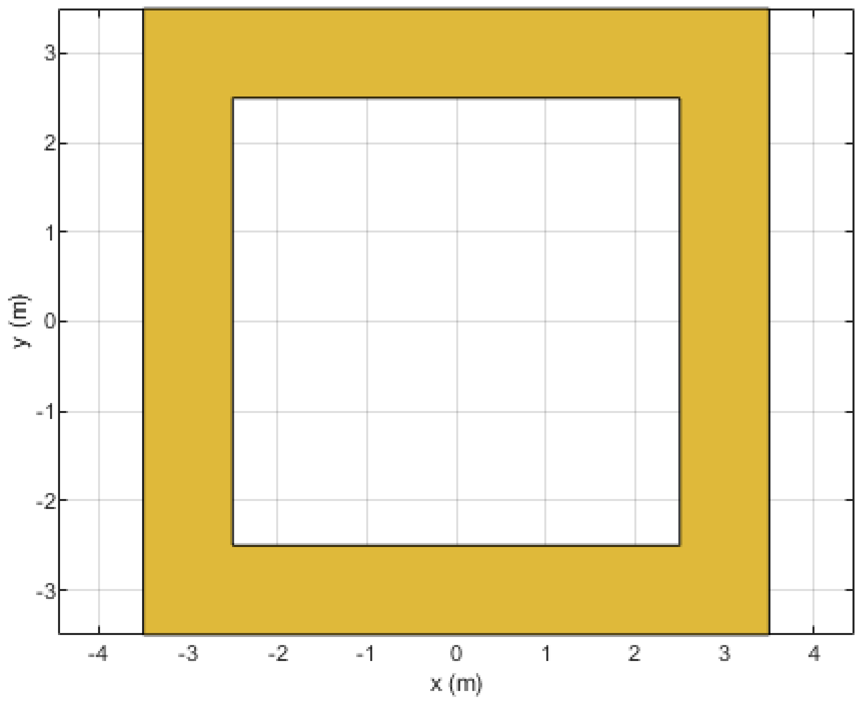
Mesh Square Ring Shape

Create a square ring with with the inner side length of 5 m and a width of 2 m.

```
ring = ringSquare(InnerSide=5,Width=2)
```

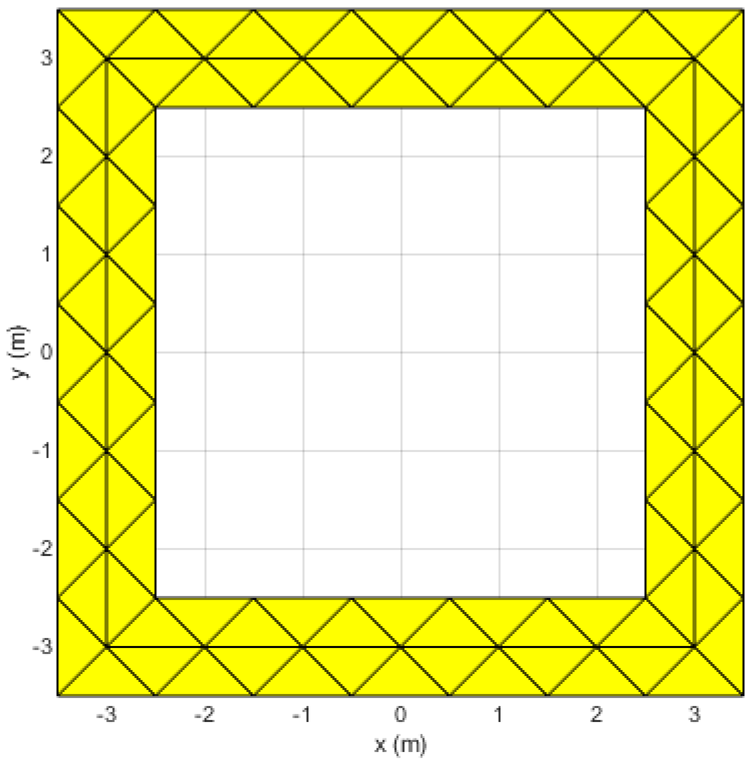
```
ring = 
  ringSquare with properties:

         Name: 'myringSquare'
       Center: [0 0]
        Width: 2
    InnerSide: 5
```

```
show(ring)
```



Mesh the square ring at a maximum edge length of 1 m.

```
mesh(ring,MaxEdgeLength=1)
```



See Also

`ringAnnular`

Introduced in R2021b

ubendCurved

Create U-bend with curved edges on X-Y plane

Description

Use the `ubendCurved` object to create a U-bend with curved edges on the X-Y plane.

Creation

Syntax

```
bend = ubendCurved
bend = ubendCurved(Name=Value)
```

Description

`bend = ubendCurved` creates a U-bend with curved edges on the X-Y plane.

`bend = ubendCurved(Name=Value)` sets “Properties” on page 1-92 using one or more name-value arguments. For example, `ubendCurved(ReferencePoint=[1 1])` creates a U-bend with curved edges at the reference point `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of curved U-bend
`'myCurvedubend'` (default) | character vector | string scalar

Name of the curved U-bend, specified as a character vector or a string scalar.

Example: `bend = ubendCurved(Name="ubendcurve1")`

Data Types: `char` | `string`

`ReferencePoint` — Reference point of curved U-bend
`[0 0]` (default) | two-element vector

Reference point of the curved U-bend in Cartesian coordinates, specified as a two-element vector.

Example: `bend = ubendCurved(ReferencePoint=[1 2])`

Data Types: `double`

`Length` — Length of curved U-bend
`[0.0150 0.0050 0.0150]` (default) | three-element vector

Length of the curved U-bend in meters, specified as a three-element vector of positive elements.

Example: `bend = ubendCurved(Length=[0.0050 0.0020 0.0050])`

Data Types: `double`

`Width` — Width of curved U-bend
`[0.0050 0.0050 0.0050]` (default) | three-element vector

Width of the curved U-bend in meters, specified as a three-element vector of positive elements.

Example: `bend = ubendCurved(Width=[0.0040 0.0040 0.0040])`

Data Types: `double`

`CurveRadius` — Radius of corner
`0.0035` (default) | positive scalar

Radius of the corner in meters, specified as a positive scalar.

Example: `bend = ubendCurved(CurveRadius=0.0025)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Curved U-Bend

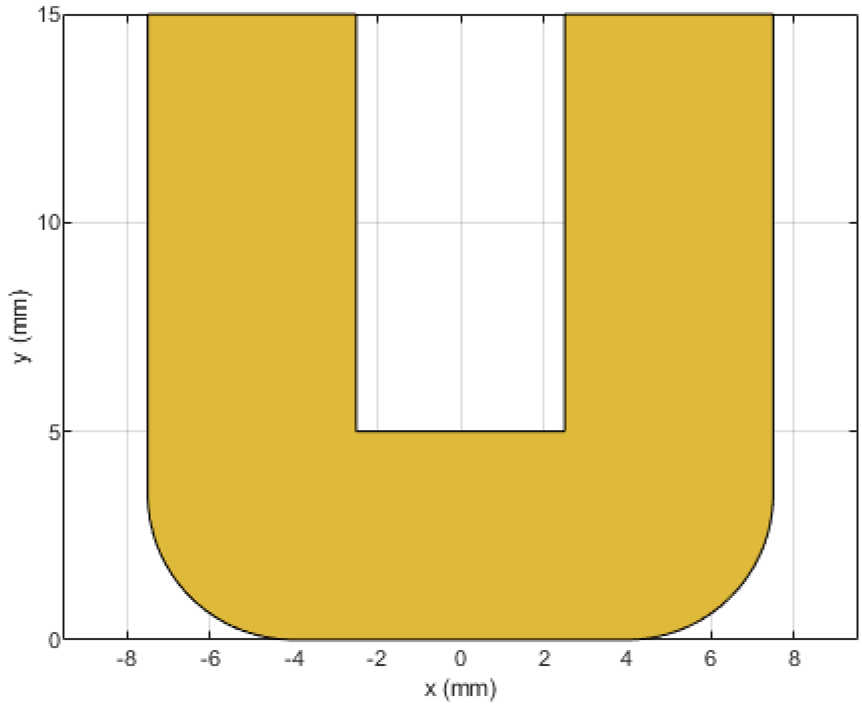
Create a curved U-bend with default properties.

```
curvedubend = ubendCurved
```

```
curvedubend = 
  ubendCurved with properties:

              Name: 'myCurvedubend'
    ReferencePoint: [0 0]
            Length: [0.0150 0.0050 0.0150]
             Width: [0.0050 0.0050 0.0050]
       CurveRadius: 0.0035
```

View the shape.

```
show(curvedubend)
```



See Also

`ubendMitered` | `ubendRightAngle`

Introduced in R2021b

ubendMitered

Create U-bend with mitered edges on X-Y plane

Description

Use the `ubendMitered` object to create a U-bend with mitered edges on the X-Y plane.

Creation

Syntax

```
bend = ubendMitered
bend = ubendMitered(Name=Value)
```

Description

`bend = ubendMitered` creates a U-bend with mitered edges on the X-Y plane.

`bend = ubendMitered(Name=Value)` sets "Properties" on page 1-95 using one or more name-value arguments. For example, `ubendMitered(ReferencePoint=[1 1])` creates a mitered U-bend at the reference point `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of mitered U-bend
`'myMiteredubend'` (default) | character vector | string scalar

Name of the mitered U-bend, specified as a character vector or a string scalar.

Example: `bend = ubendMitered(Name="ubendmitered1")`

Data Types: `char` | `string`

`ReferencePoint` — Reference point of mitered U-bend
`[0 0]` (default) | two-element vector

Reference point of the mitered U-bend in Cartesian coordinates, specified as a two-element vector.

Example: `bend = ubendMitered(ReferencePoint=[1 2])`

Data Types: `double`

`Length` — Length of mitered U-bend
`[0.0150 0.0050 0.0150]` (default) | three-element vector

Length of the mitered U-bend in meters, specified as a three-element vector of positive elements.

Example: `bend = ubendMitered(Length=[0.0250 0.0030 0.0250])`

Data Types: `double`

`Width` — Width of mitered U-bend
`[0.0050 0.0050 0.0050]` (default) | three-element vector

Width of the mitered U-bend in meters, specified as a three-element vector of positive elements.

Example: `bend = ubendMitered(Width=[0.0060 0.0060 0.0060])`

Data Types: `double`

`MiterDiagonal` — Length of miter diagonal
`0.0035` (default) | positive scalar

Length of the miter diagonal in meters, specified as a positive scalar.

Example: `bend = ubendMitered(MiterDiagonal=0.0060)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Mitered U-Bend

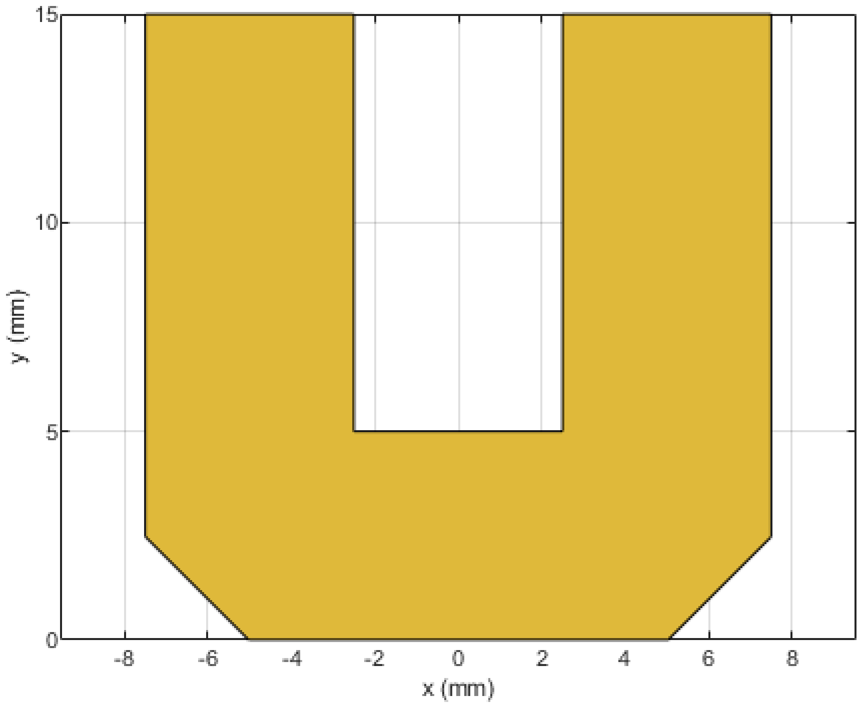
Create a mitered U-bend with default properties.

```
bend = ubendMitered
```

```
bend = 
  ubendMitered with properties:

             Name: 'myMiteredubend'
    ReferencePoint: [0 0]
           Length: [0.0150 0.0050 0.0150]
            Width: [0.0050 0.0050 0.0050]
     MiterDiagonal: 0.0035
```

View the shape.

```
show(bend)
```



See Also
ubendCurved | ubendRightAngle

Introduced in R2021b

ubendRightAngle

Create right-angle U-bend shape on X-Y plane

Description

Use the `ubendRightAngle` object to create a right-angle U-bend shape on the X-Y plane.

Creation

Syntax

```
bend = ubendRightAngle
bend = ubendRightAngle(Name=Value)
```

Description

`bend = ubendRightAngle` creates a right-angle U-bend shape on the X-Y plane.

`bend = ubendRightAngle(Name=Value)` sets "Properties" on page 1-98 using one or more name-value arguments. For example, `ubendRightAngle(ReferencePoint=[1 1])` creates a right-angle U-bend at the reference point `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of right-angle U-bend
`'myRightAngleubend'` (default) | character vector | string scalar

Name of the right-angle U-bend, specified as a character vector or a string scalar.

Example: `bend = ubendRightAngle(Name="ubendrightangle1")`

Data Types: `char` | `string`

`ReferencePoint` — Reference point of right-angle U-bend
`[0 0]` (default) | two-element vector

Reference point of the right-angle U-bend in Cartesian coordinates, specified as a two-element vector.

Example: `bend = ubendRightAngle(ReferencePoint=[1 2])`

Data Types: `double`

`Length` — Length of right-angle U-bend
`[0.0150 0.0050 0.0150]` (default) | three-element vector

Length of the right-angled U-bend in meters, specified as a three-element vector of positive elements.

Example: `bend = ubendRightAngle(Length=[0.0250 0.0150 0.0250])`

Data Types: `double`

`Width` — Width of right-angle U-bend
`[0.0050 0.0050 0.0050]` (default) | three-element vector

Width of the right-angle U-bend in meters, specified as a three-element vector of positive values.

Example: `bend = ubendRightAngle(Width=[0.0150 0.0150 0.0010])`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Right-Angle U-Bend

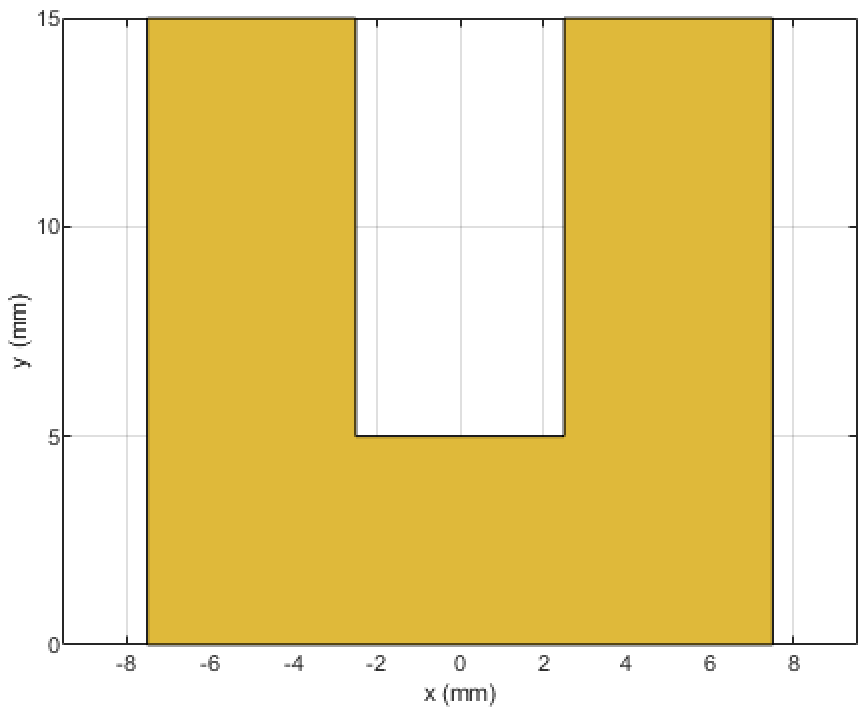
Create a right-angle U-bend with default properties.

```
bend = ubendRightAngle
```

```
bend = 
  ubendRightAngle with properties:

              Name: 'myRightAngleubend'
    ReferencePoint: [0 0]
            Length: [0.0150 0.0050 0.0150]
             Width: [0.0050 0.0050 0.0050]
```

View the shape.

```
show(bend)
```



See Also

`ubendCurved` | `ubendMitered`

Introduced in R2021b

tracePoint

Create custom line trace based on specified X and Y coordinates

Description

Use the `tracePoint` object to create a custom line trace by tracing a line along the specified X and Y coordinates.

Creation

Syntax

```
trace = tracePoint
trace = tracePoint(Name=Value)
```

Description

`trace = tracePoint` creates a line trace using default properties.

`trace = tracePoint(Name=Value)` sets "Properties" on page 1-101 using one or more name-value arguments. For example, `tracePoint(Width=0.0050)` creates a line trace with the width of `0.0050`. Properties not specified retain their default values.

Properties

`Name` — Name of custom line trace
`'tracePoint'` (default) | character vector | string scalar

Name of the custom line trace, specified as a character vector or string scalar.

Example: `trace = tracePoint(Name='tracepoint1')`

Data Types: `char` | `string`

`TracePoints` — Coordinates of custom line trace
`10-by-2`array (default) | *n*-by-2-array

Coordinates of custom line trace, specified as a *n*-by-2-array of X and Y coordinates.

Example: `trace = tracePoint(TracePoints=[0 0;0 -10;6 -10;6 0]])`

Data Types: `double`

`Width` — Width of line trace
`0.002` (default) | positive scalar

Width of the line trace, specified as a positive scalar in meters. This value is applied to all line segments in the custom trace.

Example: `trace = tracePoint(Width=0.005)`

Data Types: `double`

`Corner` — Corner where two line segments interface
`"Sharp"` (default) | `"Miter"` | `"Smooth"`

Corner where two line segments interface, specified as either `"Sharp"`, `"Miter"`, or `"Smooth"`. To apply the same value to all corners, specify a string scalar. For a different value for all corners, specify a (*n*-2)-by-1 vector of strings.

Example: `trace = tracePoint(Corner="Miter")`

Data Types: `string`

Object Functions

add	Boolean unite operation on two RF PCB shapes
subtract	Boolean subtraction operation on two RF PCB shapes
intersect	Boolean intersection operation on two RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
minus	Shape1 - Shape2 for RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
translate	Move RF PCB shape to new location
scale	Change size of RF PCB shape by fixed amount

Examples

Create Default Custom Line

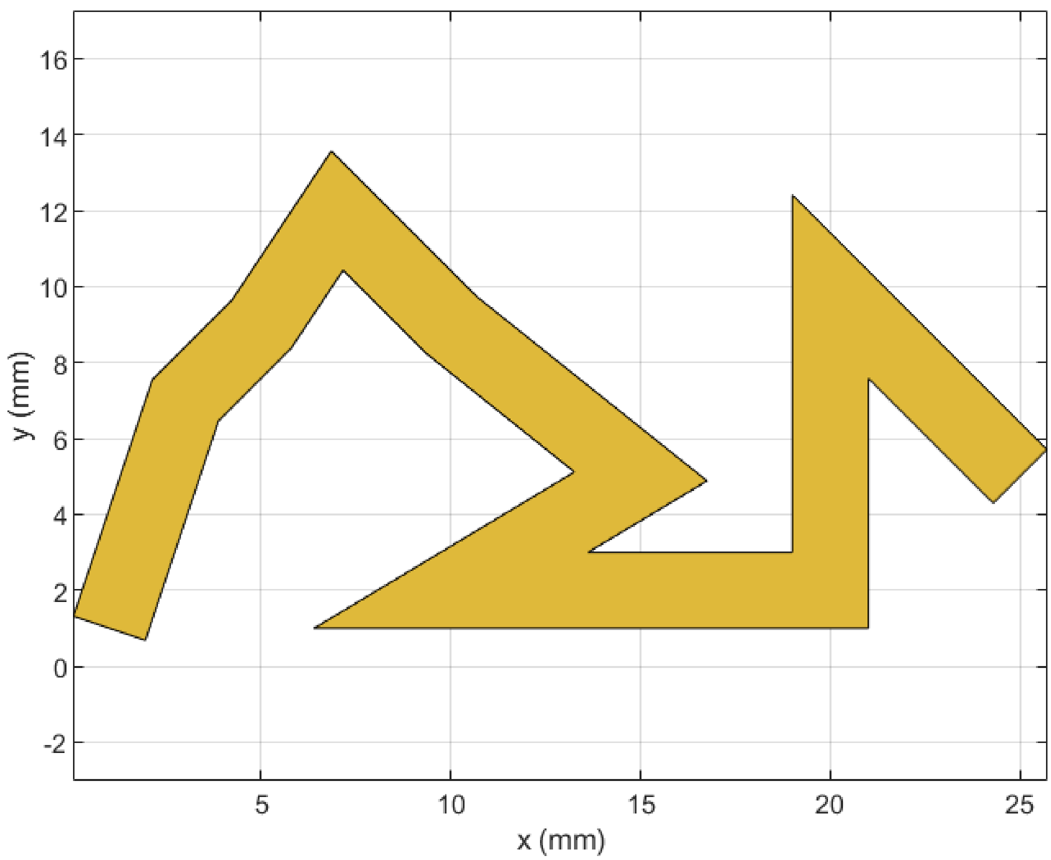
Create a custom line using default properties.

```
customLine = tracePoint
```

```
customLine = 
  tracePoint with properties:

           Name: 'mytracePoint'
    TracePoints: [10×2 double]
          Width: 0.0020
         Corner: "Sharp"
```

View the trace.

```
show(customLine)
```



Rotate and Mesh Custom Line

Create a custom line trace using default properties.

```
customLine = tracePoint;
```

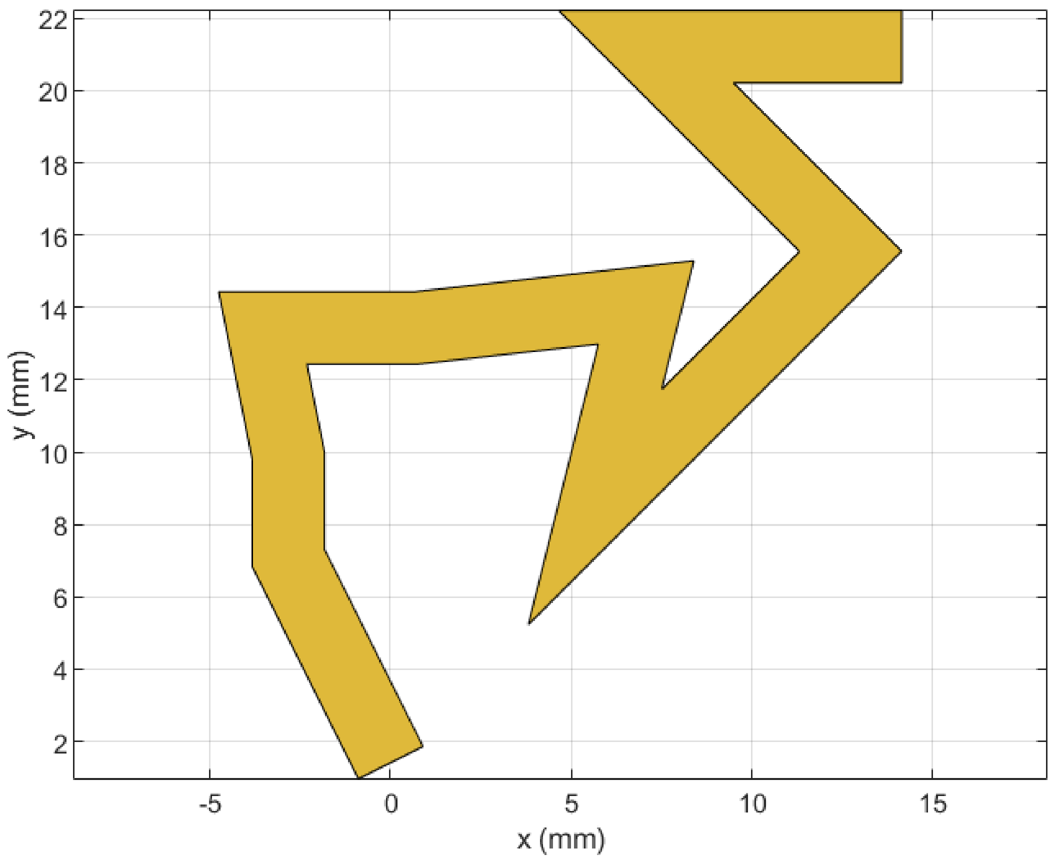
Rotate the trace by 45 degrees along the Z-axis.

```
customLine = rotateZ(customLine,45)
```

```
customLine = 
  tracePoint with properties:

           Name: 'mytracePoint'
    TracePoints: [10×2 double]
          Width: 0.0020
         Corner: "Sharp"
```

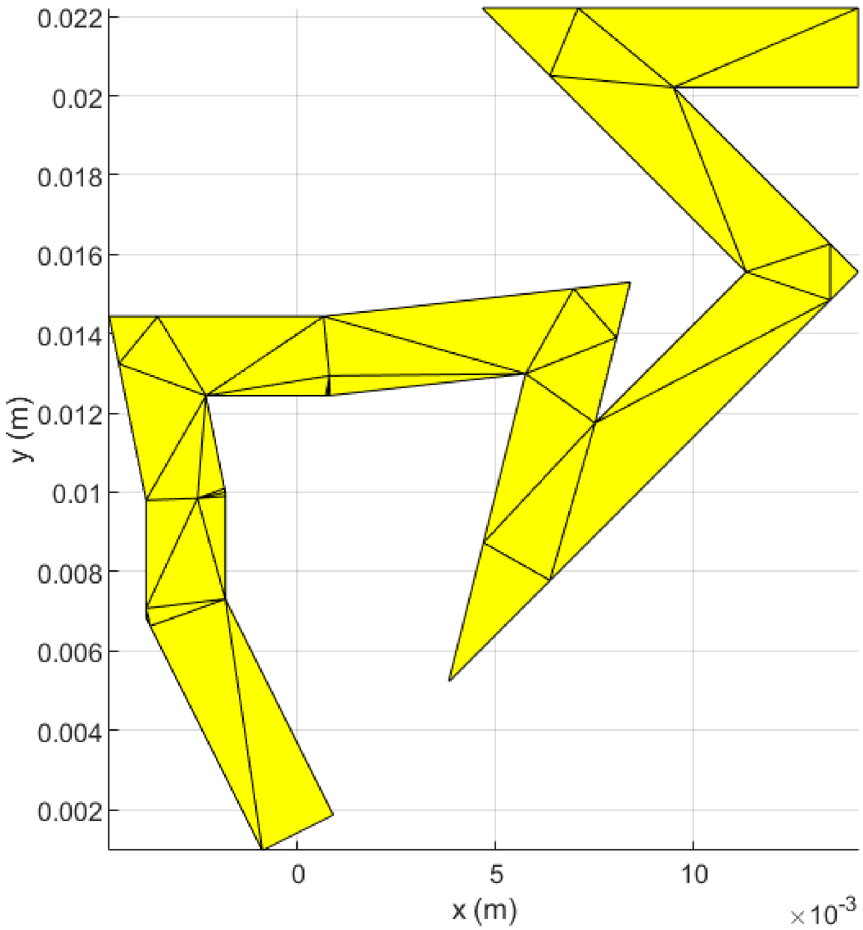
View the trace.

```
show(customLine)
```



Mesh the custom line trace at a maximum edge length of 1 m.

```
mesh(customLine,MaxEdgeLength=1)
```



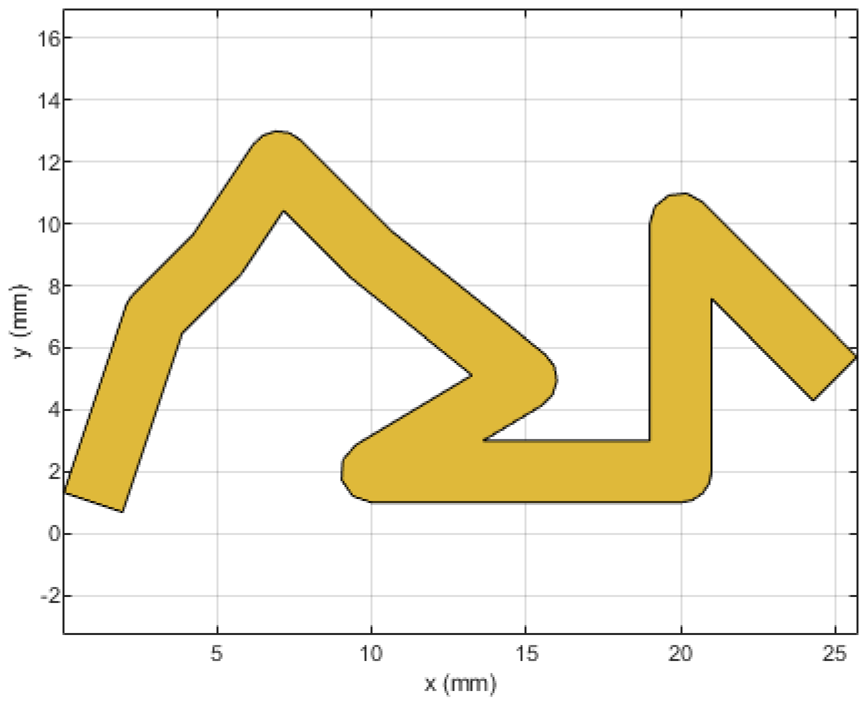
Create Custom Trace with Smooth Corners

Create a custom trace line with smooth corners.

```
customLine = tracePoint(Name='tracepoint',Corner="Smooth")
```

```
customLine =
  tracePoint with properties:

           Name: 'tracepoint'
    TracePoints: [10x2 double]
          Width: 0.0020
         Corner: "Smooth"
```

```
show(customLine)
```



See Also

traceLine | traceCross | traceTee | traceRectangular | traceSpiral

Introduced in R2021b

traceLine

Create line trace

Description

Use the `traceLine` object to create a line trace. You can use this object to create lines of different lengths and different angles

Creation

Syntax

```
trace = traceLine
trace = traceLine(Name=Value)
```

Description

`trace = traceLine` creates a line trace using default properties.

`trace = traceLine(Name=Value)` sets properties using one or more name-value arguments. For example, `traceLine('StartPoint',[1 1])` creates a line trace shape with the starting point of `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of line trace
`'traceLine'` (default) | character vector | string scalar

Name of the line trace, specified as a character vector or string scalar.

Example: `customtrace = traceLine(Name=traceline1)`

Data Types: `char` | `string`

`StartPoint` — Start point of line trace
`[0 0]` (default) | two-element vector

Start point of the line trace in Cartesian coordinates, specified as a two-element vector.

Example: `customtrace = traceLine(StartPoint=[1 1])`

Data Types: `double`

`Length` — Length of line trace
`[0.0200 0.0200 0.0200 0.0150]` (default) | *n*-by-1 vector

Length of line trace, specified as an *n*-by-1 vector in meters. Each element represents the length of a line segment.

Example: `customtrace = traceLine(Length=[0.0100 0.0100 0.0100 0.0500])`

Data Types: `double`

`Width` — Width of line trace
`0.0050` (default) | *n*-by-1 vector

Width of the line trace, specified as a scalar or an *n*-by-1 vector in meters. Each element represents the length of a line segment

Example: `customtrace = traceLine(Width=[0.0040 0.0040 0.0040 0.0050])`

Data Types: `double`

`Angle` — Angle of line trace
`[90 0 -90 45]` (default) | *n*-by-1 vector

Angle of the line trace, specified as an *n*-by-1 vector in degrees. Each element represents an angle of a line segment.

Example: `customtrace = traceLine(Angle=[40 10 -40 35)`

Data Types: `double`

`Corner` — Corner where two line segments interface
`"Sharp"` (default) | `"Miter"` | `"Smooth"`

Corner where two line segments interface, specified as either `"Sharp"`, `"Miter"`, or `"Smooth"`. To apply the same value to all corners, specify a string scalar. For a different value for all corners, specify a (*n*-2)-by-1 vector of strings.

Example: `trace = traceLine(Corner="Miter")`

Data Types: `string`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Custom Line Trace

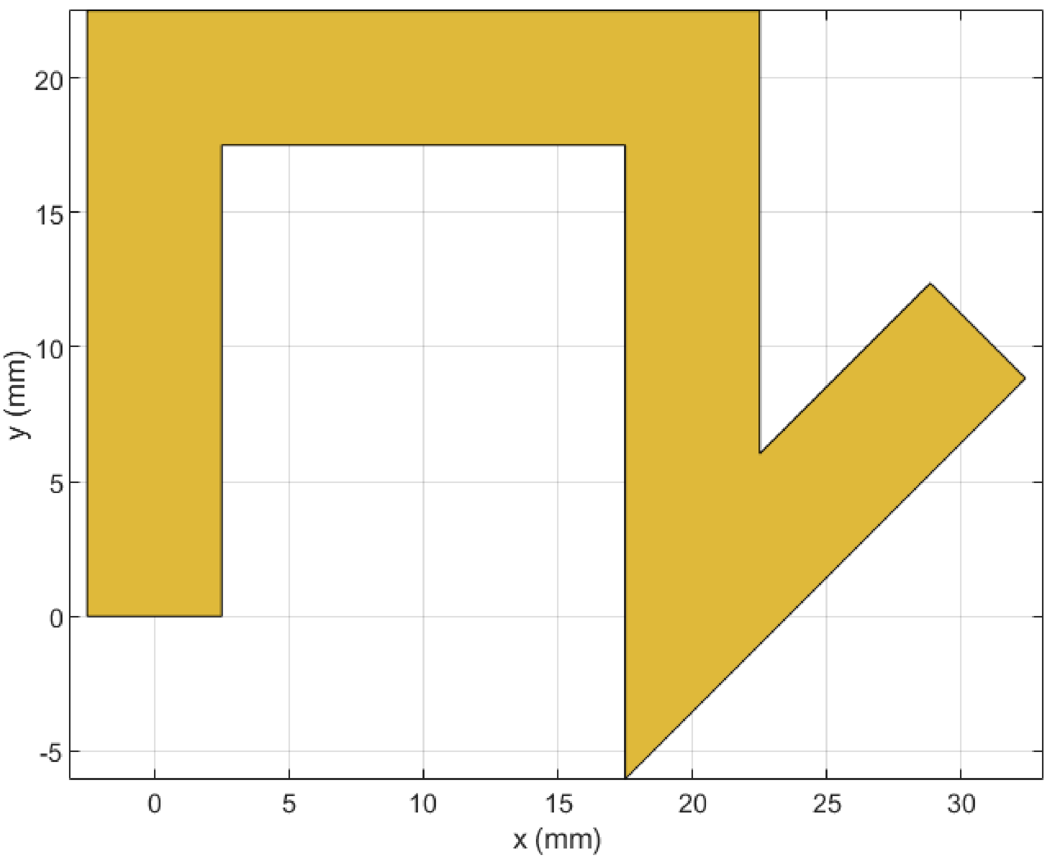
Create a custom line trace with default properties.

```
customLine = traceLine
```

```
customLine = 
  traceLine with properties:

         Name: 'mytraceLine'
   StartPoint: [0 0]
       Length: [0.0200 0.0200 0.0200 0.0150]
        Width: 0.0050
        Angle: [90 0 -90 45]
       Corner: "Sharp"
```

View the trace.

```
show(customLine)
```



Rotate and Mesh Line Trace

Create a line trace.

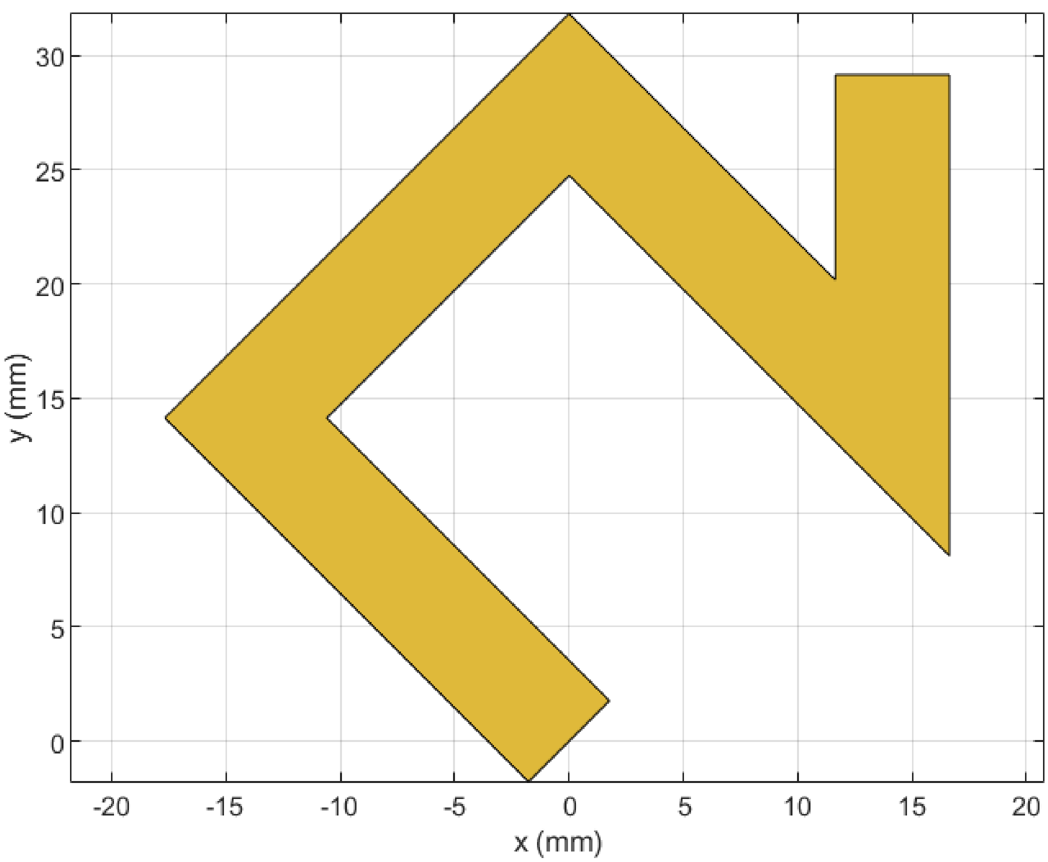
```
customLine = traceLine;
```

Rotate the trace by 45 degrees along the Z-axis.

```
customLine = rotateZ(customLine,45)
```

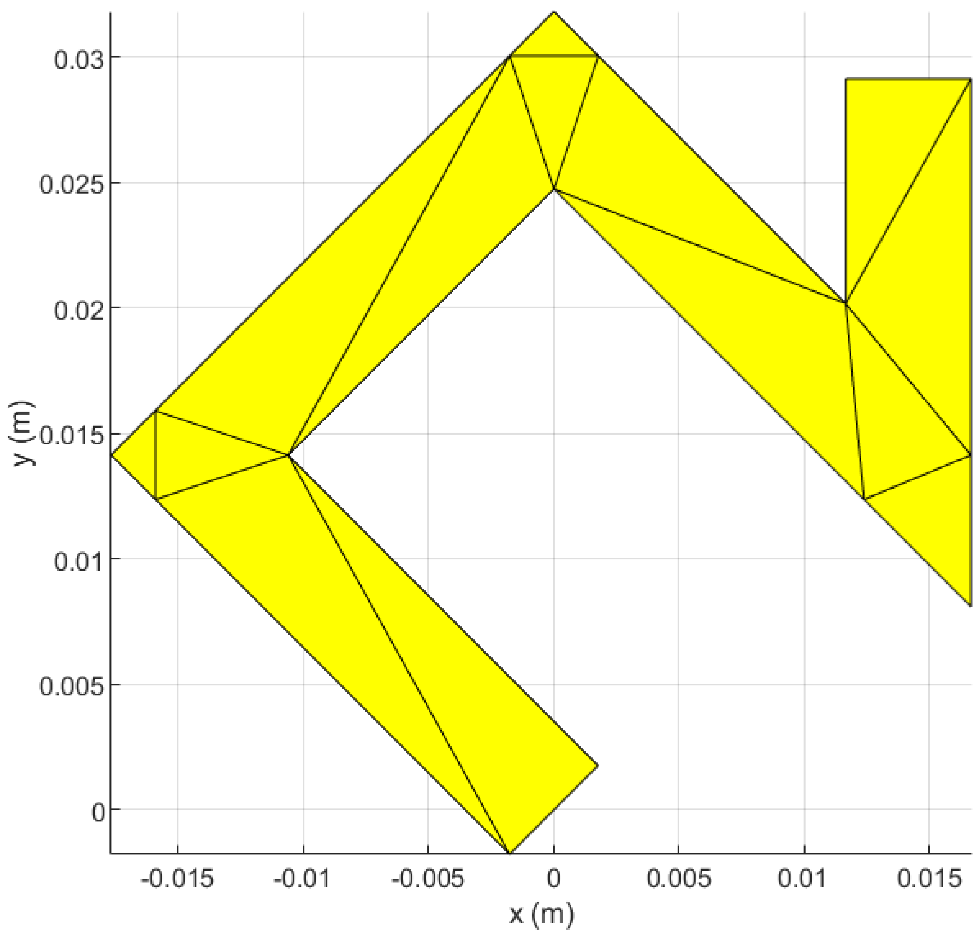
```
customLine = 
  traceLine with properties:

          Name: 'mytraceLine'
    StartPoint: [0 0]
        Length: [0.0200 0.0200 0.0200 0.0150]
         Width: 0.0050
         Angle: [90 0 -90 45]
        Corner: "Sharp"
```

```
show(customLine)
```



Mesh the line trace at a maximum edge length of 1 m.

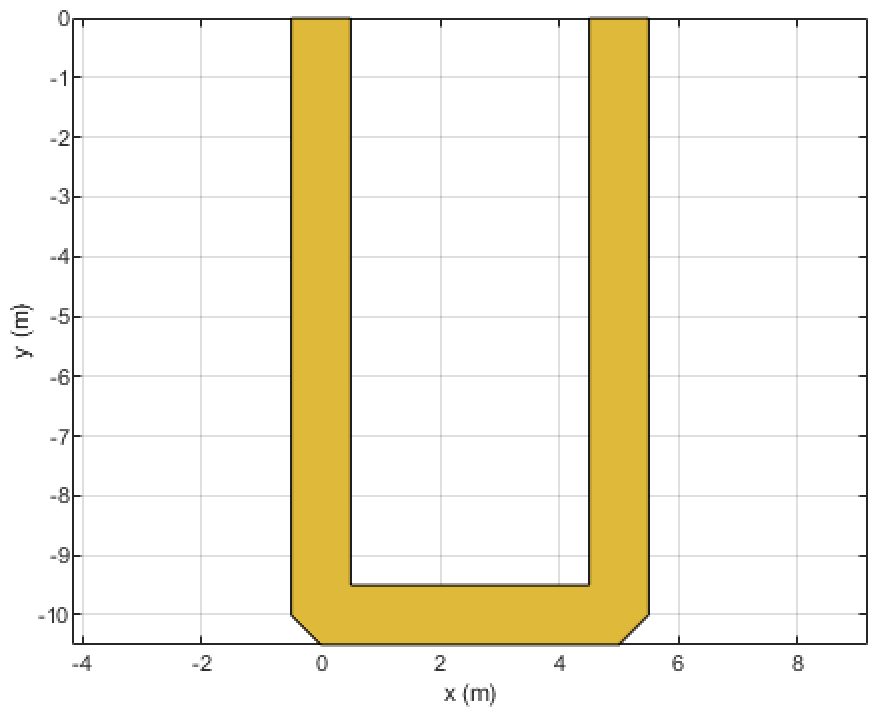
```
mesh(customLine,MaxEdgeLength=1)
```



U-Shaped Line Trace

Create and view a U-shaped line trace with mitered bends and a width of 1 m.

```
Ushapeline = traceLine;
Ushapeline.Length = [10 5 10];
Ushapeline.Angle  = [-90 0 90];
Ushapeline.Width  = 1;
Ushapeline.Corner = 2;
show(Ushapeline);
```



See Also

`traceCross` | `traceTee` | `traceRectangular` | `traceSpiral` | `tracePoint`

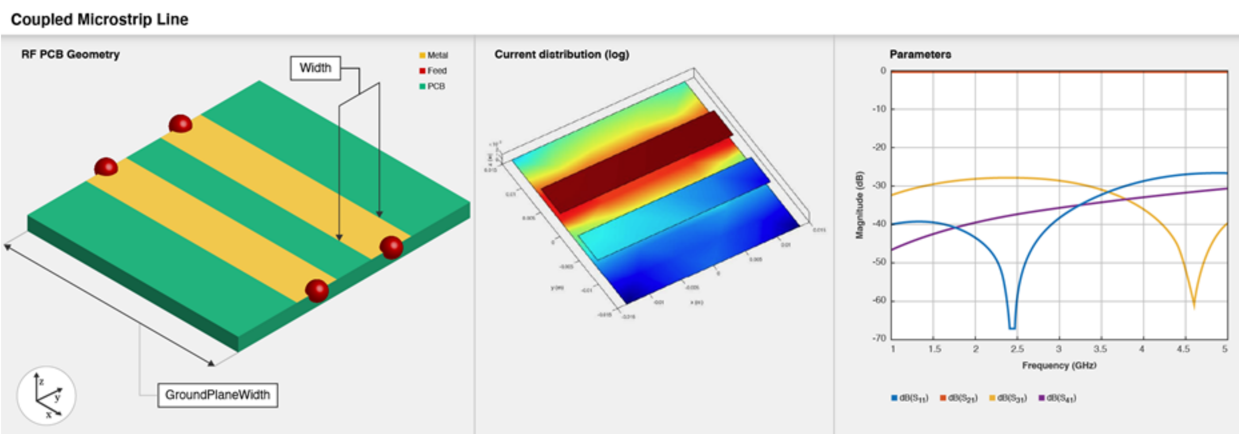
Introduced in R2021b

coupledMicrostripLine

Create coupled microstrip transmission line

Description

Use the `coupledMicrostripLine` object to create a coupled microstrip transmission line. Coupled microstrip transmission lines are used to design directional couplers and filters. The combination of even and odd mode impedances determines the coupling ratio between the direct arm and the coupled arm.



Creation

Syntax

```
coupledmicrostrip = coupledMicrostripLine
coupledmicrostrip = coupledMicrostripLine(Name=Value)
```

Description

`coupledmicrostrip = coupledMicrostripLine` creates a default coupled microstrip transmission line with a Teflon substrate. The default properties are for a resonating frequency of 1.5 GHz.

`coupledmicrostrip = coupledMicrostripLine(Name=Value)` sets "Properties" on page 1-113 using one or more name-value arguments. For example, `coupledMicrostripLine(Length=0.0300)` creates a coupled microstrip transmission line of length 0.0300 meters.

Properties

`Length` — Length of coupled microstrip line
`0.0271` (default) | positive scalar

Length of the coupled microstrip line in meters, specified as a positive scalar.

Example: `coupledmicrostrip = coupledMicrostripLine(Length=0.0300)`

Data Types: `double`

`Width` — Width of coupled microstrip line
`0.0051` (default) | positive scalar

Width of the coupled microstrip line in meters, specified as a positive scalar.

Example: `coupledmicrostrip = coupledMicrostripLine(Width=0.0041)`

Data Types: `double`

`Spacing` — Distance between the direct arm and the coupled arm
`0.0046` (default) | positive scalar

Distance between the direct arm and the coupled arm of the coupled microstrip transmission line, specified as a positive scalar in meters.

Example: `coupledmicrostrip = coupledMicrostripLine(Spacing=0.00300)`

Data Types: `double`

`Height` — Height of coupled microstrip line
`0.0016` (default) | positive scalar

Height of the coupled microstrip line from the ground plane, specified as a positive scalar in meters. In the case of a multilayer substrate, use the `Height` property to create a coupled microstrip line at the interface of the two dielectrics.

Example: `coupledmicrostrip = coupledMicrostripLine(Height=0.0023)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane
`0.0300` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `coupledmicrostrip = coupledMicrostripLine(GroundPlaneWidth=0.0400)`

Data Types: `double`

`Substrate` — Type of dielectric material
`'Teflon'` (default) | `dielectric` object

Type of the dielectric material used as a substrate, specified as a `dielectric` object. The thickness of the default dielectric material Teflon is `0.0016` m or the same as the `Height` property.

Example: `d = dielectric("FR4"); coupledmicrostrip = coupledMicrostripLine(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'PEC'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("PEC"); coupledmicrostrip =coupledMicrostripLine(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design coupled microstrip transmission line around particular frequency
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

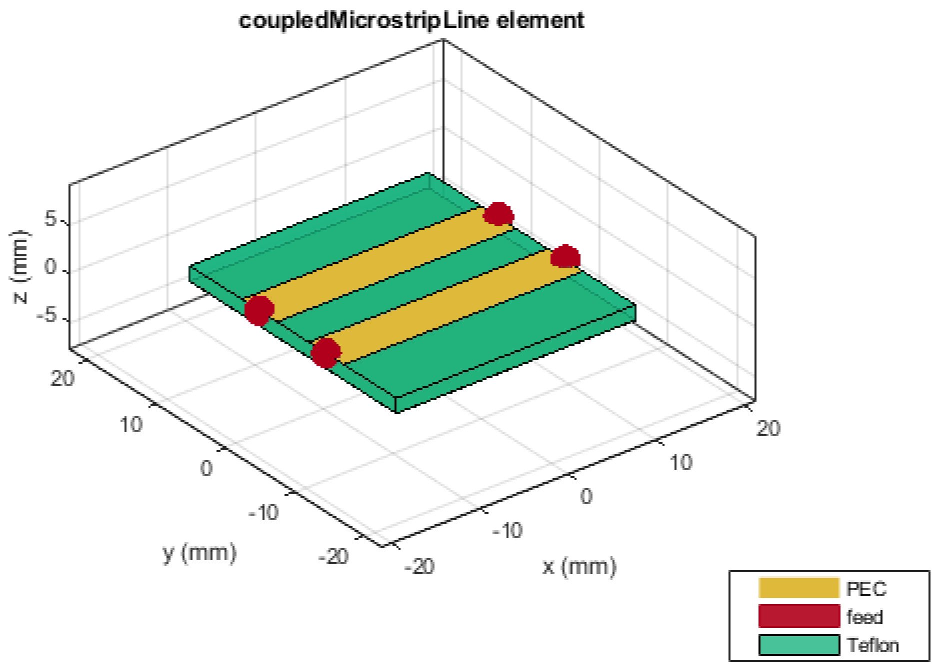
Default Coupled Microstrip Line

Create and view a default coupled microstrip line.

```
coupledmicrostripline = coupledMicrostripLine
```

```
coupledmicrostripline = 
  coupledMicrostripLine with properties:

             Length: 0.0271
              Width: 0.0051
            Spacing: 0.0046
             Height: 0.0016
    GroundPlaneWidth: 0.0300
          Substrate: [1x1 dielectric]
          Conductor: [1x1 metal]
```

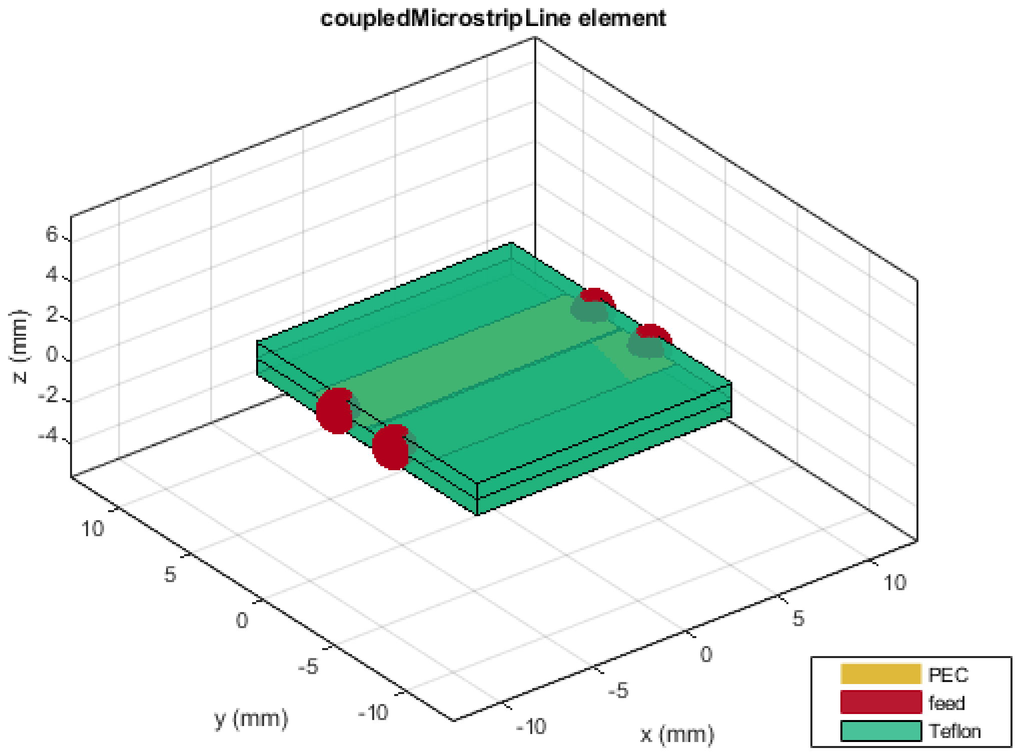
```
show(coupledmicrostripline)
```



Multilayer Coupled Microstrip Line

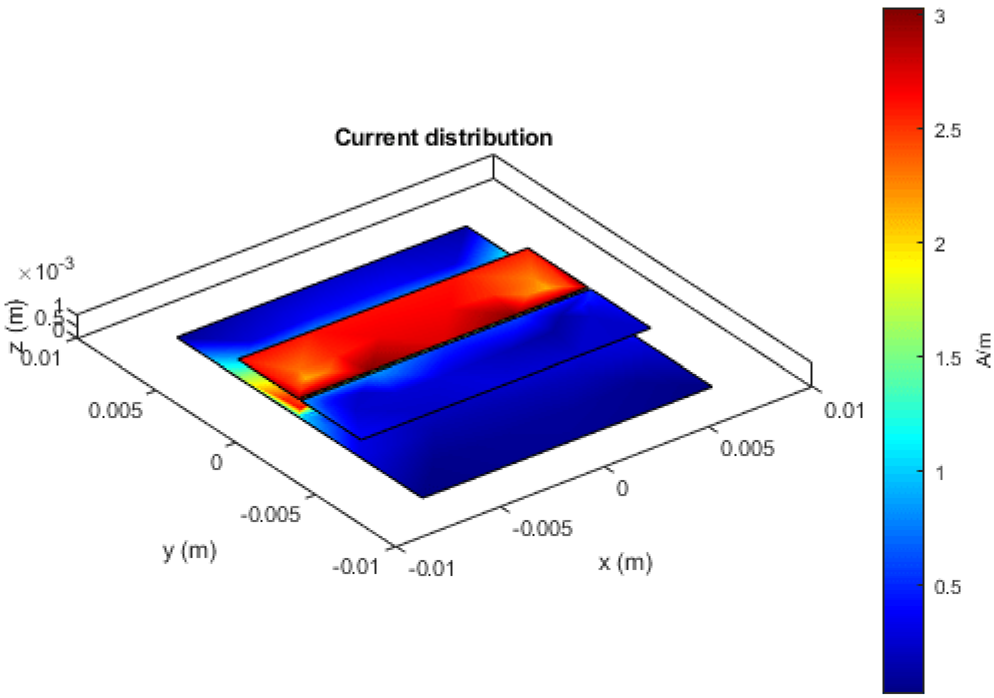
Design and view a coupled microstrip line at the interface of a multilayer dielectric.

```
coupledmicrostripline = design(coupledMicrostripLine,4e9,Z0e=75,Z0o=36);
coupledmicrostripline.Substrate = dielectric(Name=["Teflon","Teflon"],EpsilonR=[2.1 2.1], ...
    LossTangent=[0 0],Thickness=[0.8e-3 0.8e-3]);
coupledmicrostripline.Height = 0.8e-3;
show(coupledmicrostripline)
```

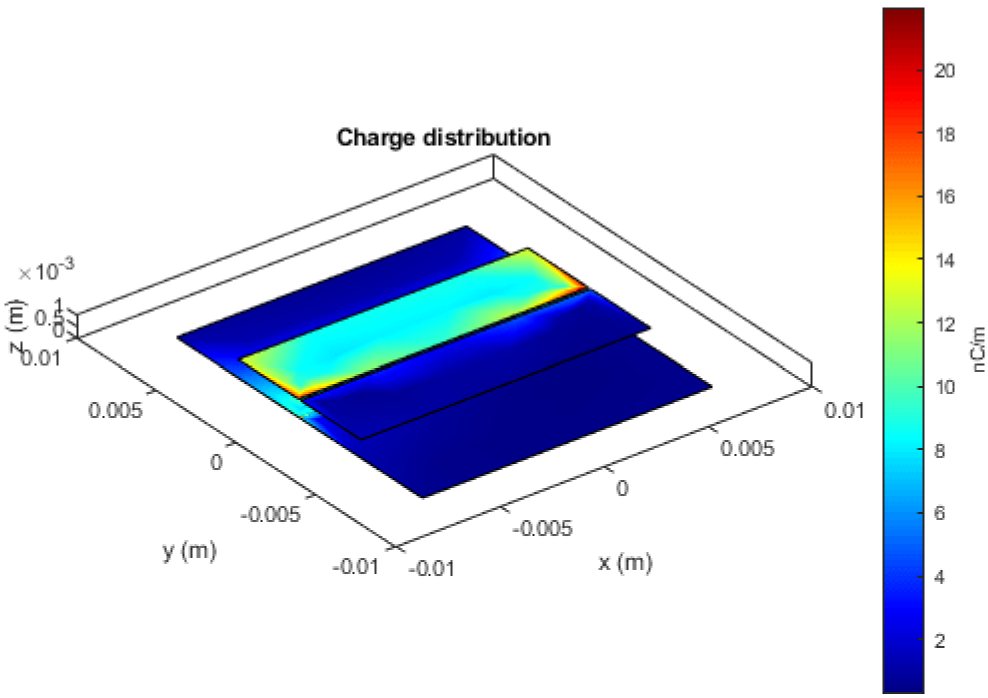


Plot the current and charge distribution on the transmission line.

```
current(coupledmicrostripline,4e9)
```




```
figure
charge(coupledmicrostripline,4e9)
```



More About

Parametric Analysis

Use the `design` function to change the even impedance (Z_{oe}) and the odd impedance (Z_{oo}) of the coupled microstrip line.

- Increasing the difference between the even impedance (Z_{oe}) and the odd impedance (Z_{oo}) decreases the distance between the lines.
- Increasing the difference between Z_{oe} and Z_{oo} increases the power at the coupled ports.
- The impedance of the coupled lines Z_o is the geometric mean of Z_{oe} and Z_{oo}. If the even and odd impedance values do not satisfy this condition, then calculate the S-parameters using Z_o to get a proper match.

References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

[2] "Microwaves101 | Coupled Line Couplers." Accessed July 7, 2021. https://www.microwaves101.com/encyclopedias/coupled-line-couplers.

See Also

`coupledStripLine` | `coplanarWaveguide` | `stripLine` | `microstripLine`

Introduced in R2021b

coupledStripLine

Create coupled transmission line in stripline form

Description

Use the `coupledStripLine` object to create a coupled transmission line in a stripline form. Coupled striplines are used to connect different RF components like couplers and dividers within the PCB board.

Types and applications of stripline routing techniques:

- `Symmetric or Plain` — Transmission lines routed on the internal layers.
- `Asymmetric` — Transmission lines not centrally embedded in the ground plane. Asymmetric striplines are placed close to one of the ground planes. When routing signals the closer ground plane is used as a reference to ensure stronger return signal.
- `Edge-Coupled` — Routing technique used for differential pairs. It has the same structure as the symmetric stripline routing but with trace spacing for the differential pair.
- `Suspended` — Printed stripline with a strip conductor centered between two parallel ground planes and placed on a dielectric surface. The substrate is suspended in a metal closer. The major portion of the EM field is confined to the air gaps between dielectric substrates and ground planes.

Creation

Syntax

```
sline = coupledStripLine
sline = coupledStripLine('propety1',value1,'property2',value2...)
```

Description

`sline = coupledStripLine` creates a default coupled transmission line in the strip line form. The default properties are for a resonant frequency of 2.5 GHz along the X-Y plane.

`sline = coupledStripLine('propety1',value1,'property2',value2...)` sets properties using one or more name value pair arguments. For example, `sline = coupledStripLine('Length',0.0300)` creates a coupled strip line of length 0.0300 meters. Properties not specified retains their default values.

Properties

`Length` — Length of coupled strip line
`0.0202` (default) | positive scalar

Length of the coupled strip line in meters, specified as a positive scalar.

Example: `strip = coupledStripLine('Length',0.0300)`

Data Types: double

Width — Width of coupled strip line
0.0026 (default) | positive scalar

Width of the coupled strip line in meters, specified as a positive scalar.

Example: strip = coupledStripLine('Width',0.0037)

Data Types: double

Spacing — Spacing between coupled lines
9.7000e-4 (default) | positive scalar

Spacing between the coupled lines in meters, specified as a positive scalar.

Example: strip = coupledStripLine('Spacing',0.00037)

Data Types: double

Height — Height from ground plane to coupled strip line
0.0016 (default) | positive scalar

Height from the ground plane to the coupled strip line in meters, specified as a positive scalar.

Example: strip = coupledStripLine('Height',9.000e-04)

Data Types: double

GroundPlaneWidth — Width of ground plane
0.0156 (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: strip = coupledStripLine('GroundPlaneWidth',0.0350)

Data Types: double

Substrate — Type of dielectric material
'Teflon' (default) | dielectric object

Type of dielectric material used as a substrate, specified as a dielectric object. For more information see dielectric. The default dielectric material Teflon has an EpsilonR of 2.2, la oss tangent of 0.03, and a thickness of 0.0032.

Example: d = dielectric('FR4'); strip = coupledStripLine('Substrate',d)

Data Types: string | char

Conductor — Type of metal used for conducting layers
'PEC' (default) | metal object

Type of metal used for the conducting layers, specified as a metal object. For more information see metal.

Example: m = metal('PEC'); strip = coupledSripLine('Conductor',m)

Data Types: string | char

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design symmetric coupled strip transmission line around given frequency
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
mesh	Change and view mesh properties of metal or dielectric in PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

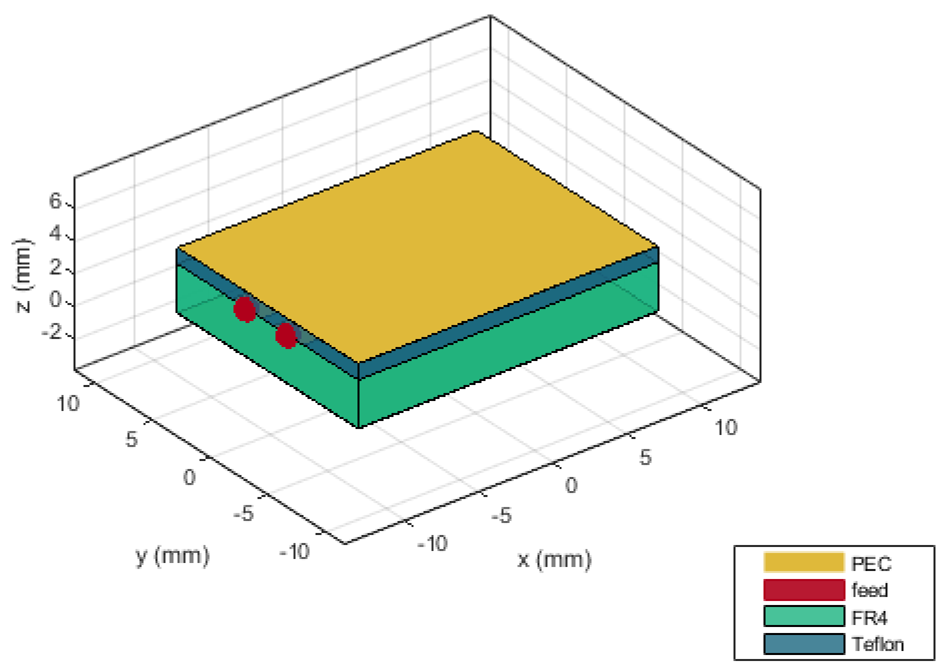
Examples

???

Multi-Layer Coupled Strip Line

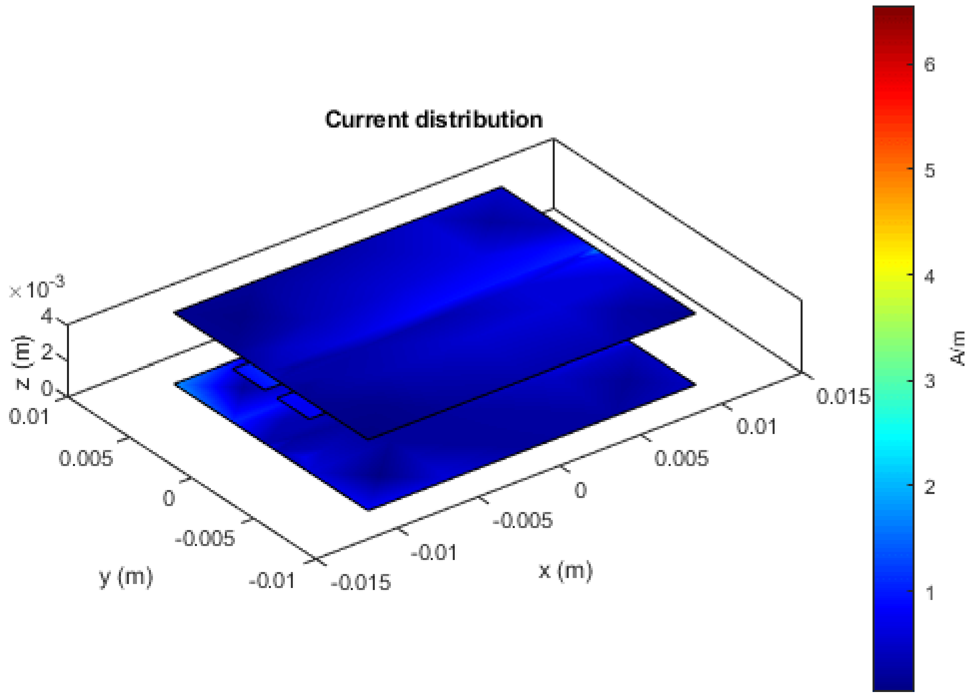
Create and view a coupled strip line at the interface of a multi-layered dielectric.

```
sub = dielectric('FR4','Teflon');
sub.Thickness = [0.003 0.001];
coupledstripline = coupledStripLine('Height',0.003,'Substrate',sub);
show(coupledstripline);
```

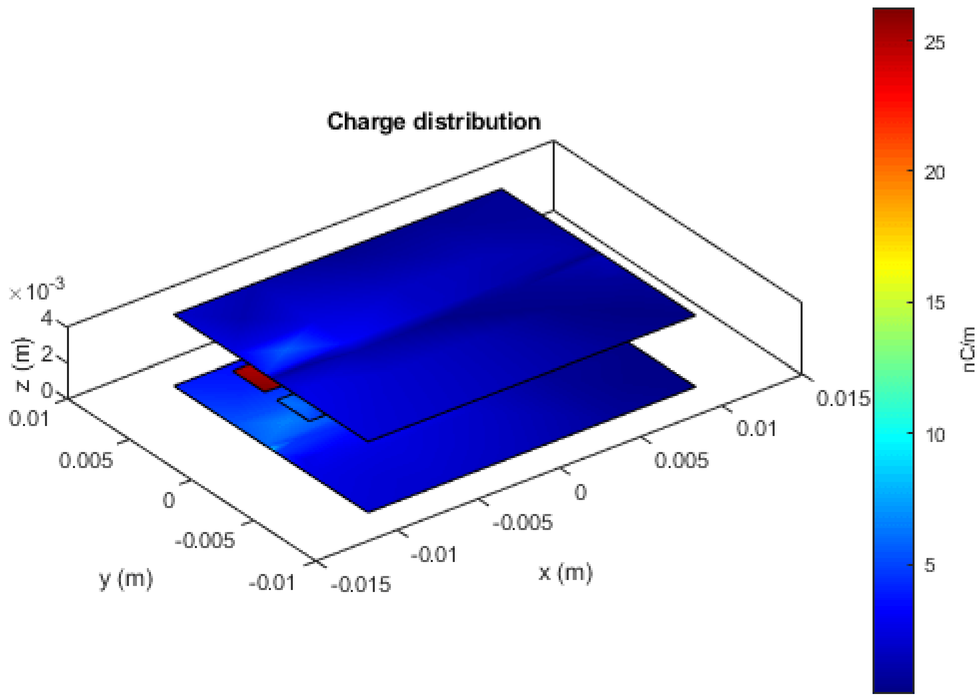


Plot the charge and current distribution on this transmission line.

```
current(coupledstripline,2.5e9)
```




```
figure;
charge(coupledstripline,2.5e9)
```

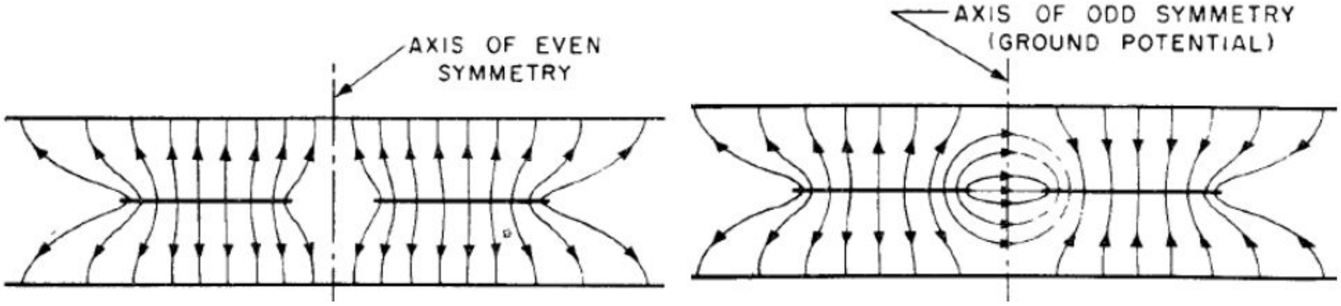


More About

Parametric Analysis Guidelines

- Increase in the difference between the even-odd mode impedance , decreases the spacing between the coupled lines.
- Increase in Z_{oe} will increase the coupled power.
- Increase in the Width and the Spacing properties decreases the characteristic impedance.
- If s tends to infinity, both odd and even mode impedances reduce to the characteristic impedance of an isolated strip between ground planes.
- The characteristic impedance of the line is the geometric mean of the even mode (Z_{oe}) and the odd mode impedances (Z_{oo}). For example, to design a line of 50 ohms for the given even mode impedance, select the odd mode impedance accordingly.
- Increase the operating frequency of the coupled strip line by decreasing the dimensions of the line.

Design Function



The figure shows transverse field distributions for two fundamental transverse electro-magnetic (TEM) modes. These modes exists on a pair of parallel conducting strips between parallel ground planes.

- Even coupled strip mode — In the first figure, strips are at the same potential and carry equal currents in the same direction. This mode is called the even coupled strip mode because of the even symmetry of the electric field about the vertical axis.
- Odd coupled strip mode — In the second figure, strips are at equal but opposite potentials and carry equal currents in the opposite direction. This mode is called odd coupled strip mode because of the odd symmetry of the electric field.

The figures also show that the capacitance per strip to ground is less for the even case and more for the odd case. The characteristic impedances of the two modes are unequal, being greater for the even mode than for the odd mode.

References

[1] Pozar, *Microwave Engineering / David M. Pozar*, University of Massachusetts at Amherst.

[2] Cohn, S.B. “Shielded Coupled-Strip Transmission Line.” *IEEE Transactions on Microwave Theory and Techniques 3*, no. 5 (October 1955): 29–38. https://doi.org/10.1109/TMTT.1955.1124973.

See Also

`coplanarWaveguide` | `stripLine` | `microstripLine` | `coupledMicrostripLine`

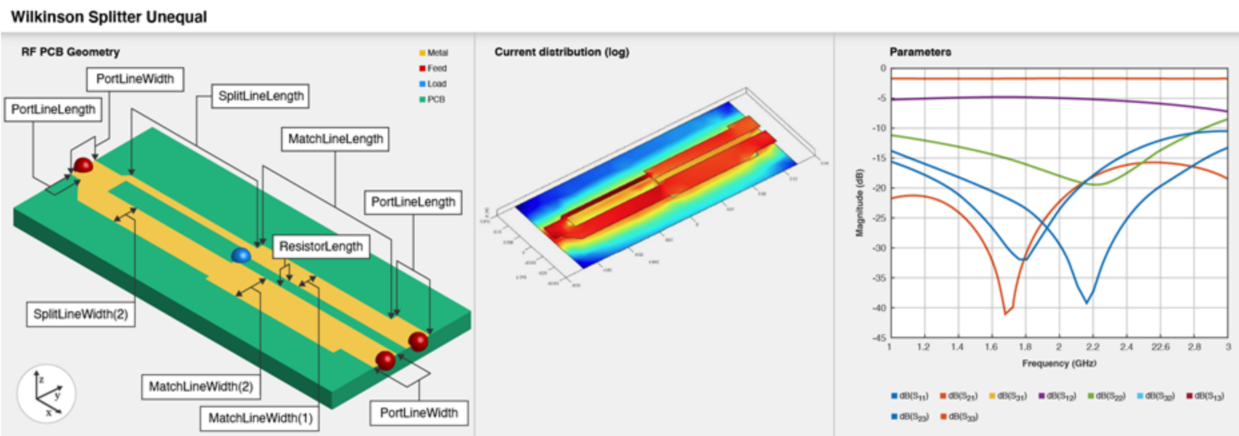
Introduced in R2021b

wilkinsonSplitterUnequal

Create unequal Wilkinson splitter

Description

Use the `wilkinsonSplitterUnequal` object to create an unequal Wilkinson power splitter. You can use the unequal Wilkinson splitter to divide power unequally between two output ports. Unequal splitters are also used to feed power to antenna arrays for beam shaping.



Creation

Syntax

```
splitter = wilkinsonSplitterUnequal
splitter = wilkinsonSplitterUnequal(Name=Value)
```

Description

`splitter = wilkinsonSplitterUnequal` creates an unequal Wilkinson splitter with a Teflon substrate. The default properties are for a resonating frequency of 1 GHz.

`splitter = wilkinsonSplitterUnequal(Name=Value)` sets “Properties” on page 1-127 using one or more name-value arguments. For example, `wilkinsonSplitterUnequal(PortLineLength=0.0300)` creates a Wilkinson splitter with an input and output line length of 0.0300 meters. Properties not specified retain their default values.

Properties

`PortLineLength` — Length of input and output line
`0.0070` (default) | positive scalar

Length of the input and the output line in meters, specified as a positive scalar.

Example: `splitter = wilkinsonSplitterUnequal(PortLineLength=0.0070)`

Data Types: `double`

`PortLineWidth` — Width of input and output line
`0.0051` (default) | positive scalar

Width of the input and the output line in meters, specified as a positive scalar.

Example: `splitter = wilkinsonSplitterUnequal(PortLineWidth=0.0070)`

Data Types: `double`

`SplitLineLength` — Length of 70-ohm line
`0.0279` (default) | positive scalar

Length of the 70-ohm line in meters, specified as a positive scalar. The typical length of a Wilkinson splitter is λ/4.

Example: `splitter = wilkinsonSplitterUnequal(SplitLineLength=0.0570)`

Data Types: `double`

`SplitLineWidth` — Width of 70-ohm line
`[0.0014 0.0049]` (default) | two-element vector

Width of the 70-ohm line in meters, specified as a two-element vector of positive elements.

Example: `splitter = wilkinsonSplitterUnequal(SplitLineWidth=[0.00780 0.00890])`

Data Types: `double`

`MatchLineLength` — Length of output matching line
`0.0277` (default) | positive scalar

Length of the output matching line in meters, specified as a positive scalar.

Example: `splitter = wilkinsonSplitterUnequal(MatchLineLength=0.0780)`

Data Types: `double`

`MatchLineWidth` — Width of output matching line
`[0.0039 0.0066]` (default) | two-element vector

Width of the output matching line in meters, specified as a two-element vector of positive elements.

Example: `splitter = wilkinsonSplitterUnequal(MatchLineWidth=[0.0049 0.0076])`

Data Types: `double`

`ResistorLength` — Length of resistor in meters
`0.0020` (default) | positive scalar

Length of the resistor in meters, specified as a positive scalar.

Example: `splitter = wilkinsonSplitterUnequal(ResistorSLength=0.0050)`

Data Types: `double`

`Resistance` — Resistance value
`106` (default) | positive scalar

Resistance value in ohms, specified as a positive scalar.

Example: `splitter = wilkinsonSplitterUnequal(Resistance=50)`

Data Types: `double`

`Height` — Height of Wilkinson splitter from ground plane
`0.0016` (default) | positive scalar

Height of the Wilkinson splitter from the ground plane in meters, specified as a positive scalar.

Example: `splitter = wilkinsonSplitterUnequal(Height=0.0076)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane in meters
`0.0300` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `splitter = wilkinsonSplitterUnequal(GroundPlaneWidth=0.046)`

Example: double

`Substrate` — Type of dielectric material
`'Teflon'` (default) | `dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The thickness of the default dielectric material Teflon is `0.0016` m or the same value as the `Height` property.

Example: `d = dielectric("FR4"); splitter = wilkinsonSplitterUnequal(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'Copper'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("PEC"); splitter = wilkinsonSplitterUnequal(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design unequal Wilkinson splitter around specified frequency
feedCurrent	Calculate current at feed port
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

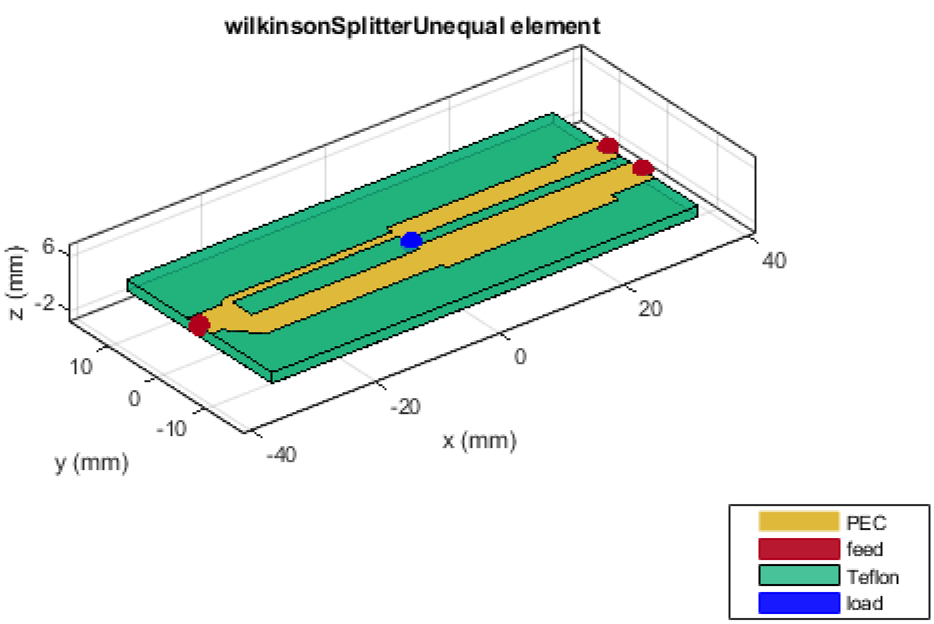
Create Default Unequal Wilkinson Splitter

Create and view a default unequal Wilkinson splitter.

```
splitter = wilkinsonSplitterUnequal
```

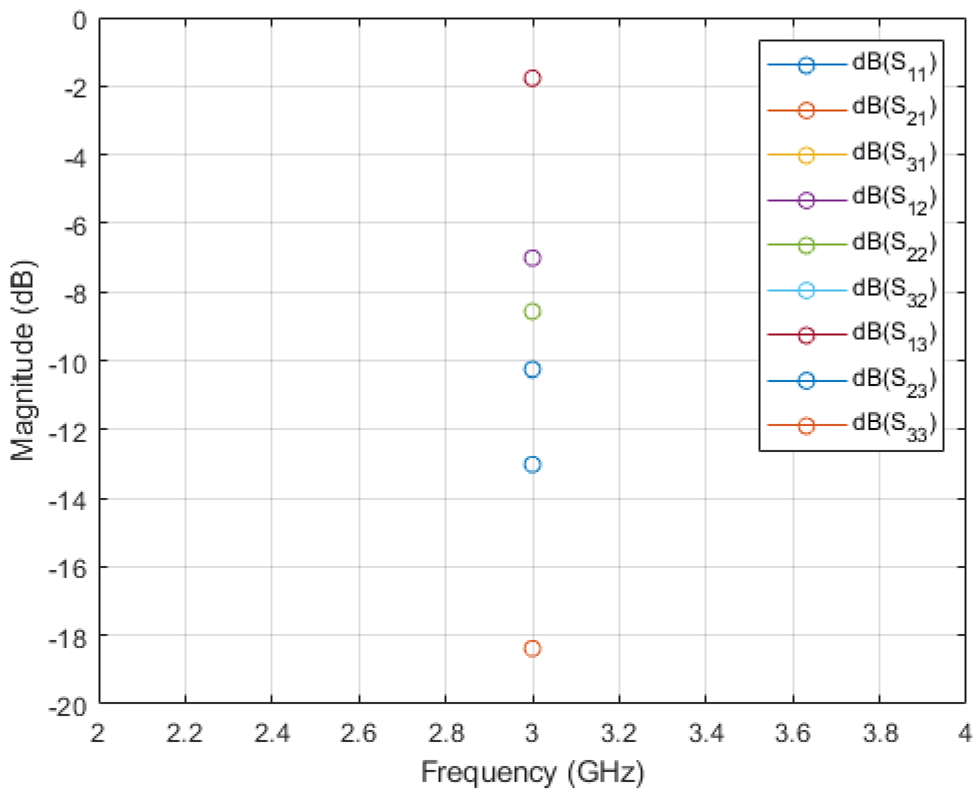
```
splitter = 
  wilkinsonSplitterUnequal with properties:

      PortLineLength: 0.0070
       PortLineWidth: 0.0051
     SplitLineLength: 0.0279
      SplitLineWidth: [0.0014 0.0049]
     MatchLineLength: 0.0277
      MatchLineWidth: [0.0039 0.0066]
      ResistorLength: 0.0020
          Resistance: 106
              Height: 0.0016
    GroundPlaneWidth: 0.0300
           Substrate: [1x1 dielectric]
           Conductor: [1x1 metal]
```

```
show(splitter)
```



Calculate and plot the S-parameters of the splitter at 3 GHz.

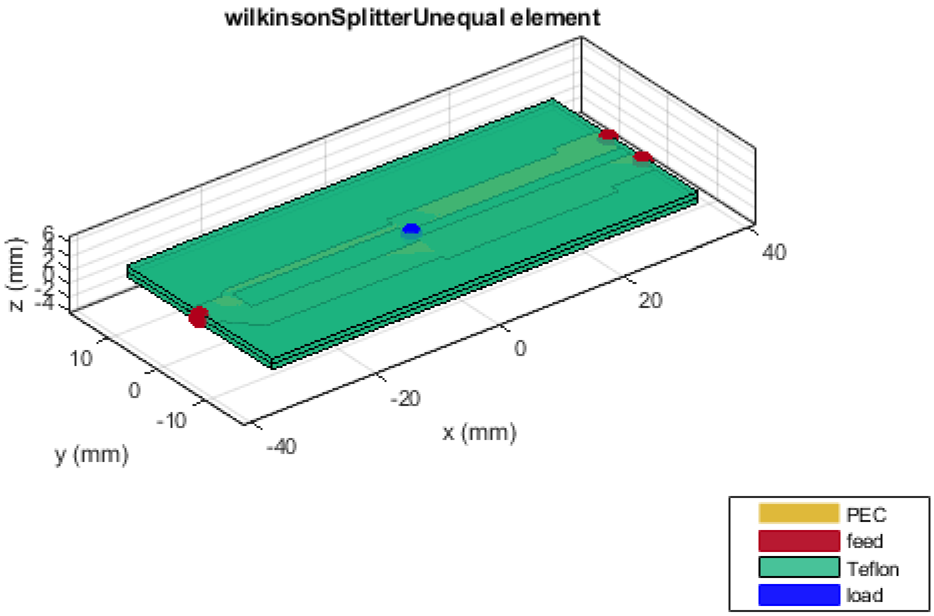
```
spar=sparameters(splitter,3e9);
figure
rfplot(spar);
```



Create Multilayer Unequal Wilkinson Splitter

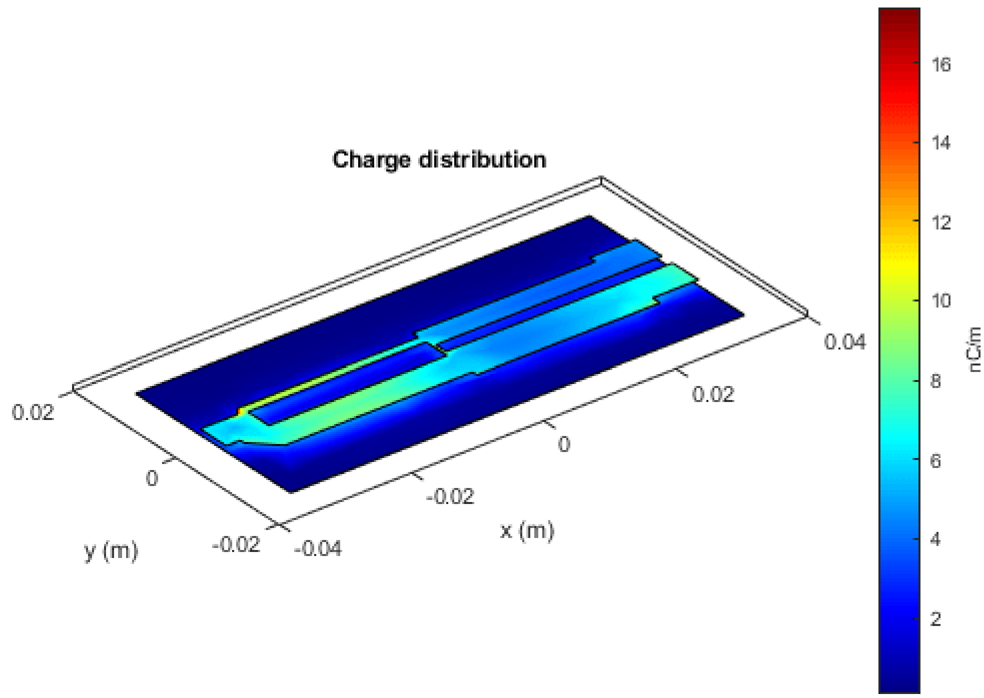
Create and view a multilayer unequal Wilkinson splitter.

```
sub =  dielectric(Name=["Teflon","Teflon"],EpsilonR=[2.1 2.1], ...
    LossTangent=[0 0],Thickness=[0.8e-3 0.8e-3]);
unsplitter = wilkinsonSplitterUnequal(Height=0.8e-3,Substrate=sub);
show(unsplitter)
```

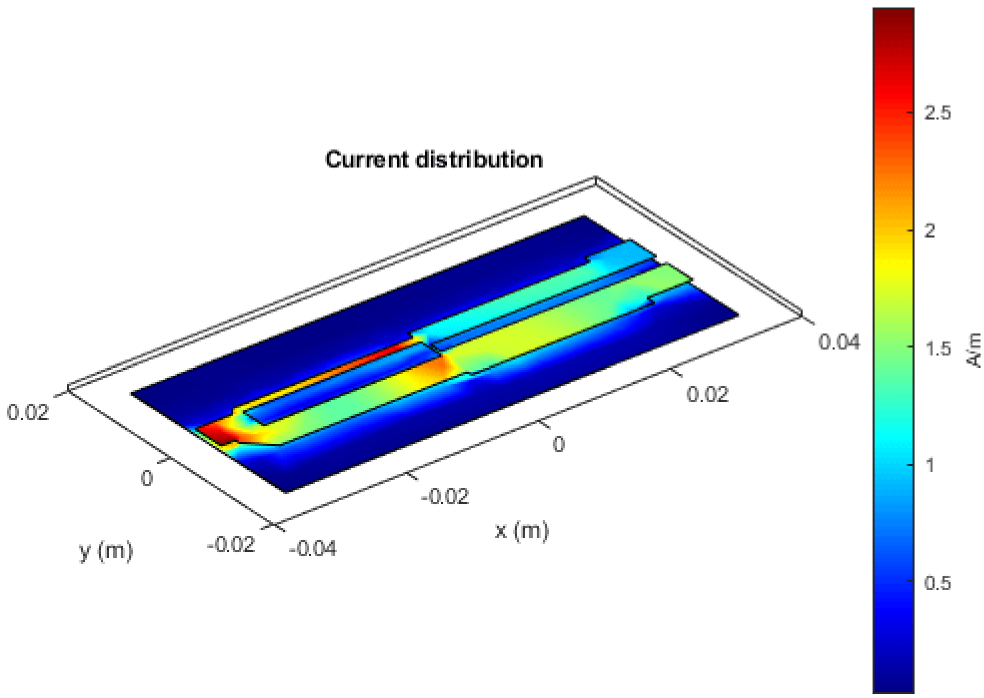


Plot the charge and current on this splitter at 3 GHz.

```
figure
charge(unsplitter,3e9)
```




```
figure
current(unsplitter,3e9)
```



References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

See Also

`wilkinsonSplitter`

Introduced in R2021b

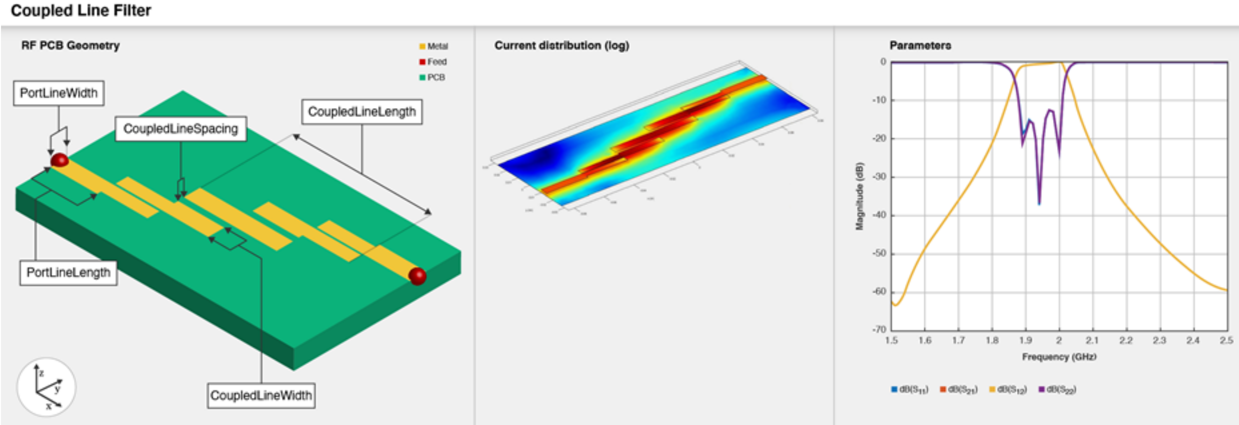
filterCoupledLine

Create coupled line filter in microstrip form

Description

Use the `filterCoupledLine` object to create a coupled line filter in microstrip form. The filter structure consists of open-circuited coupled microstrip lines. You can control the bandwidth of the filter by varying the filter order, width, and distance between the coupled lines.

Coupled Line Filter



Creation

Syntax

```
filter = filterCoupledLine
filter = filterCoupledLine(Name=Value)
```

Description

`filter = filterCoupledLine` creates a default coupled line filter using a Teflon substrate. The default passband of the filter is centered around 2 GHz.

`filter = filterCoupledLine(Name=Value)` sets "Properties" on page 1-135 using one or more name-value arguments. For example, `filterCoupledLine(FilterOrder=5)` creates a fifth-order coupled line filter. Properties not specified retain their default values.

Properties

`FilterOrder` — Filter order
3 (default) | positive scalar

Filter order, specified as a positive scalar.

Example: `filter = filterCoupledLine(FilterOrder=5)`

Data Types: `double`

`PortLineLength` — Length of input and output lines
`0.0279` (default) | positive scalar

Length of the input and output lines in meters, specified as a positive scalar.

Example: `filter = filterCoupledLine(PortLineLength=0.0553)`

Data Types: `double`

`PortLineWidth` — Width of input and output lines
`0.0051` (default) | positive scalar

Width of the input and output lines in meters, specified as a positive scalar.

Example: `filter = filterCoupledLine(PortLineWidth=0.0087)`

Data Types: `double`

`CoupledLineLength` — Lengths of coupled lines
`[0.0279 0.0279 0.0279 0.0279]` (default) | vector

Lengths of the coupled lines in meters, specified as a vector of positive elements.

Example: `filter = filterCoupledLine(CoupledLineLength=[0.0553 0.0553 0.0553 0.0553])`

Data Types: `double`

`CoupledLineWidth` — Widths of coupled lines
`[0.0036 0.0049 0.0049 0.0036]` (default) | vector

Widths of the coupled lines in meters, specified as a vector of positive elements.

Example: `filter = filterCoupledLine(CoupledLineWidth=[0.0046 0.0059 0.0059 0.0046])`

Data Types: `double`

`CoupledLineSpacing` — Distance between coupled lines
`[1.8270e-04 0.0019 0.0019 1.8270e-04]` (default) | vector

Distance between the coupled lines in meters, specified as a vector of positive elements.

Example: `filter = filterCoupledLine(CoupledLineSpacing=[2.8270e-04 0.0020 0.0020 2.8270e-04])`

Data Types: `double`

`Height` — Height of coupled line filter from ground plane
`0.0016` (default) | positive scalar

Height of the coupled line filter from the ground plane in meters, specified as a positive scalar. For multilayer dielectrics, use the `Height` property to create the filter between the two dielectric layers.

Example: `filter = filterCoupledLine(Height=0.0028)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane
`0.0551` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `filter = filterCoupledLine(GroundPlaneWidth=0.0048)`

Data Types: `double`

`Substrate` — Type of dielectric material
`'Teflon'` (default) | `dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The thickness of the default dielectric material Teflon is `0.0016` m.

Example: `d = dielectric("FR4"); filter = filterCoupledLine(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'PEC'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("Copper"); filter = filterCoupledLine(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design coupled line filter around specified frequency
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

Create Default Coupled Line Filter

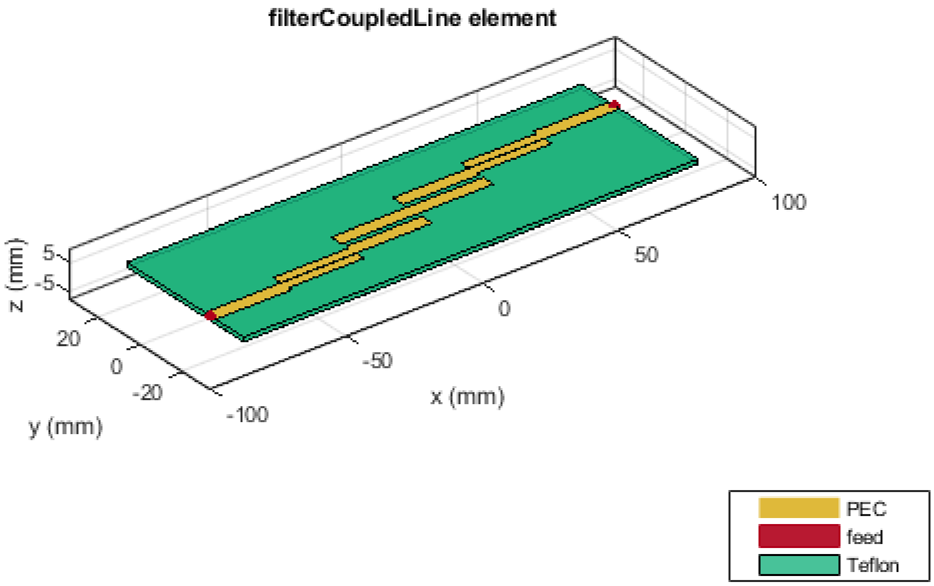
Create and view a default coupled line filter.

```
coupledfilter = filterCoupledLine
```

```
coupledfilter = 
  filterCoupledLine with properties:

           FilterOrder: 3
        PortLineLength: 0.0279
         PortLineWidth: 0.0051
     CoupledLineLength: [0.0279 0.0279 0.0279 0.0279]
      CoupledLineWidth: [0.0036 0.0049 0.0049 0.0036]
```

```
    CoupledLineSpacing: [1.8270e-04 0.0019 0.0019 1.8270e-04]
                Height: 0.0016
      GroundPlaneWidth: 0.0551
             Substrate: [1x1 dielectric]
             Conductor: [1x1 metal]
```

```
show(coupledfilter)
```



Coupled Line Filter at Specified Frequency

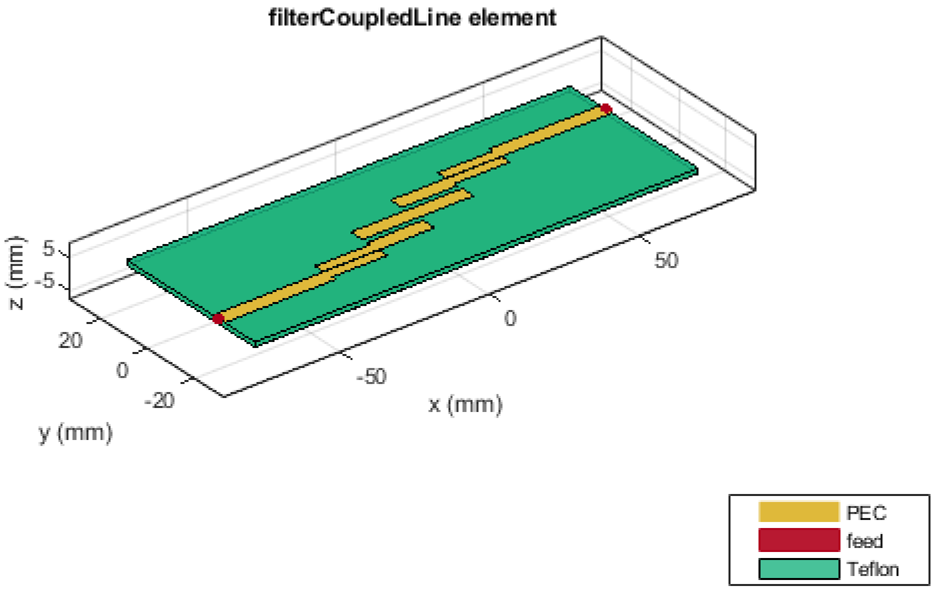
Create and view a coupled line filter at 3 GHz.

```
coupledfilter = design(filterCoupledLine,3e9)
```

```
coupledfilter = 
  filterCoupledLine with properties:

           FilterOrder: 3
        PortLineLength: 0.0372
         PortLineWidth: 0.0051
     CoupledLineLength: [0.0186 0.0186 0.0186 0.0186]
      CoupledLineWidth: [0.0036 0.0049 0.0049 0.0036]
    CoupledLineSpacing: [1.8270e-04 0.0019 0.0019 1.8270e-04]
                Height: 0.0016
```

```
      GroundPlaneWidth: 0.0551
             Substrate: [1x1 dielectric]
             Conductor: [1x1 metal]
```

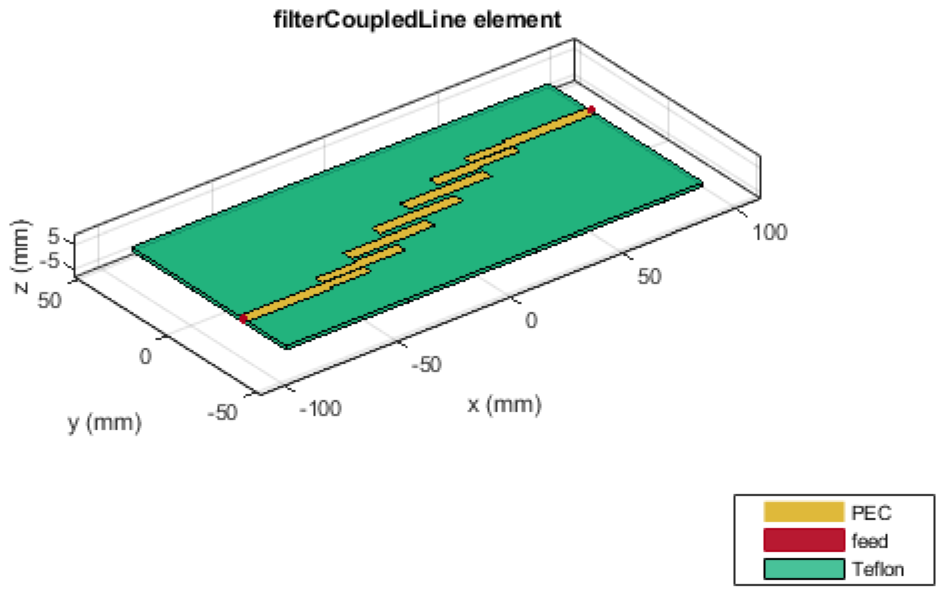
```
show(coupledfilter)
```



Fifth-Order Coupled Line Chebyshev Filter

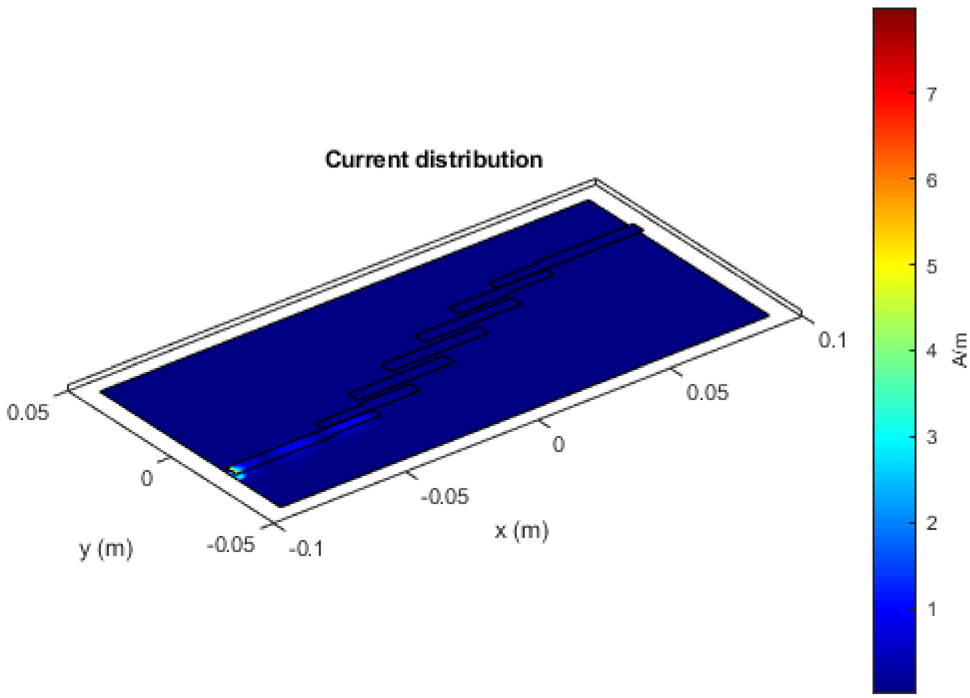
Design and view a fifth-order coupled-line Chebyshev filter at 3 GHz with a ripple factor of 0.5 dB.

```
coupledfilter = filterCoupledLine(FilterOrder=5);
coupledfilter = design(coupledfilter,3e9,FilterType="Chebyshev",RippleFactor=0.5);
show(coupledfilter)
```

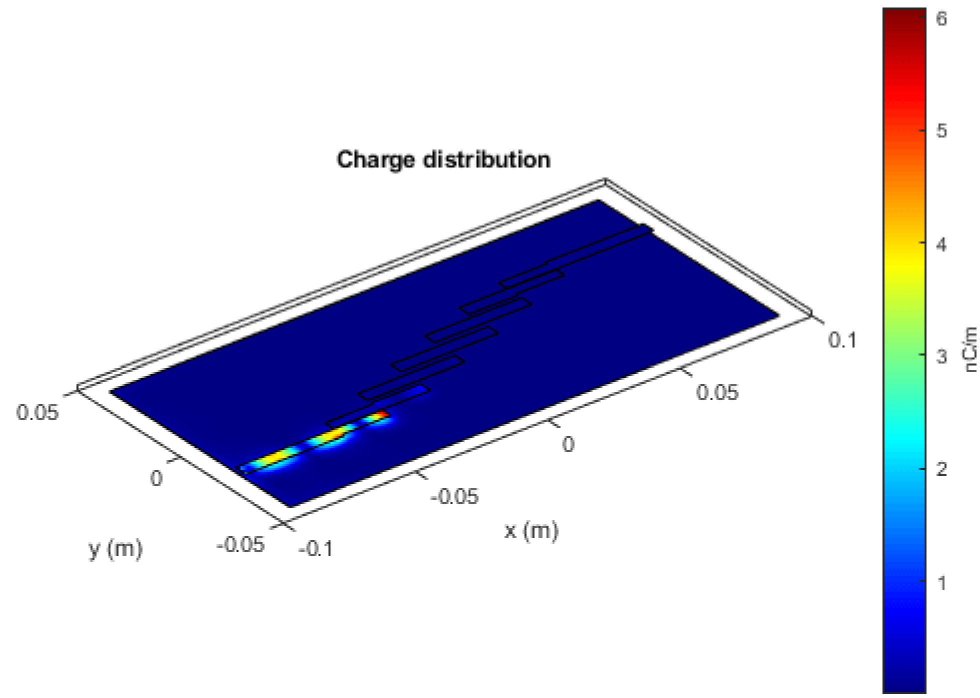


Plot the current and charge distribution of this filter at 5 GHz.

```
figure
current(coupledfilter,5e9)
```




```
figure
charge(coupledfilter,5e9)
```



References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

[2] Ragani, Taoufik, N. Amar Touhami, and M. Agoutane. "Designing a Microstrip Coupled Line Bandpass Filter." *International Journal of Engineering & Technology* 2, no. 4 (September 6, 2013): 266. https://doi.org/10.14419/ijet.v2i4.1173.

See Also

`filterHairpin` | `filterStepImpedanceLowPass`

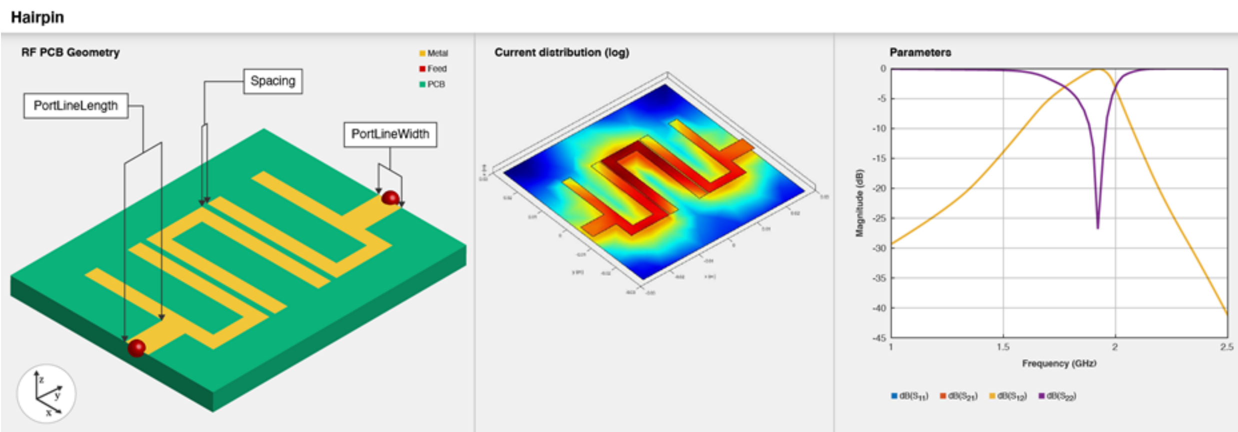
Introduced in R2021b

filterHairpin

Create hairpin filter in microstrip form

Description

Use the `filterHairpin` object to create a hairpin filter in microstrip form.



Hairpin filters are easy to fabricate. Design these filters at different frequencies by changing the roll-offs and the ripple factors. You can control the bandwidth of the filter by varying the filter order, width, and the distance between the coupled lines. There are two types of hairpin filters based on the feed: tapped-line input hairpin filters and coupled-line input hairpin filters.

Creation

Syntax

```
filter = filterHairpin
filter = filterHairpin(Name=Value)
```

Description

`filter = filterHairpin` creates a default hairpin filter using a Teflon substrate. The default passband of the filter is 2 GHz.

`filter = filterHairpin(Name=Value)` sets “Properties” on page 1-144 using one or more name-value arguments. For example, `filterHairpin(Resonator=ubendMitered)` creates a hairpin filter with a mitered u-bend element as the resonator. Properties not specified retain their default values

Properties

`Resonator` — Shape of hairpin element
`uBendRightAngle` (default) | `ubendMitered` | `ubendCurved`

Shape of the hairpin element, specified as either `uBendRightAngle`, `ubendMitered`, or `ubendCurved`.

Example: `filter = filterHairpin(Resonator=ubendCurved)`

Data Types: `char` | `string`

`FilterOrder` — Filter order
`3` (default) | positive scalar

Filter order, specified as a positive scalar.

Example: `filter = filterHairpin(FilterOrder=5)`

Data Types: `double`

`ResonatorOffset` — Y-offset of each resonator
`[0 0 0]` (default) | vector

Y-offset of each resonator in meters, specified as a vector.

Example: `filter = filterHairpin(ResonatorOffset=[0.02 0.02 0.02])`

Data Types: `double`

`Spacing` — Distance between hairpin bends
`[4.0000e-04 4.0000e-04]` (default) | vector

Distance between the hairpin bends in meters, specified as a vector of positive elements.

Example: `filter = filterHairpin(Spacing=[0.02 0.02 0.02])`

Data Types: `double`

`PortLineLength` — Length of input and output lines
`0.0080` (default) | positive scalar

Length of the input and output lines in meters, specified as a positive scalar.

Example: `filter = filterHairpin(PortLineLength=0.0553)`

Data Types: `double`

`PortLineWidth` — Width of input and output lines
`0.0050` (default) | positive scalar

Width of the input and output lines in meters, specified as a positive scalar.

Example: `filter = filterHairpin(PortLineWidth=0.0087)`

Data Types: `double`

`FeedOffset` — Y-offset for input and output lines
`[-0.0055 -0.0055]` (default) | vector

Y-offset for the input and the output lines, specified as a vector.

Example: `filter = filterHairpin(FeedOffset=[-0.002 -0.002])`

Data Types: `double`

FeedType — Type of feed at input and output ports
`"Tapped"` (default) | `"Coupled"`

Type of feed at the input and output ports, specified as either `"Tapped"` or `"Coupled"`.

Example: `filter = filterHairpin(FeedType="Coupled")`

Data Types: `char` | `string`

`CoupledLineLength` — Length of coupled feed lines
`[0.0279 0.0279 0.0279 0.0279]` (default) | vector

Length of the coupled feed lines in meters, specified as a vector of positive elements.

Example: `filter = filterHairpin(CoupledLineLength=[0.0553 0.0553 0.0553 0.0553])`

Dependencies

To enable `CoupledLineLength`, set the `FeedType` property to `"Coupled"`.

Data Types: `double`

`CoupledLineWidth` — Width of coupled feed lines
`[0.0036 0.0049 0.0049 0.0036]` (default) | vector

Width of the coupled feed lines in meters, specified as a vector of positive elements.

Example: `filter = filterHairpin(CoupledLineWidth=[0.0046 0.0059 0.0059 0.0046])`

Dependencies

To enable `CoupledLineLength`, set the `FeedType` property to `"Coupled"`.

Data Types: `double`

`CoupledLineSpacing` — Distance between feed line and hairpin
`[1.8270e-04 0.0019 0.0019 1.8270e-04]` (default) | vector

Distance between the feed line and the hairpin in meters, specified as a vector.

Example: `filter = filterHairpin(CoupledLineSpacing=[2.8270e-04 0.0020 0.0020 2.8270e-04])`

Dependencies

To enable `CoupledLineLength`, set the `FeedType` property to `"Coupled"`.

Data Types: `double`

`Height` — Height of hairpin filter from ground plane
`0.0016` (default) | positive scalar

Height of the hairpin filter from the ground plane in meters, specified as a positive scalar. In the case of a multilayer substrate, you can use the `Height` property to create a hairpin filter where the two dielectrics interface.

Example: `filter = filterHairpin(Height=0.020)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane
`0.0567` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `filter = filterHairpin(GroundPlaneWidth=[0.0679)`

Data Types: `double`

`Substrate` — Type of dielectric material
`'Teflon'` (default) | `dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The thickness of the default dielectric material Teflon is `0.0016` m or the same value as the `Height` property.

Example: `d = dielectric("FR4"); filter = filterHairpin(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'PEC'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("Copper"); filter = filterHairpin(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design hairpin filter around specified frequency
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

Create Default Hairpin Filter

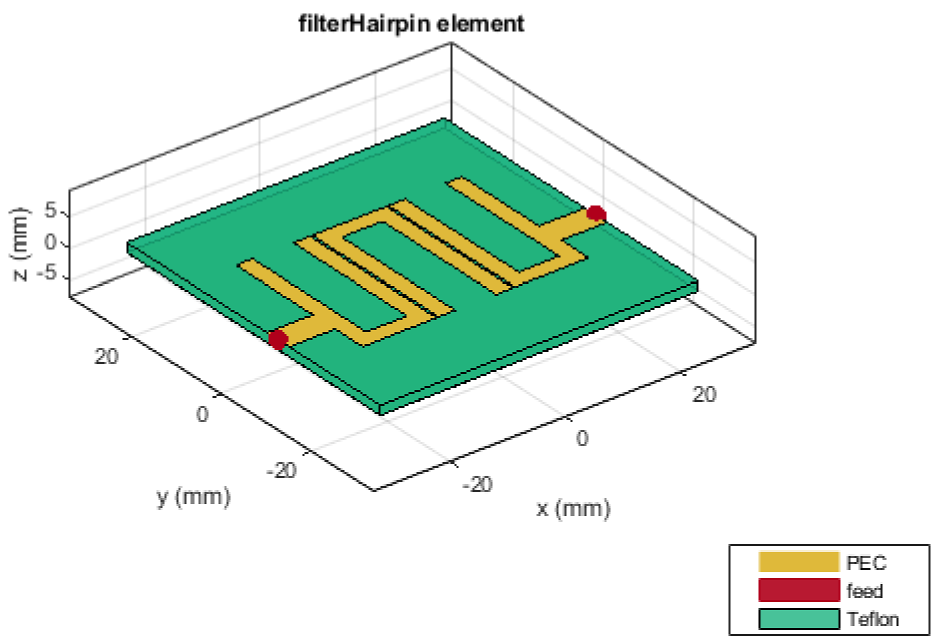
Create and view a default hairpin filter.

```
hairpinfilter = filterHairpin
```

```
hairpinfilter = 
  filterHairpin with properties:

           Resonator: [1x1 ubendRightAngle]
```

```
       FilterOrder: 3
   ResonatorOffset: [0 0 0]
           Spacing: [4.0000e-04 4.0000e-04]
    PortLineLength: 0.0080
     PortLineWidth: 0.0050
        FeedOffset: [-0.0055 -0.0055]
          FeedType: 'Tapped'
            Height: 0.0016
  GroundPlaneWidth: 0.0567
         Substrate: [1x1 dielectric]
         Conductor: [1x1 metal]
```

```
show(hairpinfilter)
```



Fifth-Order Coupled Hairpin Filter

Create and view a fifth-order coupled hairpin filter.

```
hairpinfilter = filterHairpin(FeedType="Coupled")
```

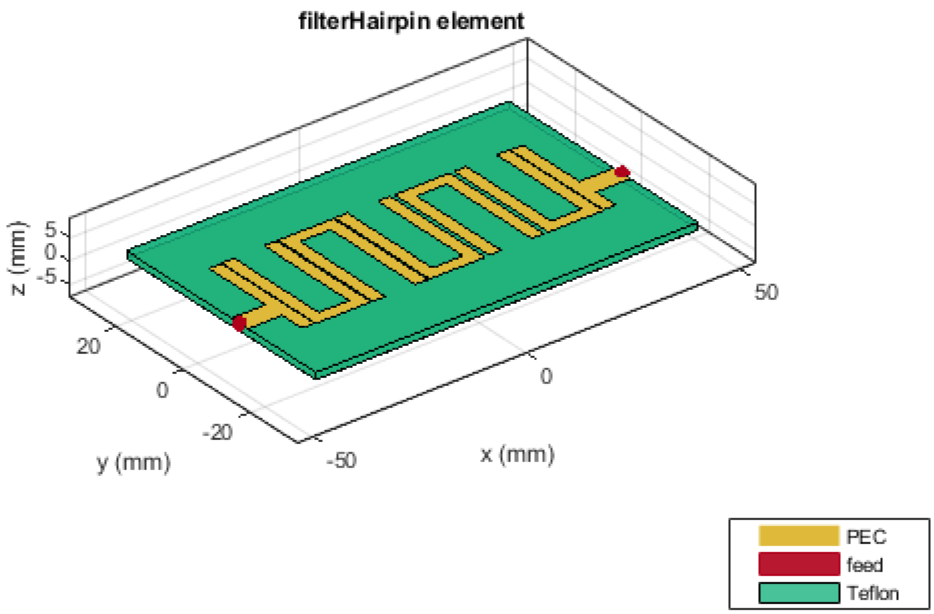
```
hairpinfilter = 
  filterHairpin with properties:

         Resonator: [1x1 ubendRightAngle]
```

```
           FilterOrder: 3
       ResonatorOffset: [0 0 0]
               Spacing: [4.0000e-04 4.0000e-04]
            FeedOffset: [-0.0055 -0.0055]
        PortLineLength: 0.0080
         PortLineWidth: 0.0050
              FeedType: 'Coupled'
     CoupledLineLength: 0.0279
      CoupledLineWidth: 0.0029
    CoupledLineSpacing: [1.8270e-04 1.8270e-04]
                Height: 0.0016
      GroundPlaneWidth: 0.0567
             Substrate: [1x1 dielectric]
             Conductor: [1x1 metal]
```

```
hairpinfilter.FilterOrder = 5
```

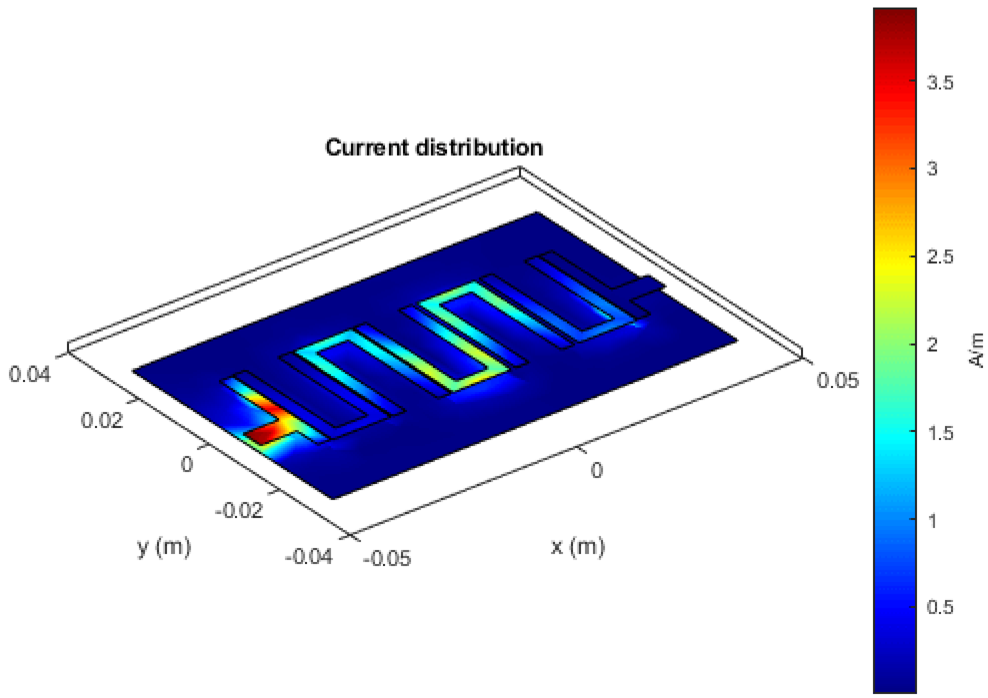
```
hairpinfilter = 
  filterHairpin with properties:

             Resonator: [1x1 ubendRightAngle]
           FilterOrder: 5
       ResonatorOffset: [0 0 0 0 0]
               Spacing: [4.0000e-04 4.0000e-04 1.0000e-03 1.0000e-03]
            FeedOffset: [-0.0055 -0.0055]
        PortLineLength: 0.0080
         PortLineWidth: 0.0050
              FeedType: 'Coupled'
     CoupledLineLength: 0.0279
      CoupledLineWidth: 0.0029
    CoupledLineSpacing: [1.8270e-04 1.8270e-04]
                Height: 0.0016
      GroundPlaneWidth: 0.0567
             Substrate: [1x1 dielectric]
             Conductor: [1x1 metal]
```

```
show(hairpinfilter)
```

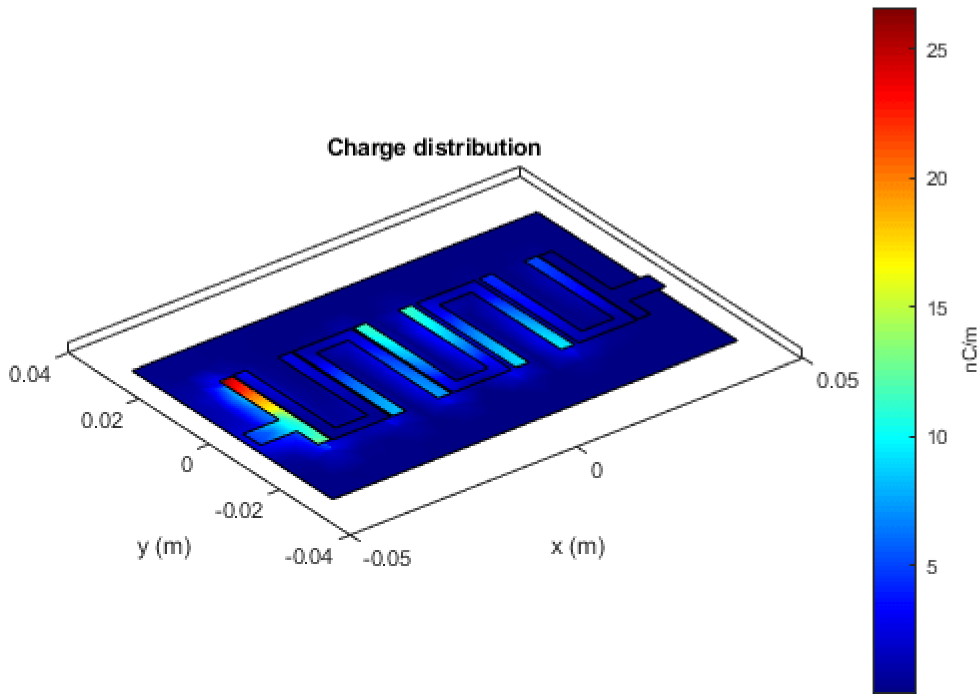


Plot the current and charge distribution of the filter at 2 GHZ.

```
figure
current(hairpinfilter,2e9)
```




```
figure
charge(hairpinfilter,2e9)
```



References

[1] Bankey, Kavita, and Abhinav Bhargava. “Design of Compact Microstrip Hairpin Multi Bandpass Filter.” *International Journal of Scientific Progress and Research* 34, no. 96 (2017): 66–69.

[2] Parikh, Nikunj, Pragya Katare, Ketan Kathal, Nandini Patel, and Gaurav Chaitanya. “Design and Analysis of Hairpin Micro-Strip Line Band Pass Filter.” *International Journal of Innovative Research in Electrical, Electronics, Instrumentation, and Control Engineering* 3 (April 2015). https://doi.org/10.17148/IJIREEICE.2015.3512.

See Also

`filterCoupledLine` | `filterStepImpedanceLowPass`

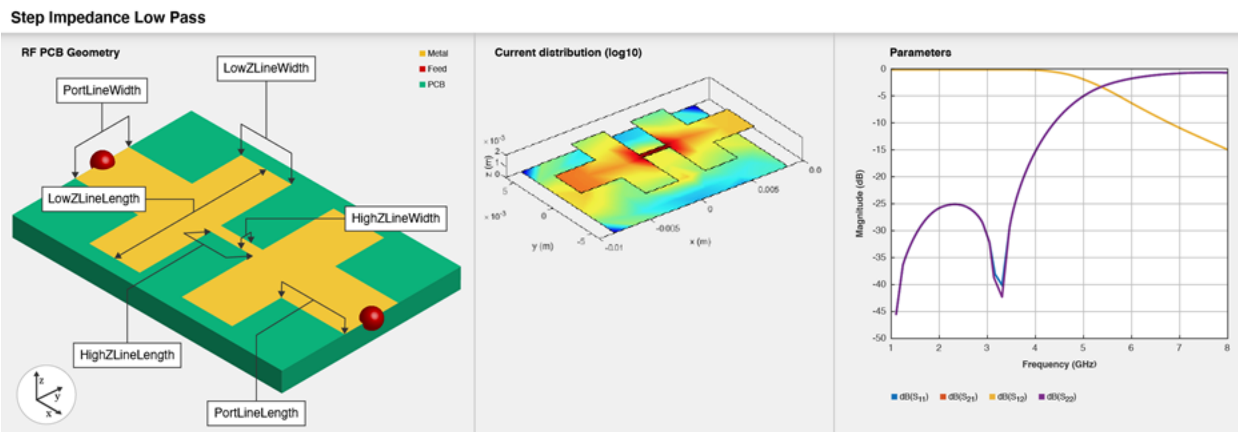
Introduced in R2021b

filterStepImpedanceLowPass

Create stepped impedance lowpass filter in microstrip form

Description

Use the `filterStepImpedanceLowPass` object to create a stepped impedance lowpass filter in microstrip form.



The stepped impedance lowpass microstrip filters have a cascaded structure of alternating high- and low-impedance transmission lines. These lines are considerably shorter in length than the design wavelength and act as semi-lumped elements. The high-impedance lines act as series inductors, and the low-impedance lines act as shunt capacitors. This filter structure realizes a Pi LC ladder type of a lowpass filter. You can control the impedance by adjusting the width of the strip. This filter is used in radar, satellite, and terrestrial communications and in electronic counter-measure applications.

Creation

Syntax

```
filter = filterStepImpedanceLowPass
filter = filterStepImpedanceLowPass(Name=Value)
```

Description

`filter = filterStepImpedanceLowPass` creates a default stepped impedance lowpass filter. The default filter dimensions are for a cutoff frequency 5 GHz.

`filter = filterStepImpedanceLowPass(Name=Value)` sets "Properties" on page 1-153 using one or more name-value arguments. For example, `filterStepImpedanceLowPass(FilterOrder=10)` creates a tenth-order stepped impedance lowpass filter. Properties not specified retain their default values

Properties

`FilterOrder` — Filter order

`3` (default) | positive scalar in the range [3,11]

Filter order, specified as a positive scalar. The minimum filter order you can specify is 3 and the maximum order is 11.

Example: `filter = filterStepImpedanceLowPass(FilterOrder=5)`

Data Types: `double`

`PortLineLength` — Length of input and output lines

`0.0034` (default) | positive scalar

Length of the input and output lines in meters, specified as a positive scalar.

Example: `filter = filterStepImpedanceLowPass(PortLineLength=0.0553)`

Data Types: `double`

`PortLineWidth` — Width of input and output lines

`0.0040` (default) | positive scalar

Width of the input and output lines in meters, specified as a positive scalar.

Example: `filter = filterStepImpedanceLowPass(PortLineWidth=0.0087)`

Data Types: `double`

`LowZLineWidth` — Width of low-impedance line

`0.0096` (default) | positive scalar

Width of the low-impedance line in meters, specified as a positive scalar.

Example: `filter = filterStepImpedanceLowPass(LowZLineWidth=0.0553)`

Data Types: `double`

`HighZLineWidth` — Width of high-impedance line

`5.0000e-04` (default) | positive scalar

Width of the high-impedance line in meters, specified as a positive scalar.

Example: `filter = filterStepImpedanceLowPass(HighZLineWidth=0.0553)`

Data Types: `double`

`LowZLineLength` — Length of low-impedance line

`0.0032` (default) | positive scalar

Length of the low-impedance line in meters, specified as a positive scalar.

Example: `filter = filterStepImpedanceLowPass(LowZLineLength=0.0553)`

Data Types: `double`

`HighZLineLength` — Length of high-impedance line

`0.0026` (default) | positive scalar

Length of the high-impedance line in meters, specified as a positive scalar.

Example: `filter = filterStepImpedanceLowPass(HighZLineWidth=0.0553)`

Data Types: `double`

`Height` — Height of filter from ground plane
`0.0016` (default) | positive scalar

Height of the filter from the ground plane in meters, specified as a positive scalar.

Example: `filter = filterStepImpedanceLowPass(Height=0.020)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane
`0.012` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `filter = filterStepImpedanceLowPass(GroundPlaneWidth=0.013)`

Data Types: `double`

`Substrate` — Type of dielectric material
`dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The default value is a `dielectric` object with these properties:

- `Name—{'CustomDielectric'}`
- `EpsilonR—3.7`
- `LossTangent—0.001`
- `Thickness—1.6e-3`

Example: `d = dielectric("FR4"); filter = filterStepImpedanceLowPass(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'PEC'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("Copper"); filter = filterStepImpedanceLowPass(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
design	Design stepped impedance low pass filter around desired cut-off frequency
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component

show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

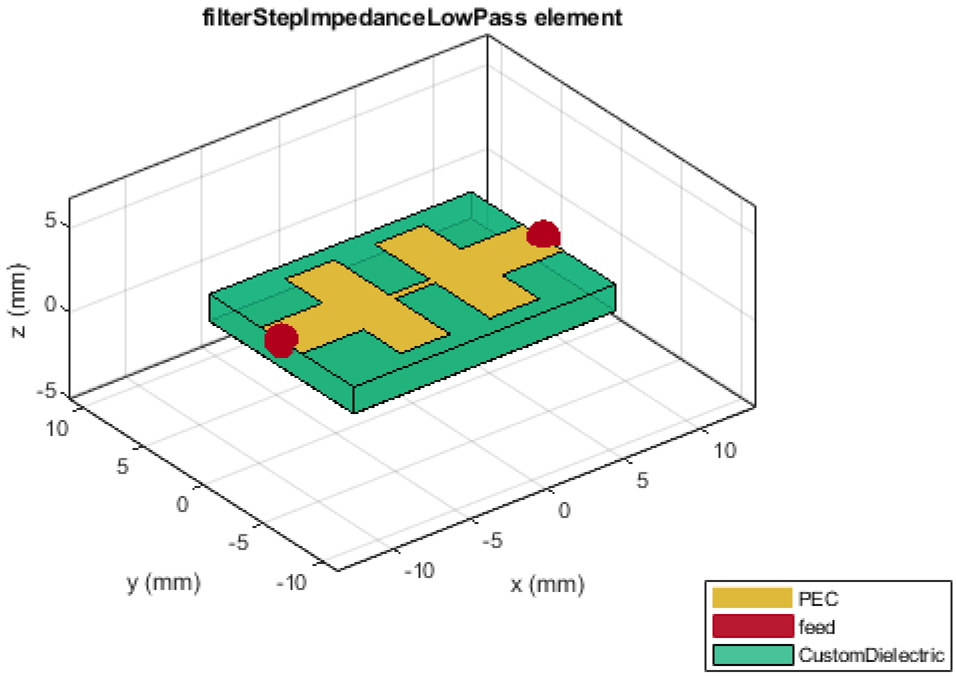
Default Stepped Impedance Lowpass Filter

Create and view a default stepped impedance lowpass filter.

```
steppedfilter = filterStepImpedanceLowPass
```

```
steppedfilter = 
  filterStepImpedanceLowPass with properties:

         FilterOrder: 3
       PortLineWidth: 0.0034
      PortLineLength: 0.0040
      HighZLineWidth: 5.0000e-04
       LowZLineWidth: 0.0096
     HighZLineLength: 0.0026
      LowZLineLength: 0.0032
              Height: 0.0016
    GroundPlaneWidth: 0.0120
           Substrate: [1x1 dielectric]
           Conductor: [1x1 metal]
```

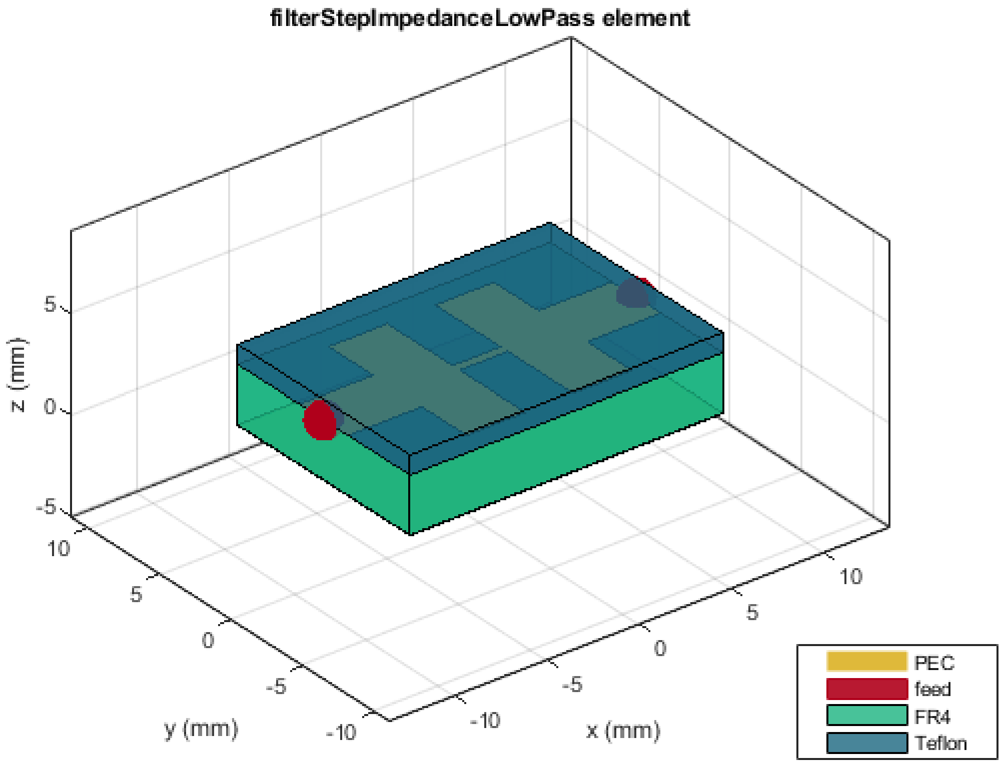
```
show(steppedfilter)
```



Create Stepped Impedance Lowpass Filter with Multilayer Dielectric Substrate

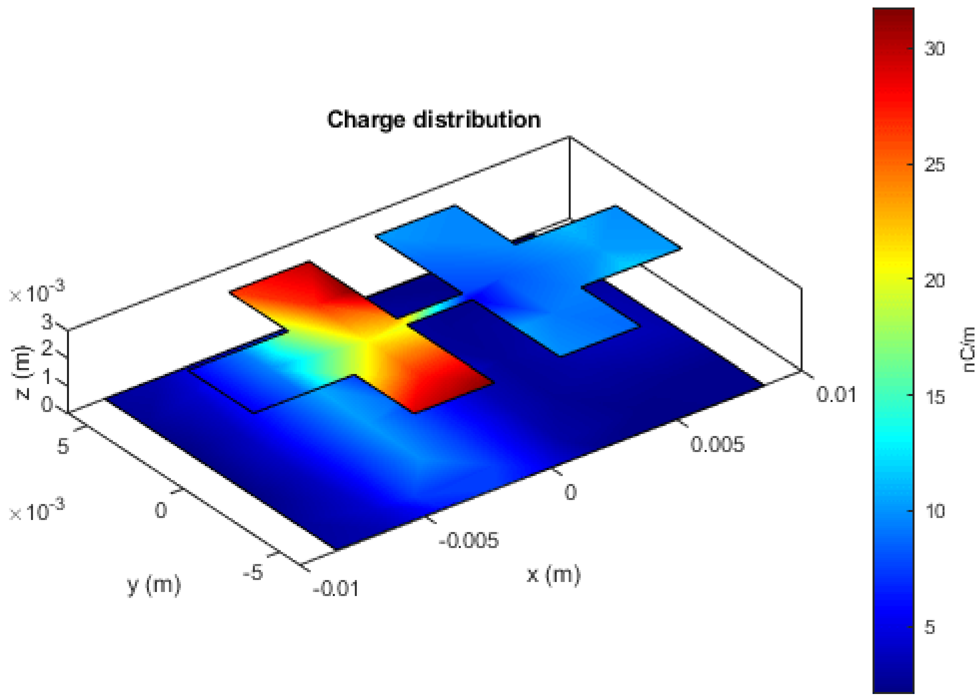
Create and view a stepped impedance lowpass filter with a multilayer dielectric substrate.

```
sub = dielectric("FR4","Teflon");
sub.Thickness =[0.003 0.001];
steppedfilter = filterStepImpedanceLowPass;
steppedfilter.Height = 0.003;
steppedfilter.Substrate = sub;
figure
show(steppedfilter)
```

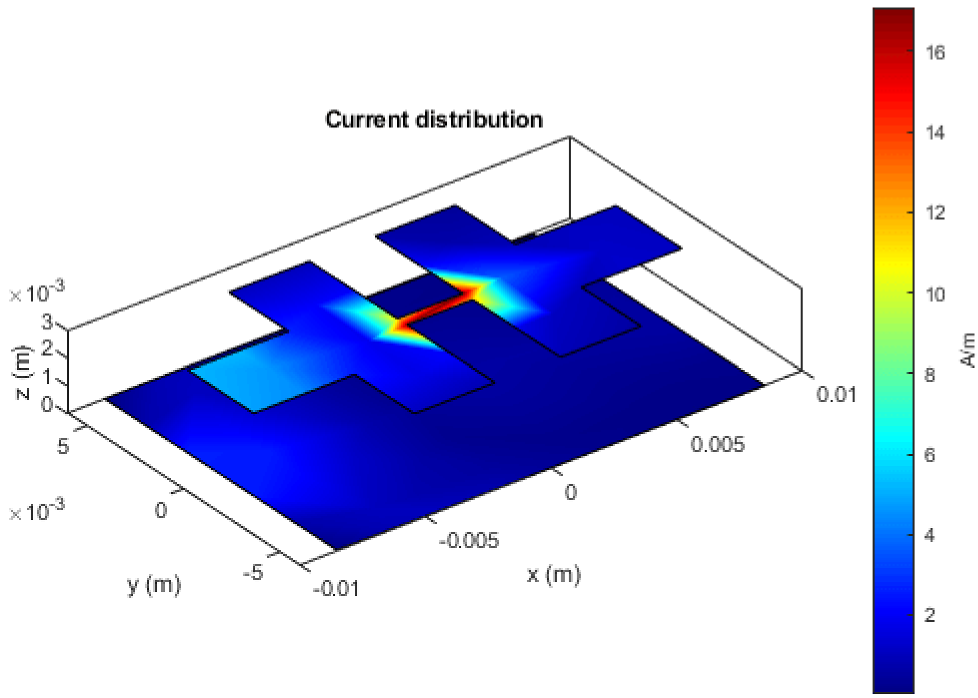


Plot the charge and current on the filter at 5 GHz.

```
figure
charge(steppedfilter,5e9)
```




```
figure
current(steppedfilter,5e9)
```




```
info(steppedfilter)
```

```
ans = struct with fields:
           IsSolved: "true"
          IsMeshed: "true"
        MeshingMode: "auto"
       HasSubstrate: "true"
            HasLoad: "false"
      PortFrequency: []
     MemoryEstimate: "790 MB"
```

References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

[2] Garvansh, Abhay, Singh Kushwaha, Navita Singh, and Arun Kumar. “Implementation of Stepped Impedance Low Pass Microstrip Line Filter for Wireless Communication.” *International Journal of Advanced Research in Computer and Communication Engineering* 3, no. 7 (July 2014): 7608–10.

[3] Maity, Budhadeb. “Stepped Impedance Low Pass Filter Using Microstrip Line for C-Band Wireless Communication.” In *2016 International Conference on Computer Communication and Informatics (ICCCI)*, 1–4. Coimbatore, India: IEEE, 2016. https://doi.org/10.1109/ICCCI.2016.7480008.

See Also
filterCoupledLine | filterHairpin

Introduced in R2021b

couplerLange

Create Lange coupler

Description

Use the `couplerLange` object to create a Lange coupler. Lange couplers are passive devices used as power combiners and RF or microwave splitters for a large range of frequencies. This PCB component has four ports. By default, port 1 is the input port, port 4 is the through port, port 3 is the coupled port, and port 2 is the isolated port.

Creation

Syntax

```
coupler = couplerLange
coupler = couplerLange(Name=Value)
```

Description

`coupler = couplerLange` creates a Lange coupler. The default property values are for a resonant frequency of 10 GHz.

`coupler = couplerLange(Name=Value)` sets “Properties” on page 1-161 using one or more name-value arguments. For example, `couplerLange(FingerLength=0.0028)` creates a Lange coupler with the finger length of 0.0028 meters. Properties not specified retain their default values.

Properties

`FingerLength` — Length of center finger
`0.00356` (default) | positive scalar

Length of the center finger in meters, specified as a positive scalar.

Example: `coupler = couplerLange(FingerLength=0.0286)`

Data Types: `double`

`FingerWidth` — Width of each finger
`2.6600e-04` (default) | positive scalar

Width of each finger in meters, specified as a positive scalar.

Example: `coupler = couplerLange(FingerWidth=0.0286)`

Data Types: `double`

`FingerSpacing` — Distance between fingers
`6.0000e-05` (default) | positive scalar

Distance between the fingers in meters, specified as a positive scalar.

Example: `coupler = couplerLange(FingerSpacing=0.0286)`

Data Types: `double`

`NumWireBond` — Number of bonds used to connect interdigitated strips
`2` (default) | `1`

Number of bonds used to connect the interdigitated strips, specified as a `1` or `2`.

Example: `coupler = couplerLange(NumWireBond=1)`

Data Types: `double`

`WireBondDiameter` — Width of each wire bond strip
`1.0000e-04` (default) | positive scalar

Width of each wire bond strip in meters, specified as a positive scalar. The wire bonding strips are used to connect alternate fingers using vias.

Example: `coupler = couplerLange(WireBondDiameter=0.0286)`

Data Types: `double`

`WireBondSpacing` — Distance between wire bonding strips
`1.40000e-04` (default) | positive scalar

Distance between the wire bonding strips in meters, specified as a positive scalar. The wire bonding strips are used to connect alternate fingers using vias.

Example: `coupler = couplerLange(WireBondSpacing=0.0386)`

Data Types: `double`

`PortLineWidth` — Width of input and output line
`0.0023` (default) | positive scalar

Width of the input and output line in meters, specified as a positive scalar.

Example: `coupler = couplerLange(PortLineWidth=0.0070)`

Data Types: `double`

`PortLineLength` — Length of input and output port
`0.0028` (default) | positive scalar

Length of the input and output line in meters, specified as a positive scalar.

Example: `coupler = couplerLange(PortLineLength=0.0070)`

Data Types: `double`

`Height` — Height from ground plane to coupler
`0.0015` (default) | positive scalar

Height from the ground plane to the coupler in meters, specified as a positive scalar.

Example: `coupler = couplerLange(Height=0.0025)`

Data Types: `double`

`GroundPlaneWidth` — Width of ground plane in meters
`0.0115` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `coupler = couplerLange(GroundPlaneWidth=0.046)`

Example: double

`Substrate` — Type of dielectric material
`dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The default value is a `dielectric` object with these properties:

- `Name—{'Alumina','Air'}`
- `EpsilonR—[6,1]`
- `LossTangent—[0.00023,0]`
- `Thickness—[1.524e-3,1.3050e-4]`

Example: `d = dielectric("FR4"); coupler = couplerLange(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`PEC` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("PEC"); coupler = couplerLange(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
mesh	Change and view mesh properties of metal or dielectric in PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

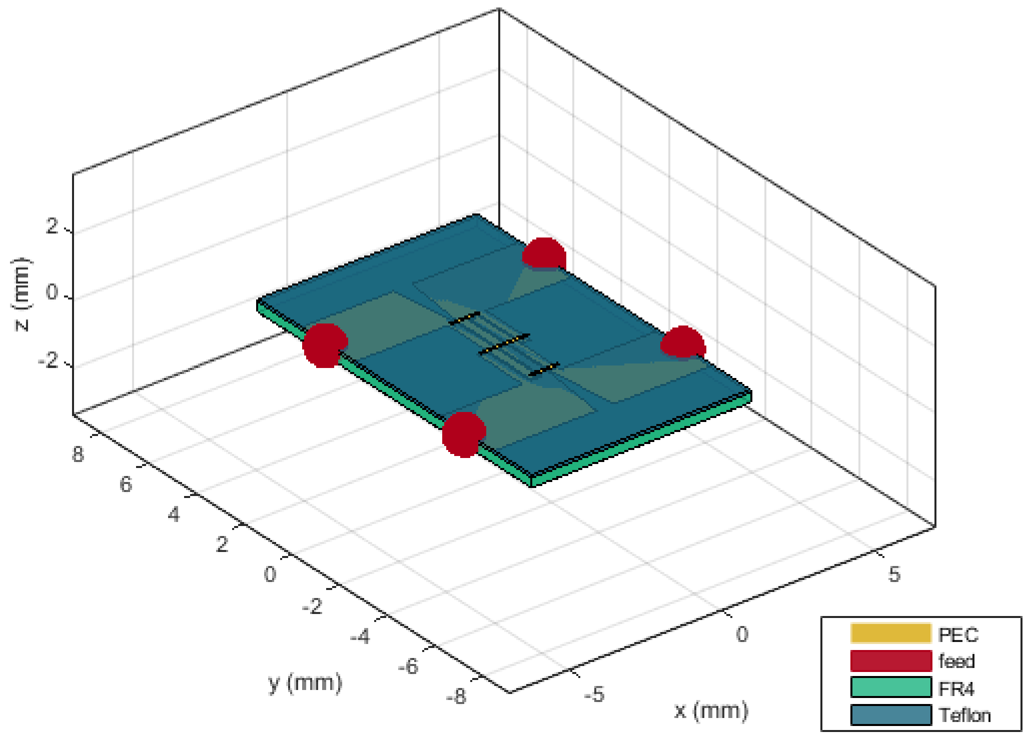
Examples

???

Top-Layer Dielectric Lange Coupler

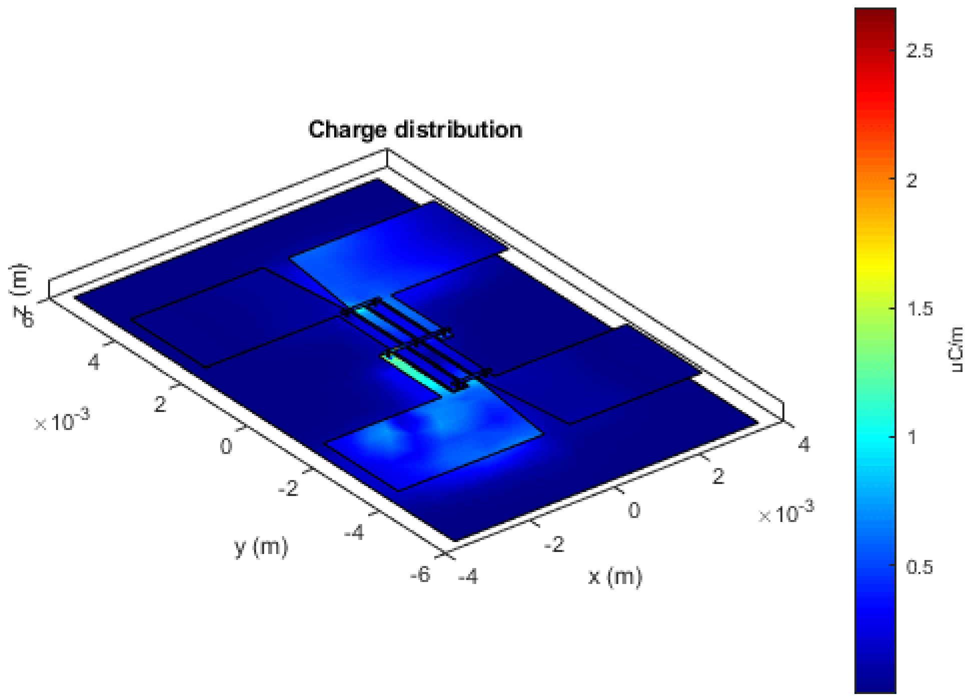
Create and view a Lange coupler with a dielectric as the top layer and the number of wire bonds set to 1.

```
sub = dielectric("FR4","Teflon");
sub.Thickness =[0.0003 0.0001];
coupler = couplerLange(Height=0.0003,Substrate=sub,NumWireBond=1);
show(coupler);
```



Plot the charge on this coupler at 10 GHz.

```
charge(coupler,10e9)
```



More About

Parametric Analysis Guidelines

- The length of each finger is equal to a quarter wavelength at the design frequency.
- Increase the FingerLength to decrease the frequency.
- Decrease the Height (thickness of the substrate above the Lange coupler layer) to increase the value of S31 and decrease the value of S21.
- Adjust the Height, WireBondDiameter, and WireBondSpacing to improve the coupling between S31 and S21.

References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

[2] Caloz, Christophe, and Tatsuo Itoh. *Electromagnetic Metamaterials: Transmission Line Theory and Microwave Applications: The Engineering Approach*. Hoboken, NJ, USA: John Wiley & Sons, Inc., 2005. https://doi.org/10.1002/0471754323.

See Also

`couplerBranchline` | `couplerRatrace`

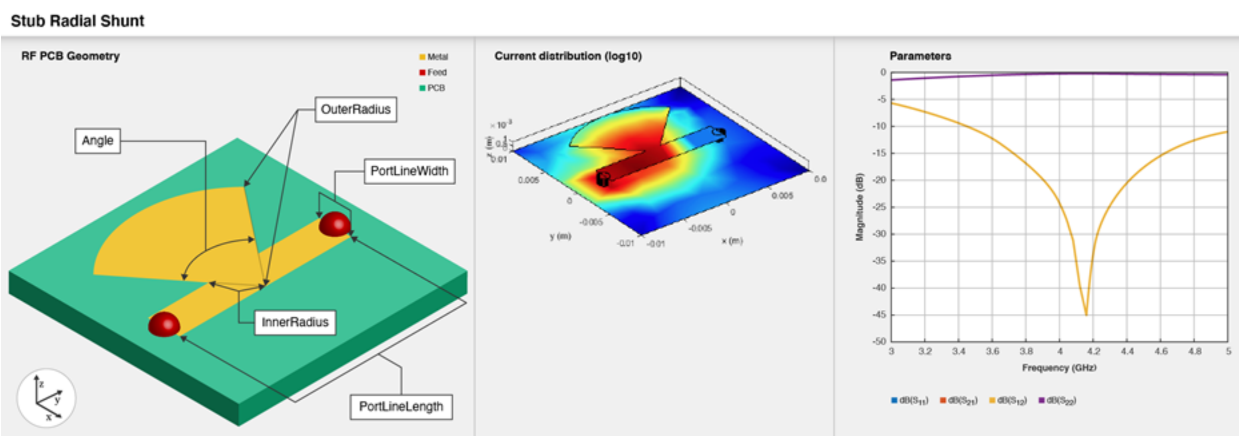
Introduced in R2021b

stubRadialShunt

Create single- and double-radial stub shunt on X-Y plane

Description

Use the `stubRadialShunt` object to create a single- or double-radial stub shunt on the X-Y plane.



Radial stubs provide broadly resonant RF short circuits by introducing the short at a concentrated point. When cascaded with high-impedance quarter-wavelength transmission lines, radial stubs provide an effectively decoupled network for microwave amplifiers and other active components.

Creation

Syntax

```
stub = stubRadialShunt
stub = stubRadialShunt(Name=Value)
```

Description

`stub = stubRadialShunt` creates a single-radial stub shunt in the X-Y plane. The stub dimensions are for the frequency range of 3-5 GHz with a resonant frequency of 4.2 GHz on the X-Y plane.

`stub = stubRadialShunt(Name=Value)` sets “Properties” on page 1-167 using one or more name-value arguments. For example, `stubRadialStub(OuterRadius=0.0070)` creates a radial stub shunt with an outer radius of `0.0070` meters. Properties not specified retain their default values.

Properties

StubType — Type of radial stub
`"Single"` (default) | `"Double"`

Type of radial stub, specified as "Single" or "Double".

Example: `stub = stubRadialShunt(StubType="Double")`

Data Types: `string` | `char`

`OuterRadius` — Outer radius of radial stub
`0.0085` (default) | positive scalar | two-element vector

Outer radius of the radial stub in meters, specified as a positive scalar or a two-element vector of positive elements. Specify a two-element vector for a double-radial stub.

Example: `stub = stubRadialShunt(OuterRadius=0.0070)`

Data Types: `double`

`InnerRadius` — Inner radius of radial stub
`0.0012` (default) | positive scalar | two-element vector

Inner radius of the radial stub in meters, specified as a positive scalar or a two-element vector of positive elements. Specify a two-element vector for a double-radial stub.

Example: `stub = stubRadialShunt(InnerRadius=0.0023)`

Data Types: `double`

`Angle` — Angle of stub
`90` (default) | positive scalar in the range [5, 175] | two-element vector in the range [5, 175]

Angle of the stub in degrees, specified as a positive scalar or a two-element vector of positive elements. Specify a two-element vector for a double-radial stub. The stub angles must be greater than or equal to 5 degrees and less than or equal to 175 degrees.

Example: `stub = stubRadialShunt(Angle=60)`

Data Types: `double`

`PortLineWidth` — Width of microstrip line
`0.0025` (default) | positive scalar

Width of the microstrip line in meters, specified as a positive scalar.

Example: `stub = stubRadialShunt(PortLineWidth=0.0035)`

Data Types: `double`

`PortLineLength` — Length of microstrip line
`0.0137` (default) | positive scalar

Length of the microstrip line in meters, specified as a positive scalar.

Example: `stub = stubRadialShunt(PortLineLength=0.0237)`

Data Types: `double`

`Height` — Height of radial stub from ground plane
`0.0016` (default) | positive scalar

Height of the radial stub from the ground plane, specified as a positive scalar.

Example: `stub = stubRadialShunt(Height=0.0015)`

Data Types: `double`

`GroundPlaneLength` — Length of ground plane
`0.0200` (default) | positive scalar

Length of the ground plane in meters, specified as a positive scalar.

Example: `stub = stubRadialShunt(GroundPlaneLength=0.046)`

Example: double

`GroundPlaneWidth` — Width of ground plane
`0.0200` (default) | positive scalar

Width of the ground plane in meters, specified as a positive scalar.

Example: `stub = stubRadialShunt(GroundPlaneWidth=0.046)`

Example: double

`Substrate` — Type of dielectric material
`'Teflon'` (default) | `dielectric` object

Type of dielectric material used as a substrate, specified as a `dielectric` object. The thickness of the default dielectric material Teflon is `0.8` mm or the same as the `Height` property.

Example: `d = dielectric("FR4"); stub = stubRadialShunt(Substrate=d)`

Data Types: `string` | `char`

`Conductor` — Type of metal used in conducting layers
`'PEC'` (default) | `metal` object

Type of metal used in the conducting layers, specified as a `metal` object.

Example: `m = metal("PEC"); stub = stubRadialShunt(Conductor=m)`

Data Types: `string` | `char`

Object Functions

charge	Calculate and plot charge distribution
current	Calculate and plot current distribution
feedCurrent	Calculate current at feed port
getZ0	Calculate characteristic impedance of transmission line
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
shapes	Extract all metal layer shapes of PCB component
show	Display PCB component structure or PCB shape
sparameters	Calculate S-parameters for RF PCB objects

Examples

Create Default Radial Stub Shunt

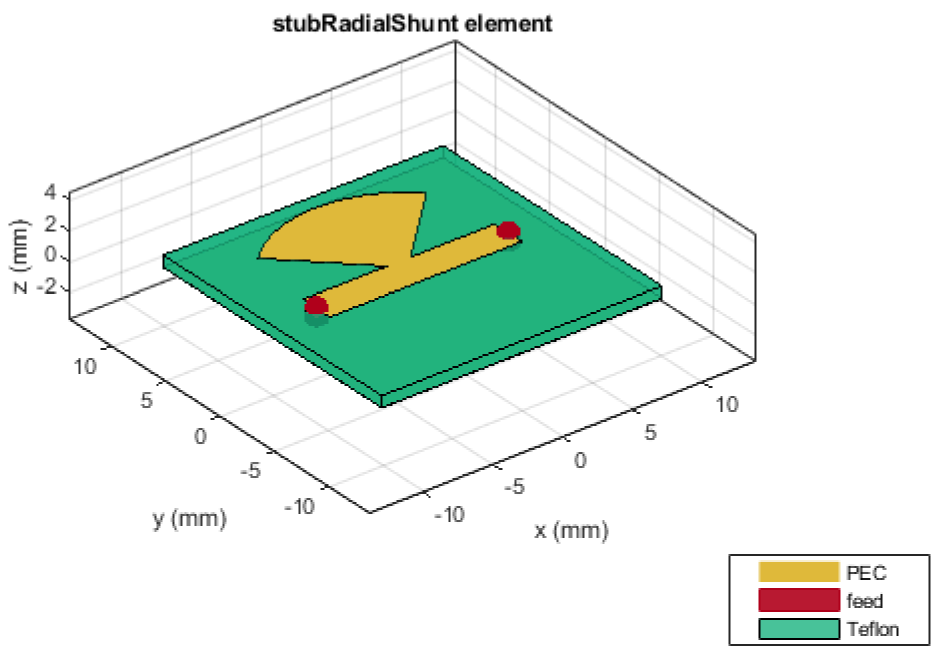
Create a default radial stub shunt.

```
stub = stubRadialShunt
```

```
stub = 
  stubRadialShunt with properties:

              StubType: 'Single'
           OuterRadius: 0.0085
           InnerRadius: 0.0012
                 Angle: 90
        PortLineLength: 0.0137
         PortLineWidth: 0.0025
                Height: 8.0000e-04
     GroundPlaneLength: 0.0200
      GroundPlaneWidth: 0.0200
             Substrate: [1x1 dielectric]
             Conductor: [1x1 metal]
```

View the radial stub shunt.

```
show(stub)
```



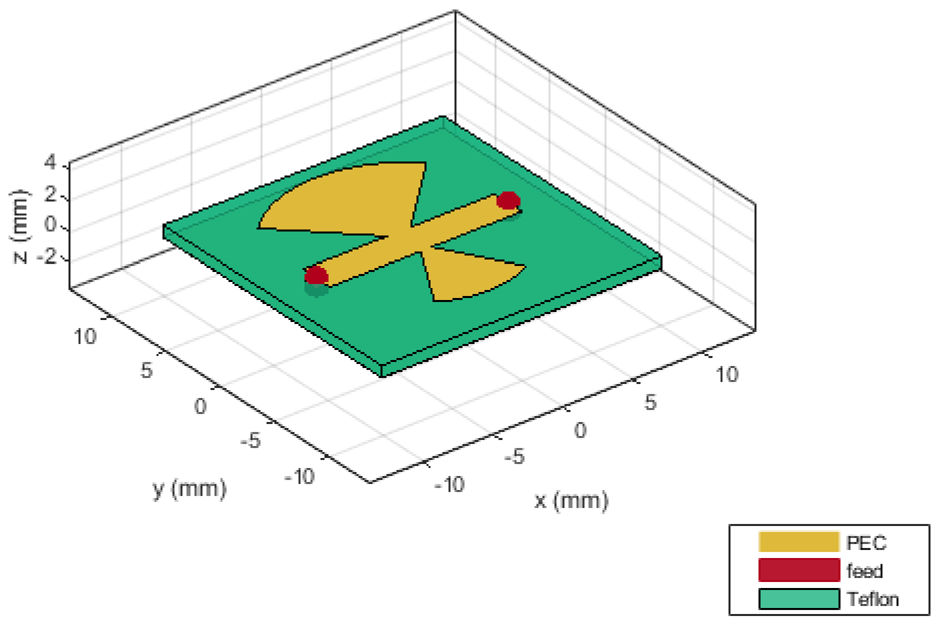
Create Double Shunt Radial Stub

Create shunt radial stub of type double.

```
stub = stubRadialShunt(StubType='double');
stub.OuterRadius = [0.0085 0.0065];
stub.InnerRadius = [0.0012 0.0008];
stub.Angle       = [90 60];
```

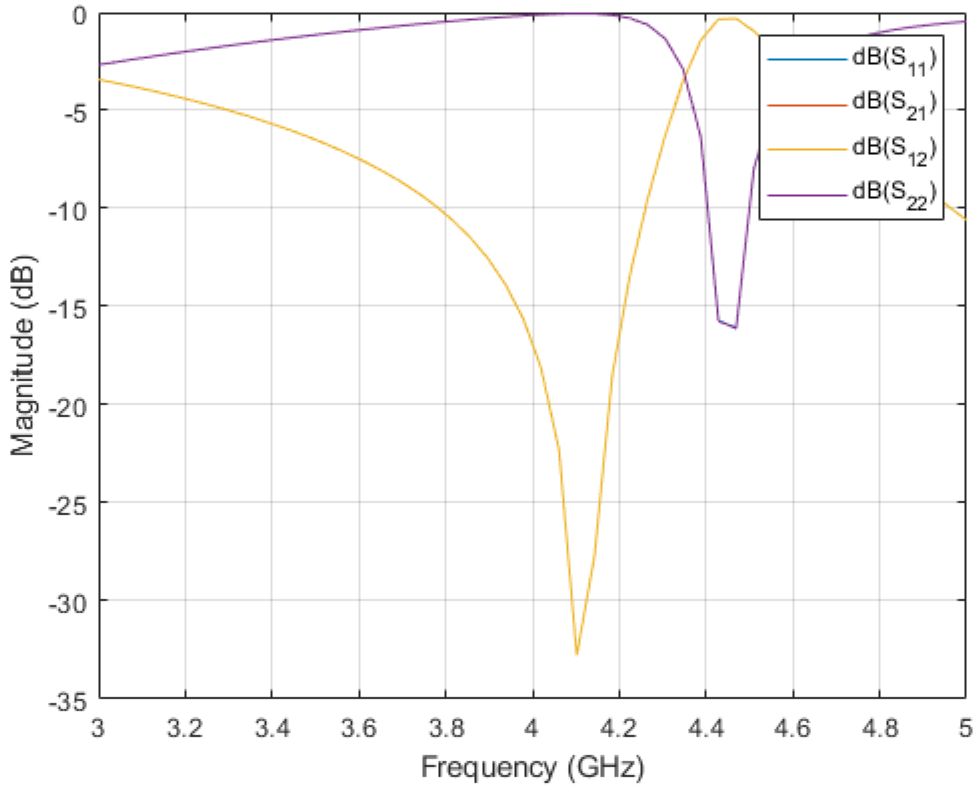
View shunt radial stub.

```
show(stub)
```



Plot s-parameters.

```
spar = sparameters(stub,linspace(3e9,5e9,50));
rfplot(spar)
```



More About

Parametric Analysis Guidelines

- Increase the Angle to shift the resonance of the stub to a lower frequency.
- Increase the OuterRadius to shift the resonance of the stub to a lower frequency.
- Adjust the OuterRadius and increase the Angle to design a radial stub at a desired frequency, line length, low insertion loss, and wide bandwidth.

References

[1] Wang , Zhebin, and Chan-Wang Park. “Novel Wideband GaN HEMT Power Amplifier Using Microstrip Radial Stub to Suppress Harmonics.” In *2012 IEEE/MTT-S International Microwave Symposium Digest*, 1–3. Montreal, QC, Canada: IEEE, 2012. https://doi.org/10.1109/MWSYM.2012.6259464.

[2] Singh, Prashant, and Tiwary Anjini. “Novel Compact Dual Bandstop Filter Using Radial Stub.” *Microwave Review* 21 (September 1, 2015): 17–22.

See Also

`microstripLine`

Introduced in R2021b

traceSpiral

Create even-sided polygon trace in spiral form

Description

Use the `traceSpiral` object to create an even-sided polygon trace such as a square, hexagon, octagon, decagon, or a circle in a spiral form.

Creation

Syntax

```
trace = traceSpiral
trace = traceSpiral(Name=Value)
```

Description

`trace = traceSpiral` creates a square spiral trace. The spiral trace is centered at the origin on the X-Y plane.

`trace = traceSpiral(Name=Value)` sets "Properties" on page 1-174 using one or more name-value arguments. For example, `traceSpiral(ReferencePoint=[1 1])` creates a spiral trace with the reference point `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of spiral trace
`'myspiral'` (default) | character vector | string scalar

Name of the spiral trace, specified as a character vector or a string scalar.

Example: `trace = traceSpiral(Name="spiraltrace1")`

Data Types: `char` | `string`

`ReferencePoint` — Point of reference of spiral trace
`[0 0]` (default) | two-element vector

Point of reference of the spiral trace in Cartesian coordinates, specified as a two-element vector. Use the reference point to modify the shape from its initial position.

Example: `trace = traceSpiral(ReferencePoint=[1 1])`

Data Types: `double`

`InnerDiameter` — Inner diameter of spiral trace
`0.0040` (default) | positive scalar

Inner diameter of the spiral trace in meters, specified as a positive scalar. If the polygon is a square, the inner diameter is the distance between the innermost vertex and the midpoint of the opposite side of the inner square. For all other shapes, the value is the distance between the innermost vertex and the opposite vertex of the inner turn.

Example: `trace = traceSpiral(InnerDiameter=0.0015)`

Data Types: `double`

`TraceWidth` — Width of spiral trace

`0.0020` (default) | positive scalar

Width of the spiral trace in meters, specified as a positive scalar.

Example: `trace = traceSpiral(TraceWidth=0.0050)`

Data Types: `double`

`Spacing` — Distance between traces of spiral

`5.0000e-04` (default) | positive scalar

Distance between the traces of the spiral in meters, specified as a positive scalar. For a square spiral trace, the spacing is the gap between the flat edges of adjacent turns. For all other shapes, the spacing is the gap between vertices of adjacent turns.

Example: `trace = traceSpiral(Spacing=6.0000e-04)`

Data Types: `double`

`NumTurns` — Number of turns in spiral

`4` (default) | positive scalar

Number of turns in the spiral, specified as a positive scalar.

Example: `trace = traceSpiral(NumTurns=6)`

Data Types: `double`

`NumSides` — Number of sides in each turn

`4` (default) | positive, even scalar

Number of sides in each turn of the spiral based on the polygon, specified as a positive, even scalar. The minimum number of sides is `4` and the maximum number is `10`. If the number exceeds `10`, then the shape is a circle.

Example: `trace = traceSpiral(NumSides=6)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis

rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Spiral Trace

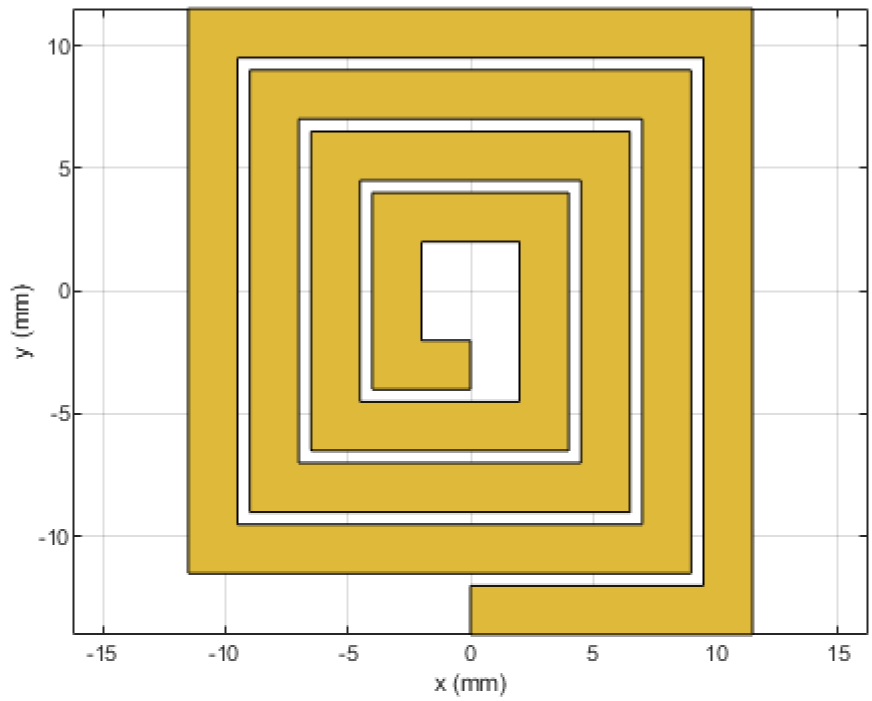
Create a spiral trace with default properties.

```
trace = traceSpiral
```

```
trace = 
  traceSpiral with properties:

             Name: 'myspiral'
    ReferencePoint: [0 0]
     InnerDiameter: 0.0040
        TraceWidth: 0.0020
           Spacing: 5.0000e-04
          NumTurns: 4
          NumSides: 4
```

View the trace.

```
show(trace)
```



See Also

traceLine | traceCross | traceTee | tracePoint | traceRectangular

Introduced in R2021b

traceRectangular

Create rectangular trace

Description

Use the `traceRectangular` object to create a rectangular trace centered at the origin on the X-Y plane.

Creation

Syntax

```
trace = traceRectangular
trace = traceRectangular(Name=Value)
```

Description

`trace = traceRectangular` creates a rectangular trace centered at the origin and on the X-Y plane.

`trace = traceRectangular(Name=Value)` sets “Properties” on page 1-178 using one or more name-value arguments. For example, `traceRectangular(Center=[1 1])` creates a rectangular trace centered at `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of rectangular trace
`'mytraceRectangular'` (default) | character vector | string scalar

Name of the rectangular trace, specified as a character vector or a string scalar.

Example: `trace = traceRectangular(Name="rectangletrace1")`

Data Types: `char` | `string`

`Center` — Center of rectangular trace
`[0 0]` (default) | two-element vector

Center of the rectangular trace in Cartesian coordinates, specified as a two-element vector.

Example: `trace = traceRectangular(Center=[1 1])`

Data Types: `double`

`Length` — Length of rectangle
`0.0200` (default) | positive scalar

Length of the rectangle in meters, specified as a positive scalar.

Example: `trace = traceRectangular(Length=0.0500)`

Data Types: `double`

`Width` — Width of rectangle
`0.0050` (default) | positive scalar

Width of the rectangle in meters, specified as a positive scalar.

Example: `trace = traceRectangular(Width=0.015)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Rectangular Trace

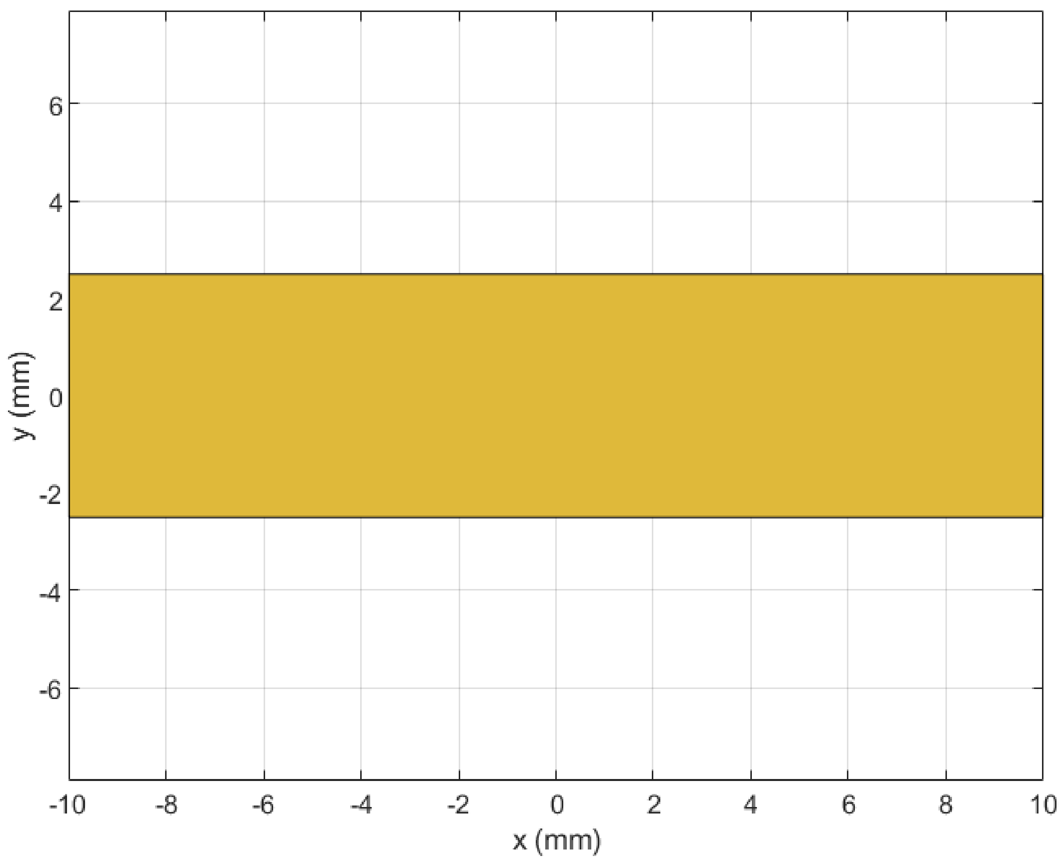
Create a rectangular trace with default properties.

```
trace = traceRectangular
```

```
trace = 
  traceRectangular with properties:

      Name: 'mytraceRectangular'
    Center: [0 0]
    Length: 0.0200
     Width: 0.0050
```

View the trace.

```
show(trace)
```



Mesh Rectangular Trace

Create a 2 cm-by-2 cm rectangular trace.

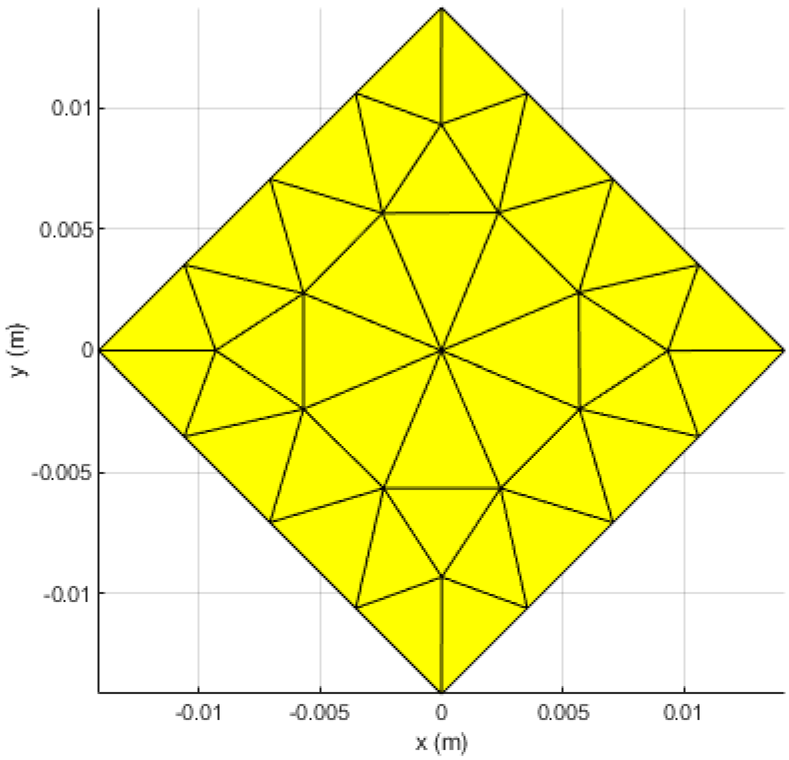
```
trace = traceRectangular(Length=0.02,Width=0.02);
```

Rotate the trace by 45 degrees about the z-axis.

```
trace = rotateZ(trace,45);
```

Mesh the trace at a maximum edge length of 5 mm.

```
mesh(trace,MaxEdgeLength=5e-3)
```



See Also

`traceLine` | `traceCross` | `traceTee` | `tracePoint` | `traceSpiral`

Introduced in R2021b

traceTee

Create tee trace

Description

Use the `traceTee` object to create a tee trace on the X-Y plane.

Creation

Syntax

```
trace = traceTee
trace = traceTee(Name=Value)
```

Description

`trace = traceTee` creates a tee trace with default properties on the X-Y plane.

`trace = traceTee(Name=Value)` sets "Properties" on page 1-182 using one or more name-value arguments. For example, `traceTee(ReferencePoint=[1 1])` creates a tee trace with the reference point `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of tee trace
`'mytraceTeeShape'` (default) | character vector | string scalar

Name of the tee trace, specified as a character vector or a string scalar.

Example: `trace = traceTee(Name="traceTeeShape")`

Data Types: `char` | `string`

`ReferencePoint` — Reference point of tee trace
`[0 0]` (default) | two-element vector

Reference point of the tee trace in meters, specified as a two-element vector of nonnegative elements.

Example: `trace = traceTee(ReferencePoint=[1 1])`

Data Types: `double`

`Length` — Length of horizontal and vertical lines
`[0.0200 0.0100]` (default) | two-element vector

Length of the horizontal and vertical lines in meters, specified as a two-element vector of positive elements.

Example: `trace = traceTee(Length=[0.0300 0.0200])`

Data Types: `double`

`Width` — Width of horizontal and vertical lines
`[0.0050 0.0050]` (default) | two-element vector

Width of the horizontal and vertical lines in meters, specified as a two-element vector of positive elements.

Example: `trace = traceTee(Width=[0.0060 0.0060])`

Data Types: `double`

`Offset` — Offset along X-axis
`0` (default) | nonnegative scalar

Offset along the X-axis in meters, specified as a nonnegative scalar.

Example: `trace = traceTee(Offset=0.0005)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
subtract	Boolean subtraction operation on two RF PCB shapes
intersect	Boolean intersection operation on two RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
minus	Shape1 - Shape2 for RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
translate	Move RF PCB shape to new location
scale	Change size of RF PCB shape by fixed amount

Examples

Create Default Tee Trace

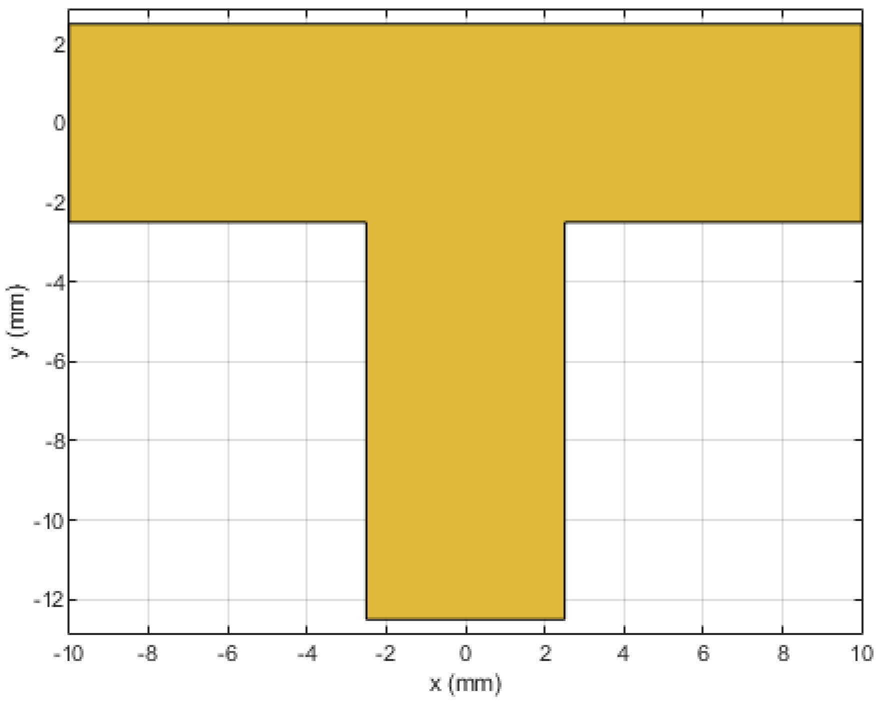
Create a tee trace with default properties.

```
trace = traceTee
```

```
trace = 
  traceTee with properties:

              Name: 'mytraceTeeShape'
    ReferencePoint: [0 0]
            Length: [0.0200 0.0100]
             Width: [0.0050 0.0050]
            Offset: 0
```

View the trace.

```
show(trace)
```



Use Behavioral Model to Calculate S-Parameters of Microstrip T-Junction

Design a microstrip transmission line at 3 GHz for FR4 substrate.

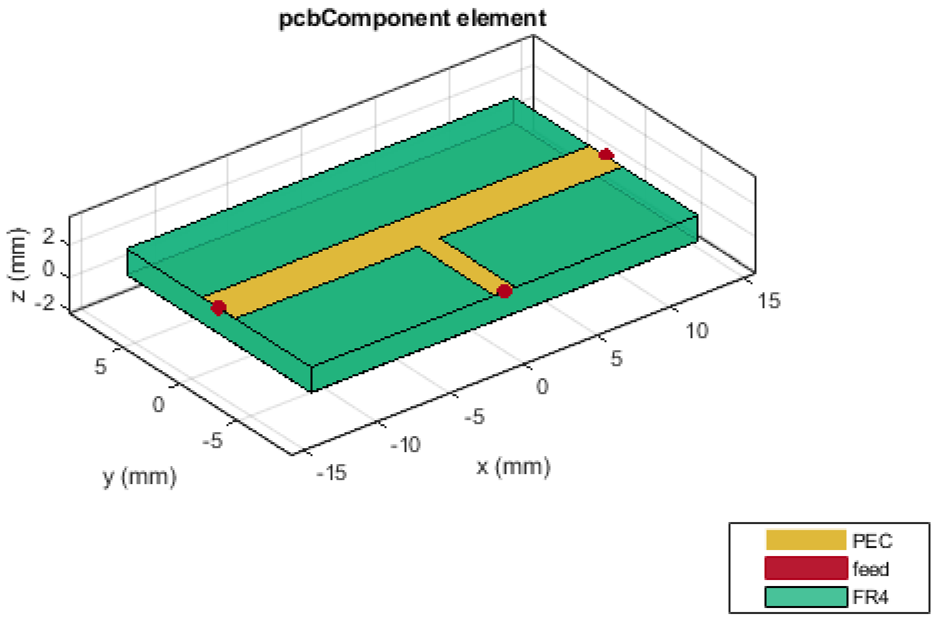
```
m = design(microstripLine('Substrate',dielectric('FR4')),3e9);
```

Create a microstrip T-junction.

```
layer2d = traceTee('Length',[m.Length m.Length/4],...
"Width",[m.Width m.Width/2]);
```

Convert the T-junction trace to a 3-D component.

```
robj = pcbComponent(layer2d);
robj.BoardThickness = m.Substrate.Thickness;
robj.Layers{2} = m.Substrate;
show(robj)
```



Define frequency points to calculate the s-parameters.

```
freq = (1:40)*100e6;
```

Calculate the s-parameters of the T-junction trace using the behavioral model.

```
Sckt = sparameters(robj,freq,75,'Behavioral',true);
```

```
Warning: Behavioral model is valid only when Z0 of main line is 50 ohms and for EpsilonR of 9.9.
```

Calculate the s-parameters of the T-junction trace using the electromagnetic solver.

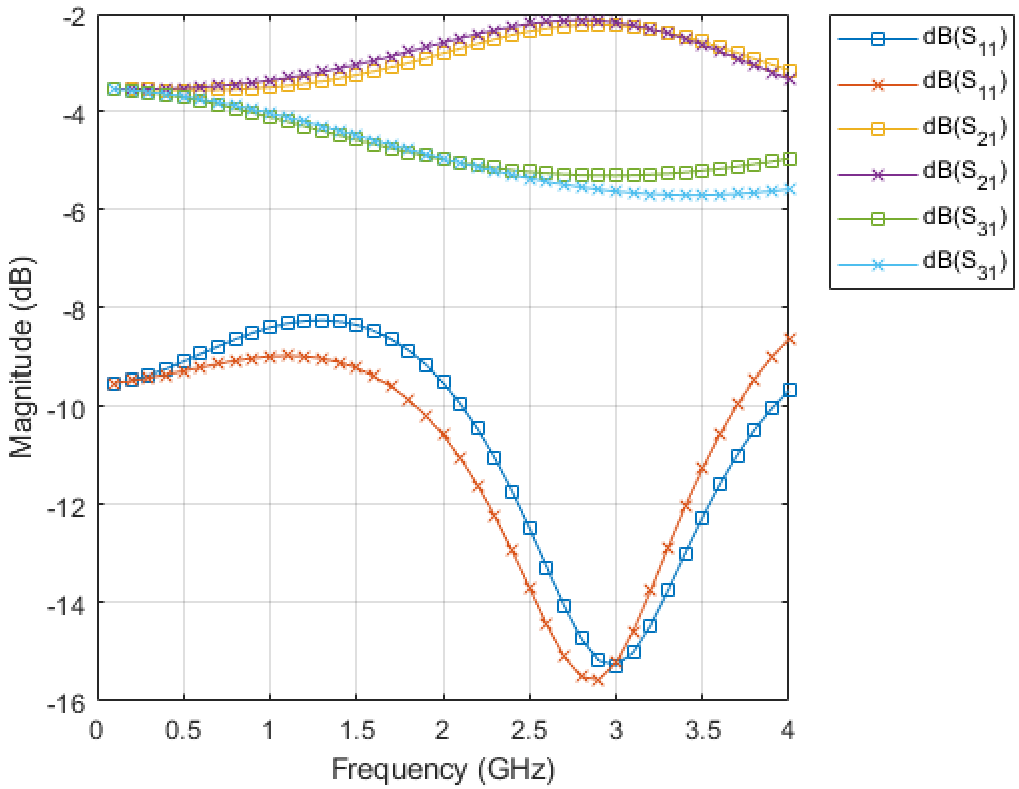
```
Sem = sparameters(robj,freq,75)
```

```
Sem = 
  sparameters: S-parameters object

       NumPorts: 3
    Frequencies: [40x1 double]
     Parameters: [3x3x40 double]
      Impedance: 75

  rfparam(obj,i,j) returns S-parameter Sij
```

Plot the s-parameter data using the `rfplot` function.

```
rfplot(Sckt,1,1,'db','-s')
hold on
```

```
rfplot(Sem,1,1,'db','-x')
rfplot(Sckt,2,1,'db','-s')
rfplot(Sem,2,1,'db','-x')
rfplot(Sckt,3,1,'db','-s')
rfplot(Sem,3,1,'db','-x')
```



References:

1 Ramesh Garg & I. J. Bahl (1978) Microstrip discontinuities, International Journal of Electronics, 45:1, 81-87, DOI: 10.1080/00207217808900883

2 Wadell, Brian C. *Transmission Line Design Handbook*. The Artech House Microwave Library. Boston: Artech House, 1991.

See Also

`traceLine` | `traceCross` | `traceRectangular` | `tracePoint` | `traceSpiral`

Introduced in R2021b

traceCross

Create cross-shaped trace

Description

Use the `traceCross` object to create a cross-shaped trace on the X-Y plane.

Creation

Syntax

```
trace = traceCross
trace = traceCross(Name=Value)
```

Description

`trace = traceCross` creates a cross-shaped trace with default properties on the X-Y plane.

`trace = traceCross(Name=Value)` sets “Properties” on page 1-187 using one or more name-value arguments. For example, `traceCross(ReferencePoint=[1 1])` creates a cross-shaped trace at the reference point `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of cross-shaped trace
`'mytraceCross'` (default) | character vector | string scalar

Name of the cross-shaped trace, specified as a character vector or a string scalar.

Example: `trace = traceCross(Name="traceCrossShape")`

Data Types: `char` | `string`

`ReferencePoint` — Reference point for cross-shaped trace
`[0 0]` (default) | two-element vector

Reference point for the cross-shaped trace in Cartesian coordinates, specified as a two-element vector.

Example: `trace = traceCross(ReferencePoint=[1 1])`

Data Types: `double`

`Length` — Length of cross-shaped trace
`[0.0100 0.0100]` (default) | two-element vector

Length of the cross-shaped trace in meters, specified as a two-element vector of positive elements.

Example: `trace = traceCross(Length=[0.0800 0.0400])`

Data Types: `double`

`Width` — Width of cross-shaped trace
`[0.0020 0.0020]` (default) | two-element vector

Width of the cross-shaped trace in meters, specified as a two-element vector of positive elements.

Example: `trace = traceCross(Width=[0.005 0.005])`

Data Types: `double`

`Offset` — Offset along X and Y direction
`[0 0]` (default) | two-element vector

Offset along the X and Y direction in meters, specified as a two-element vector.

Example: `trace = traceCross(Offset=[1 1])`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Cross Trace

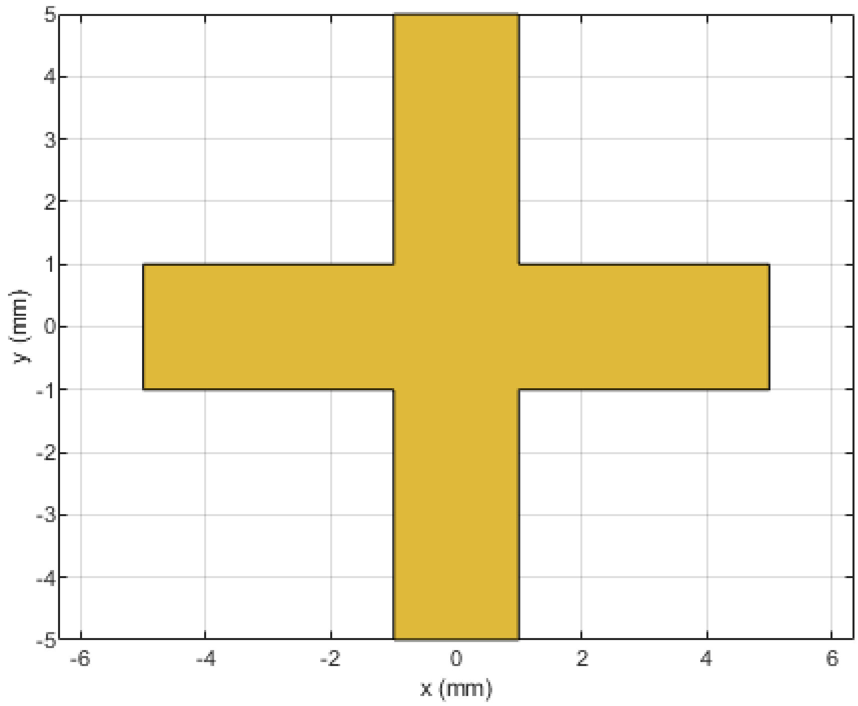
Create a cross-shaped trace with default properties.

```
trace = traceCross
```

```
trace = 
  traceCross with properties:

              Name: 'mytraceCross'
    ReferencePoint: [0 0]
            Length: [0.0100 0.0100]
             Width: [0.0020 0.0020]
            Offset: [0 0]
```

View the trace.

```
show(trace)
```



Use Behavioral Model to Calculate S-Parameters of Microstrip Cross

Design a microstrip transmission line at 3 GHz for FR4 substrate.

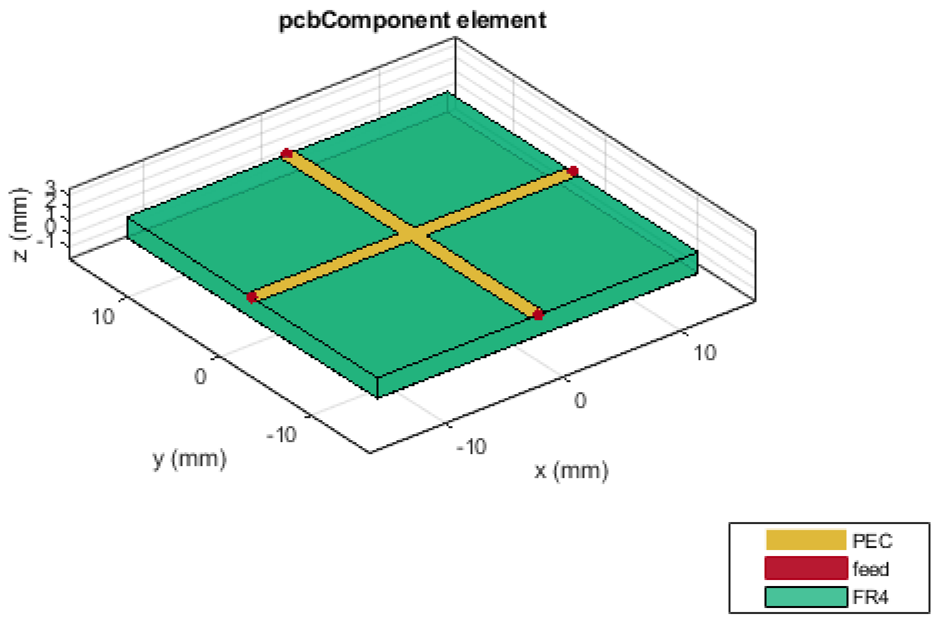
```
d = dielectric('FR4');
d.LossTangent = 0;
m = design(microstripLine('Substrate',d),3e9,'Z0',75,...
    'LineLength',0.5);
```

Create a microstrip cross.

```
layer2d = traceCross('Length',[m.Length m.Length], ...
    'Width',[m.Width m.Width]);
```

Convert the cross trace to a PCB component.

```
robj = pcbComponent(layer2d);
robj.BoardThickness = m.Substrate.Thickness;
robj.Layers{2} = m.Substrate;
show(robj)
```



Define frequency points to calculate the s-parameters.

```
freq = (1:3:40)*100e6;
```

Calculate the s-parameters of the cross trace using the behavioral model.

```
Sckt = sparameters(robj,freq,'Behavioral',true);
```

```
Warning: Behavioral model is valid only when EpsilonR is 9.9.
```

Calculate the s-parameters of the cross trace using the electromagnetic solver.

```
Sem = sparameters(robj,freq);
```

References:

1. Ramesh Garg & I. J. Bahl (1978) Microstrip discontinuities, International Journal of Electronics, 45:1, 81-87, DOI: 10.1080/00207217808900883
2. Wadell, Brian C. *Transmission Line Design Handbook*. The Artech House Microwave Library. Boston: Artech House, 1991.

See Also

`traceTee` | `traceRectangular` | `traceSpiral` | `tracePoint` | `traceLine`

Introduced in R2021b

delta

Create delta shape

Description

Use `delta` object to create a delta shape on the X-Y plane.

Creation

Syntax

```
deltashape = delta
deltashape = delta(Name=Value)
```

Description

`deltashape = delta` creates a delta shape on the X-Y plane.

`deltashape = delta(Name=Value)` sets "Properties" on page 1-192 using one or more name-value arguments. For example, `delta(ReferencePoint=[1 1])` creates a delta shape with the reference point at `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of delta shape
`'mydelta'` (default) | character vector | string scalar

Name of the delta shape, specified as a character vector or string scalar.

Example: `deltashape = delta(Name='deltaShape')`

Data Types: `char`

`ReferencePoint` — Reference point of delta shape
`[0 0]` (default) | two-element vector

Reference point of delta shape in Cartesian coordinates, specified as a two-element vector of nonnegative elements. Use the reference point to modify the shape relative to its initial position.

Example: `deltashape = delta(ReferencePoint=[1 1])`

Data Types: `double`

`OuterRadius` — Outer radius of delta
`0.0016` (default) | positive scalar

Outer radius of the delta, specified as a positive scalar in meters.

Example: `shape = delta(OuterRadius=0.0024)`

Data Types: `double`

`InnerRadius` — Inner radius of delta
`0` (default) | nonnegative scalar

Inner radius of the delta, specified as a nonnegative scalar in meters. This value truncates the delta from the tip.

Example: `shape = delta(InnerRadius=0.3)`

Data Types: `double`

`Angle` — Angel of delta
`90` (default) | positive scalar

Angel of the delta shape, specified as a positive scalar in degrees. The value of the angle must be greater than 0 degrees and lesser than 180 degrees.

Example: `shape = delta(Angle=50)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Delta Shape

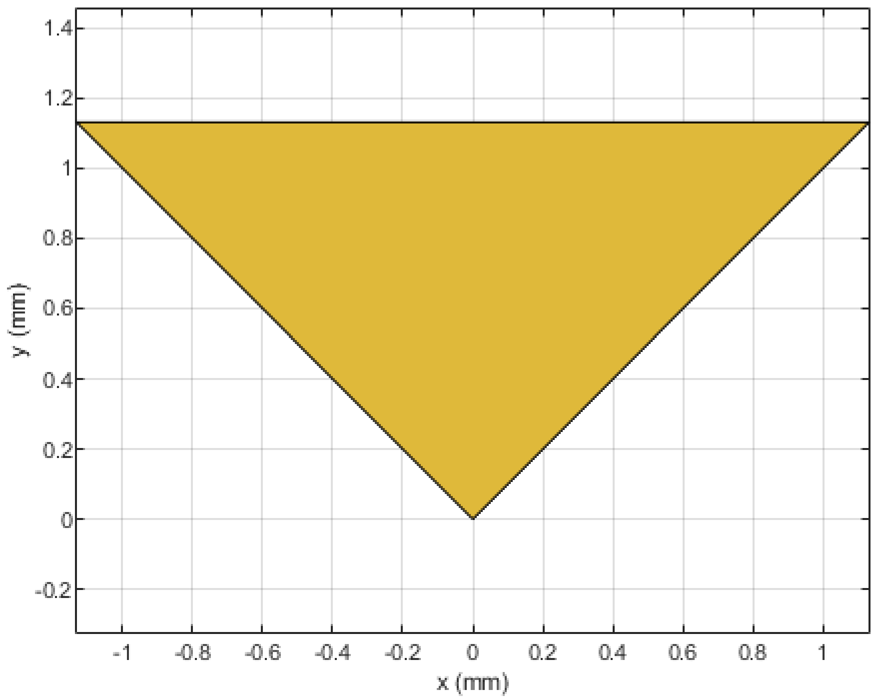
Create a delta shape with default properties.

```
deltashape = delta
```

```
deltashape = 
  delta with properties:

             Name: 'mydelta'
    ReferencePoint: [0 0]
       OuterRadius: 0.0016
       InnerRadius: 0
             Angle: 90
```

View the shape.

```
show(deltashape)
```



See Also

`delta` | `radial`

Introduced in R2021b

radial

Create radial shape

Description

Use the `radial` object to create a radial shape on the X-Y plane.

Creation

Syntax

```
radialshape = radial
radialshape = radial(Name=Value)
```

Description

`radialshape = radial` creates a radial shape on the X-Y plane.

`radialshape = radial(Name=Value)` sets "Properties" on page 1-195 using one or more name-value arguments. For example, `radial(ReferencePoint=[1 1])` creates a radial shape with the reference point at `[1 1]`. Properties not specified retain their default values.

Properties

`Name` — Name of radial shape
`'myradial'` (default) | character vector | string scalar

Name of the radial shape, specified as a character vector or string scalar.

Example: `radialshape = radial(Name='radialShape')`

Data Types: `char` | `string`

`ReferencePoint` — Reference point of radial shape
`[0 0]` (default) | two-element vector

Reference point of radial shape, specified as a two-element vector of nonnegative elements in Cartesian coordinates. Use the reference point to modify the shape relative to its initial position.

Example: `radialshape = radial(ReferencePoint=[1 1])`

Data Types: `double`

`OuterRadius` — Outer radius of radial
`0.0016` (default) | positive scalar

Outer radius of the radial shape, specified as a positive scalar in meters.

Example: `radialshape = radial(OuterRadius=0.0024)`

Data Types: `double`

`InnerRadius` — Inner radius of radial
`0` (default) | positive scalar

Inner radius of the radial shape, specified as a positive scalar in meters. This value truncates the radial from the tip.

Example: `radialshape = radial(InnerRadius=0.4)`

Data Types: `double`

`Angle` — Angel of radial
`90` (default) | positive scalar

Angel of the radial shape, specified as a positive scalar in degrees. The value of the angle must be greater than 0 degrees and lesser than 180 degrees.

Example: `radialshape = radial(Angle=50)`

Data Types: `double`

Object Functions

add	Boolean unite operation on two RF PCB shapes
and	Shape1 & Shape2 for RF PCB shapes
area	Calculate area of RF PCB shape in square meters
intersect	Boolean intersection operation on two RF PCB shapes
mesh	Change and view mesh properties of metal or dielectric in PCB component
minus	Shape1 - Shape2 for RF PCB shapes
plus	Shape1 + Shape2 for RF PCB shapes
rotate	Rotate RF PCB shape about defined axis
rotateX	Rotate RF PCB shape about x-axis
rotateY	Rotate RF PCB shape about y-axis and angle
rotateZ	Rotate RF PCB shape about z-axis
subtract	Boolean subtraction operation on two RF PCB shapes
scale	Change size of RF PCB shape by fixed amount
show	Display PCB component structure or PCB shape
translate	Move RF PCB shape to new location

Examples

Create Default Radial Shape

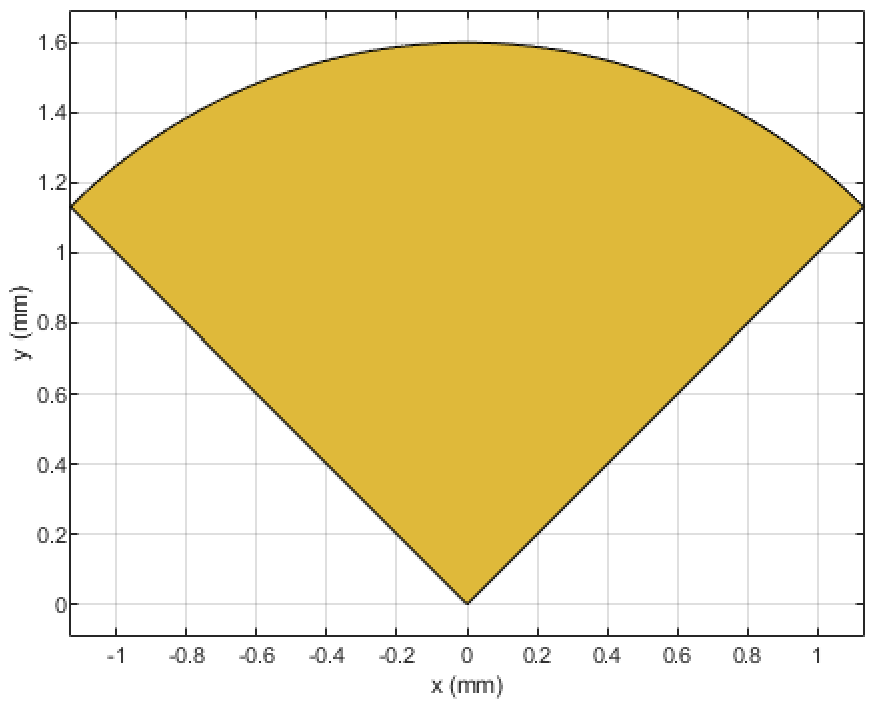
Create a radial shape with default properties.

```
radialshape = radial
```

```
radialshape = 
  radial with properties:

              Name: 'myradial'
    ReferencePoint: [0 0]
       OuterRadius: 0.0016
       InnerRadius: 0
             Angle: 90
```

View the shape.

```
show(radialshape)
```



See Also

`delta`

Introduced in R2021b

PCBConnectors

RF connector at RF PCB feedpoint

Description

Use `PCBConnectors` object to specify RF connectors used for RF printed circuit board (PCB) feed points. The result is generally a set of modifications to the PCB design files. The changes to the PCB include new copper landing pads and traces, and changes to solder mask, silk screen, and solder paste files.

Creation

Syntax

`c = PCBConnectors.connectorType`

Description

`c = PCBConnectors.connectorType` creates Gerber files based on the type of connector to use at the PCB feedpoint specified in `connectorType`.

Input Arguments

`connectorType` — Type of connector from PCB connector package
character vector

Type of connector from PCB connector package, specified as one of the following:

- Coax Connectors - Coax RG11, RG174, RG58, and RG59 connectors directly soldered to PCB pads.
- IPX Connectors - LightHorse IPX SMT jack or plug surface mount RF connector.
- MMCX Connectors - MMCX Cinch or Samtec surface mount RF connectors.
- SMA Connectors - Generic 5-pad SMA surface mount RF connectors, with four corner rectangular pads, one round center pin. Cinch and Multicomp SMA RF connectors.
- SMAEdge Connectors- Generic SMA edge-launch surface mount RF connector. Cinch and Samtec SMA edge-launch RF connectors.
- SMB Connectors - Johnson/Emerson and Pasternack SMB surface mount RF connectors.
- SMC Connectors - Pasternack SMC and SMC edge-launch surface mount RF connectors.
- Coaxial Cable Connectors - Semi-rigid `0.020 inch`, `0.034 inch`, `0.047 inch`, and `0.118 inch` coaxial cable soldered to PCB pads.

For list of connectors, see “PCB Connectors List” on page 1-205.

Example: `c = PCBConnectors.Semi_020` creates Gerber files configured to use semi-rigid `0.020 inch` coaxial cables.

Properties

Common Properties for All Connectors

Type — Type of connector
character vector

This property is read-only.

Type of connector, specified as a character vector.

Example: `'Coax_RG11'`

Data Types: `char` | `string`

`Mfg` — Name of component manufacturer
character vector

This property is read-only.

Name of component manufacturer, specified as a character vector.

Example: `'Belden'`

Data Types: `char` | `string`

`Part` — Manufacturer part number
character vector | string

This property is read-only.

Manufacturer part number, specified as a character vector or string.

Example: `'RG11'`

Data Types: `char` | `string`

`Annotation` — Text added to PCB to identify component
character vector

This property is read-only.

Text added to PCB to identify component, specified as a character vector.

Example: `'RG59U'`

Data Types: `char` | `string`

`Impedance` — Connector impedance
`50` | positive scalar

This property is read-only.

Connector impedance, specified as a positive scalar in ohms.

Example: `c = PCBConnectors.MMCX_Cinch; c.Impedance = 70;`

Data Types: `double`

`Datasheet` — URL for component specifications
character vector

This property is read-only.

URL for component specifications, specified as a character vector. Data sheets are typically PDF files.

Data Types: `char` | `string`

`Purchase` — URL for purchasing connector
character vector

This property is read-only.

URL for purchasing connector, specified as a character vector.

Data Types: `char` | `string`

Common Properties for All Coax Connectors

`PinDiameter` — Circular pad diameter
positive scalar

Circular pad diameter connecting the signal wire of the coax to the feedpoint, specified as a positive scalar in meters. The pin diameter is greater than the diameter of the signal wire.

Example: `c = PCBConnectors.Coax_RG59; c.PinDiameter = 1.0000e-03;`

Data Types: `double`

`DielectricDiameter` — Dielectric diameter
positive scalar

Dielectric diameter (white material around signal wire), specified as a positive scalar in meters. Dielectric diameter specifies the size of the non-conductive isolation ring on the PCB between the signal wire and the ground plane.

Example: `c = PCBConnectors.Coax_RG59; c.DielectricDiameter = 0.0073;`

Data Types: `double`

`ShieldDiameter` — Ground ring diameter
positive scalar

Ground ring diameters used to solder coax shield, specified as a positive scalar in meters.

Example: `c = PCBConnectors.Coax_RG59; c.ShieldDiameter = 0.0085;`

Data Types: `double`

`AddThermals` — Thermal relief
`1` | `0`

Thermal relief around coaxial shield connection, specified as `0` or `1`. Thermal relief reduces the heat needed to solder the coax shield to the ground.

Example: `c = PCBConnectors.Coax_RG59; c.AddThermals = 0;`

Data Types: `logical`

`ThermalsDiameter` — Arc-shaped gaps outer diameter
positive scalar

Arc-shaped gaps outer diameter in the ground plane, specified as a positive scalar in meters.

Example: `c = PCBConnectors.Coax_RG59; c.ThermalsDiameter = 0.0100;`

Data Types: `double`

`ThermalsBridgeWidth` — Width of four conductive bridges
positive scalar

Width of four conductive bridges created across thermal gap, specified as a positive scalar in meters. The bridges are established during electrical grounding.

Example: `c = PCBConnectors.Coax_RG59; c.ThermalBridgeWidth = 0.0015;`

Data Types: `double`

Common Properties for All 5-Pad Symmetric Surface Mount Connectors

`TotalSize` — Total length of each side of rectangular connector footprint
two-element vector

Total length of each side of rectangular connector footprint, specified as a two-element vector with each element unit in meters.

Example: `c = PCBConnectors.SMA_Multicomp; c.TotalSize = [0.0063 0.0063];`

Data Types: `double`

`GroundPadSize` — Length of each side of ground pad
two-element vector

Length of each side of ground pad, specified as a two-element vector with each element unit in meters. The pads are located in each of the four corners of the connector footprint.

Example: `c = PCBConnectors.SMA_Multicomp; c.GroundPadSize = [0.0016 0.0016];`

Data Types: `double`

`SignalPadDiameter` — Circular pad diameter
positive scalar

Circular pad diameter connecting the signal pin of the coax connector, specified as a positive scalar in meters. The pad is at the center of the connector footprint.

Example: `c = PCBConnectors.SMA_Multicomp; c.SignalPadDiameter = 0.0012;`

Data Types: `double`

`PinHoleDiameter` — Via pin diameter
positive scalar

Via pin diameter, specified as a positive scalar in meters.

Example: `c = PCBConnectors.SMA_Multicomp; c.ViaPinDiameter = 0.0012;`

Data Types: `double`

`IsolationRing` — Diameter of isolation ring that removes semicircle of copper from inner corner of ground pads
scalar

Diameter of isolation ring that removes semicircle of copper from inner corner of ground pads, specified as a scalar in meters.

Example: `c = PCBConnectors.SMA_Multicomp; c.IsoltationRing = 0.0012;`

Data Types:

`VerticalGroundStrips` — Vertical ground strips between upper and lower ground pads
scalar

Vertical ground strips between upper and lower ground pads, specified as a scalar.

Example: `c = PCBConnectors.SMA_Multicomp; c.VerticalGroundStrips = 1;`

Data Types: `double`

Common Properties for All Edge-Launch Surface Mount Connectors

`GroundPadSize` — Ground pad size
two-element vector

Ground pad size, specified as a two-element vector with each element unit in meters.

Example: `c = PCBConnectors.SMAEdge; c.GroundPadSize = [0.0014 0.0042];`

Data Types: `double`

`GroundSeparation` — Space between ground pads
positive scalar

Space between ground pads on the ground side of the board, specified as a positive scalar in meters.

Example: `c = PCBConnectors.SMAEdge; c.GroundSeparation = 0.0043;`

Data Types: `double`

`GroundPadIsolation` — Width of copper removed around top layer ground pads
positive scalar

Width of copper removed around top layer ground pads, specified as a positive scalar in meters. This property isolates the ground pads from any signal traces or structures.

Example: `c = PCBConnectors.SMAEdge; c.GroundPadIsolation = 2.5000e-04;`

Data Types: `double`

`SignalPadSize` — Signal pad size
two-element vector

Signal pad size, specified as a two-element vector with each element unit in meters.

Example: `c = PCBConnectors.SMAEdge; c.SignalPadSize = [0.0013 0.0036];`

Data Types: `double`

`SignalGap` — Gap between PCB edge and start of signal pad copper
positive scalar

Gap between PCB edge and start of signal pad copper, specified as a positive scalar in meters.

Example: `c = PCBConnectors.SMAEdge; c.SignalGap = 1.0000e-04;`

Data Types: `double`

`SignalLineWidth` — Width of signal trace
positive scalar

Width of signal trace extending from the signal pad to the feedpoint location, specified as a positive scalar in meters.

Example: `c = PCBConnectors.SMAEdge; c.SignalLineWidth = 8.0000e-04;`

Data Types: `double`

`EdgeLocation` — PCB side that receives edge connector
`'north'` | `'south'` | `'east'` | `'west'`

PCB side that receives edge connector, specified as `'north'`, `'south'`, `'east'`, `'west'`.

Example: `c = PCBConnectors.SMAEdge; c.EdgeLocation = 'south';`

Data Types: `char`

`EdgeBoardProfile` — Extend PCB to add connector beyond design area
`0` | `1`

Extend PCB to add connector beyond design area, specified as `0` or `1`

Example: `c = PCBConnectors.SMAEdge; c.EdgeBoardProfile = 1;`

Data Types: `logical`

`FillGroundSide` — Fill connector region on ground side of board with copper
`0` | `1`

Fill connector region on ground side of the board with copper, specified as `0` or `1`

Example: `c = PCBConnectors.SMAEdge; c.FillGroundSide = 1;`

Data Types: `logical`

Common Properties for All Staggered Surface Mount Connectors

`GroundPadSize` — Ground pad size
two-element vector

Ground pad size, specified as a two-element vector with each element unit in meters.

Example: `c = PCBConnectors.IPX_Plug_Lighthorse; c.GroundPadSize = [0.0010 0.0022];`

Data Types: `double`

`GroundPadXSeparation` — Distance between pair of ground pads along X-axis
positive scalar

Distance between pair of ground pads along X-axis, specified as a positive scalar in meters.

Example: `c = PCBConnectors.IPX_Plug_Lighthorse; c.GroundPadXSeparation = 0.0019;`

Data Types: double

GroundPadYOffset — Y-offset from signal pad to signal pad center line
positive scalar

Y-offset from signal pad to signal pad center line, specified as a positive scalar in meters.

Example: c = PCBConnectors.IPX_Plug_Lighthorse; c.GroundPadYOffset = 0.0015;

Data Types: double

SignalPadSize — Signal pad size
2-element vector

Signal pad size, specified as a 2-element vector with each element unit in meters.

Example: c = PCBConnectors.IPX_Plug_Lighthorse; c.SignalPadSize = [1.0000e-03 1.0000e-03];

Data Types: double

SignalMinYSeparation — Minimum separation from ground at bottom or top for signal pad
positive scalar

Minimum separation from ground at bottom or top for signal pad, specified as a positive scalar in meters.

Example: c = PCBConnectors.IPX_Plug_Lighthorse; c.SignalMinYSeparation = 1.0000e-03;

Data Types: double

Examples

Authoring Custom RF Connector

This example shows how to define custom RF connector class.

```
classdef SMA_Jack_Cinch < PCBConnectors.BaseSMT5PadSymmetric
    % Cinch SMA surface mount RF connector.

    properties (Constant) % Abstract
        Type       = 'SMA'
        Mfg        = 'Cinch'
        Part       = '142-0701-631'
        Annotation = 'SMA'
        Impedance  = 50
        Datasheet  = 'http://www.farnell.com/datasheets/1720451.pdf?_ga=2.164811836.2075200750.14
        Purchase   = 'http://www.newark.com/johnson/142-0701-631/rf-coaxial-sma-jack-straight-50/
    end

    methods
        function RFC = SMA_Jack_Cinch
            RFC.TotalSize           = [0.5 0.5]*25.4e-3;
            RFC.GroundPadSize       = [0.102 0.102]*25.4e-3;
            RFC.SignalPadDiameter   = 0.1*25.4e-3;
            RFC.PinHoleDiameter     = 1.27e-3;
```

```
            RFC.IsolationRing       = 0.22*25.4e-3;
            RFC.VerticalGroundStrips = false;
        end
    end
end
```

Generate Gerber Format Files for Default PCB Component

Create a default PCB component.

```
p = pcbComponent;
```

Use 2 Cinch SMA connectors and the Mayhew Labs PCB viewer.

```
W = PCBServices.MayhewWriter;
C1 = PCBConnectors.SMA_Cinch;
C2 = PCBConnectors.SMA_Cinch;
```

Generate the Gerber-format files.

```
[A,g] = gerberWrite(p,W,{C1,C2})
```

```
A =
  PCBWriter with properties:

                      Design: [1×1 struct]
                      Writer: [1×1 PCBServices.MayhewWriter]
                   Connector: {[1×1 PCBConnectors.SMA_Cinch]  [1×1 PCBConnectors.SMA_Cinch]}
         UseDefaultConnector: 0
   ComponentBoundaryLineWidth: 8
        ComponentNameFontSize: []
           DesignInfoFontSize: []
                        Font: 'Arial'
                   PCBMargin: 5.0000e-04
                  Soldermask: 'both'
                 Solderpaste: 1

   See info for details

g =
'C:\Users\vgopalak\OneDrive - MathWorks\Documents\MATLAB\Examples\rfpcb-ex06685827\untitled'
```

More About

PCB Connectors List

PCB Connectors	Descriptions
PCBConnectors.Coax_RG11	RG11 coaxial cable direct soldered to PCB pads.
PCBConnectors.Coax_RG58	RG58 coaxial cable direct soldered to PCB pads.
PCBConnectors.Coax_RG59	RG59 coaxial cable direct soldered to PCB pads.
PCBConnectors.Coax_RG174	RG174 coaxial cable direct soldered to PCB pads.

PCB Connectors	Descriptions
PCBConnectors.SMA	Generic 5-pad SMA surface mount RF connector, with four corner rectangular ground pads, one round.
PCBConnectors.SMAEdge	Generic SMA edge-launch surface mount RF connector.
PCBConnectors.SMACinch	Cinch SMA surface mount RF connector
PCBConnectors.SMAEdge_Cinch	Cinch SMA edge-launch surface mount RF connector
PCBConnectors.SMAEdge_Samtec	Samtec SMA edge-launch surface mount RF connector
PCBConnectors.SMAEdge_Amphenol	Amphenol SMA edge-launch surface mount RF connector
PCBConnectors.SMAEdge_Linx	Linx SMA edge-launch surface mount RF connector
PCBConnectors.SMA_Multicomp	Multicomp SMA surface mount RF connector
PCBConnectors.SMB_Johnson	Johnson/Emerson SMB surface mount RF connector
PCBConnectors.SMB_Pasternack	Pasternack SMB surface mount RF connector
PCBConnectors.SMC_Pasternack	Pasternack SMC surface mount RF connector
PCBConnectors.SMCEdge_Pasternack	Pasternack SMC edge-launch surface mount RF connector
PCBConnectors.MMCX_Cinch	Cinch MMCX surface mount RF connector
PCBConnectors.MMCX_Samtec	Samtec MMCX surface mount RF connector
PCBConnectors.IPX_Jack_LightHorse	LightHorse IPX SMT jack surface mount RF connector
PCBConnectors.IPX_Plug_LightHorse	LightHorse IPX SMT plug surface mount RF connector
PCBConnectors.UFL_Hirose	Hirose u.fl surface mount RF connector
PCBConnectors.Semi_020	Pasternack semi-rigid 0.020" coaxial cable soldered to PCB pads
PCBConnectors.Semi_034	Pasternack semi-rigid 0.020" coaxial cable soldered to PCB pads
PCBConnectors.Semi_047	Pasternack semi-rigid 0.047" coaxial cable soldered to PCB pads
PCBConnectors.Semi_118	Pasternack semi-rigid 0.118" coaxial cable soldered to PCB pads

See Also

`PCBWriter` | `PCBServices` | `gerberWrite`

Introduced in R2021b

PCBReader

Import and update Gerber files

Description

Use the `PCBReader` object to create a printed circuit board (PCB) reader to import Gerber files and to facilitate the creation of a PCB model. A Gerber file is a set of manufacturing files used to describe a PCB. A Gerber file uses an ASCII vector format to describe 2-D binary images.

Creation

You can create a `PCBReader` object using the following methods:

- `gerberRead` — Create a `PCBReader` object with the specified Gerber and drill files.
- The `PCBReader` function described here.

Syntax

```
B = PCBReader(S)
B = PCBReader(Name=Value)
```

Description

`B = PCBReader(S)` creates a `PCBReader` object that imports multilayer PCB design files described in `S`.

Note The `PCBReader` object reads RS-274X Gerber files. It does not support RS-274D Gerber files.

`B = PCBReader(Name=Value)` sets “Properties” on page 1-208 using name-value arguments. `Name` is the property name and `Value` is the corresponding value. You can specify several name-value pair arguments in any order as Name1,Value1,...,NameN,ValueN. Properties not specified retain their default values. For example, `B = PCBReader('StackUp',S,'Drillfile','ant.txt')` imports the layer and drill files into the `PCBReader`.

Input Arguments

S — PCB stackup definition
`stackUp` object

PCB stackup definition, specified as a `stackUp` object.

Example: `S = stackUp; B = PCBReader(S)`

Example: `B = PCBReader('StackUp',S)`

Properties

`StackUp` — PCB stackup definition
`stackUp` object

PCB stackup definition, specified as a `stackUp` object.

Example: `S = stackUp; B.StackUp = S;`

Example: `B = PCBReader('StackUp',S)`

`DrillFile` — Name of Excellon drill file
[ ] (default) | character vector | string scalar

Name of Excellon drill file, specified as a character vector or string scalar. You can specify either a DRL or a TXT file.

Example: `B.DrillFile = 'ant.drl'`

`NumPointsOnCurves` — Discretization points on curved segments
`50` (default) | positive scalar

Discretization points on curved segments, specified as a positive scalar.

Example: `B.NumPointsOnCurves = 80`

Examples

Import Gerber Files Using Stackup Definition

Create a PCB stack up definition object using default properties.

```
S = stackUp;
```

Set the thickness of the dielectric Air in layer 1 to 0.1 mm.

```
S.Layer1.Thickness = 0.1e-3;
```

Import a top layer Gerber file to layer 2.

```
S.Layer2 = 'interdigital_Capacitor.gtl';
```

Create a `PCBReader` object using the `stackUp` object, `S`.

```
p = PCBReader('StackUp',S);
```

To update the Gerber file, convert the `PCBReader` object to a `pcbComponent` object.

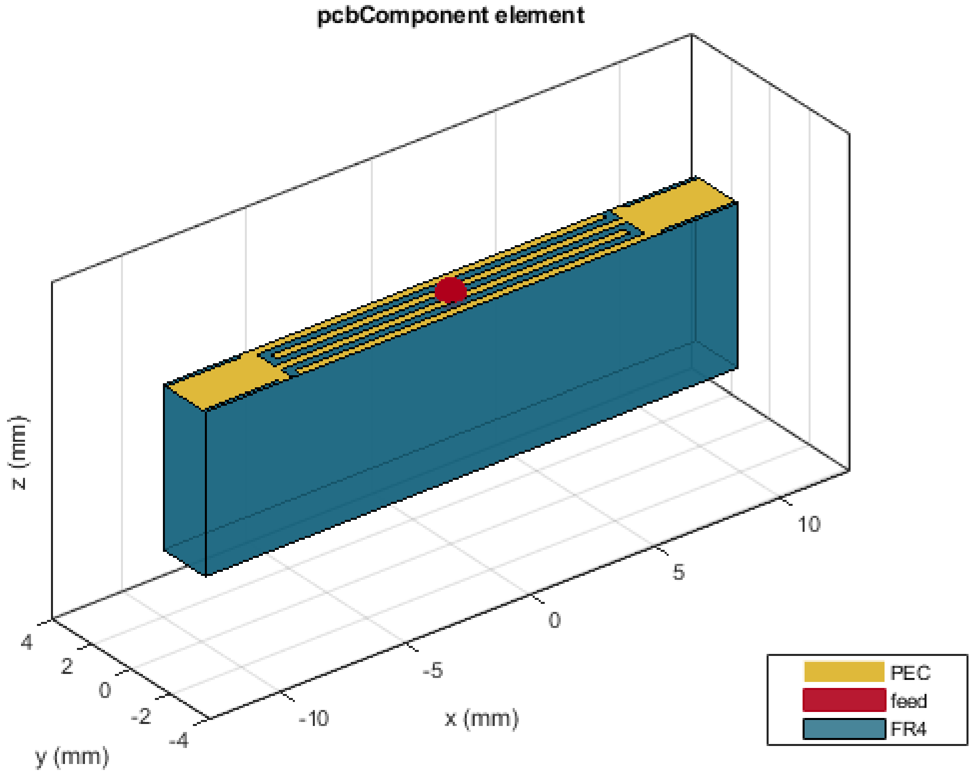
```
pcbcapacitor = pcbComponent(p);
pcbcapacitor.FeedDiameter = 0.001
```

```
pcbcapacitor = 
  pcbComponent with properties:

              Name: 'interdigital_Capacitor'
          Revision: 'v1.0'
        BoardShape: [1×1 antenna.Rectangle]
```

```
    BoardThickness: 0.0062
            Layers: {[1×1 dielectric]  [1×1 antenna.Polygon]  [1×1 dielectric]  [1×1 dielectric]
     FeedLocations: [0 0 2]
      FeedDiameter: 1.0000e-03
      ViaLocations: []
       ViaDiameter: []
      FeedViaModel: 'square'
         Conductor: [1×1 metal]
              Tilt: 0
          TiltAxis: [0 0 1]
              Load: [1×1 lumpedElement]
```

View the PCB component in the Gerber file.

```
show(pcbcapacitor)
```



See Also

PCBWriter | PCBServices | PCBConnectors | stackUp | gerberRead

Introduced in R2021b

PCBServices

Customize PCB file generation for PCB manufacturing service

Description

Use the `PCBServices` object to customize printed circuit board (PCB) file generation for a PCB manufacturing service.

Creation

Syntax

`w = PCBServices.serviceType`

Description

`w = PCBServices.serviceType` creates a Gerber file based on the type of service specified in `serviceType`.

Input Arguments

`serviceType` — Type of service from PCB services package
character vector

Type of service from PCB services package, specified as one of the following:

- AdvancedCircuitsWriter – Configure Gerber file generation for Advanced Circuits manufacturing.
- CircuitPeopleWriter – Configure Gerber file generation for CircuitPeople online viewer.
- DirtyPCBsWriter – Configure Gerber file generation for Dirty PCBs manufacturing.
- EuroCircuitsWriter – Configure Gerber file generation for EuroCircuits online viewer.
- GerberLookWriter – Configure Gerber file generation for GerbLook online viewer.
- GerberViewerWriter – Configure Gerber file generation for GerberViewer online viewer.
- MayhewWriter – Configure Gerber file generation for Mayhew Labs online 3-D viewer.
- OSHParkWriter – Configure Gerber file generation for OSH Park PCB manufacturing.
- PCBWayWriter – Configure Gerber file generation for PCBWay PCB manufacturing.
- ParagonWriter – Configure Gerber file generation for Paragon Robotics online viewer.
- SeeedWriter – Configure Gerber file generation for Seeed Fusion PCB manufacturing.
- SunstoneWriter – Configure Gerber file generation for Sunstone PCB manufacturing.
- ZofzWriter – Configure Gerber file generation for Zofz 3-D viewer.

Example: `w = PCBServices.SunstoneWriter` creates Gerber files configured to use Sunstone PCB manufacturing service.

Output Arguments

`w` — PCB manufacturing service
object

PCB manufacturing service, returned as an object.

Properties

`BoardProfileFile` — File type for board profile
`'legend'` | `'profile'`

File type for board profile, specified as `'legend'` or `'profile'`.

Example: `w = PCBServices.SunstoneWriter`; `w.BoardProfileFile = 'profile'`.

Data Types: `char` | `string`

`BoardProfileLineWidth` — Width of line
`1` | positive scalar

Width of line, specified as a positive scalar in mils.

PCB manufacturers vary on board profile. The most common line width is zero of a fraction width in the chosen unit, for example, 0.1 mil.

Example: `w = PCBServices.SunstoneWriter`; `w.BoardProfileLineWidth = 0.1`

Data Types: `double`

`CoordPrecision` — Precision of X and Y coordinates written to file
`[2 6]` | 1-by-2 vector

Precision of X and Y coordinates written to file, specified as a 1-by2 vector [*I F*], where,

- *I* – Number of digits in the integer part, $0<=I<=6$.
- *F* – Number of digits in the fractional part, $4<=F<=6$.

Example: `w = PCBServices.SunstoneWriter`; `w.CoordPrecision = [1 3]`

Data Types: `double`

`CoordUnits` — Units of X and Y coordinate
`'in'` | `'mm'`

Units of X and Y coordinates, specified as inches or millimeters.

Example: `w = PCBServices.SunstoneWriter`; `w.CoordUnits = 'mm'`

Data Types: `char` | `string`

`CreateArchiveFile` — Creates single archive file with all Gerber files
`1` (default) | `0`

Creates single archive file with all Gerber files, specified as `1` or `0`.

Example: `w = PCBServices.SunstoneWriter`; `w.CreateArchiveFile = 0`

Data Types: `logical`

DefaultViaDiameter — Via drill diameter
3.0000e-04 | positive scalar

Via drill diameter, specified as a positive scalar in meters. PCB manufacturers also call it minimum drilling hole diameter.

Example: `w = PCBServices.SunstoneWriter`; `w.DefaultViaDiameter = 0.1`

Data Types: `double`

DrawArcsUsingLines — Force arcs to be drawn using lines
0 | 1

Force arcs to be drawn using lines, specified as `1` or `0`.

Example: `w = PCBServices.SunstoneWriter`; `w.DrawArcsUsingLines = 0`

Data Types: `logical`

ExtensionLevel — Feature content for Gerber file format
1 (default) | 2

Feature content for Gerber file format, specified as:

- `1` - Extension 1 is the most compatible setting for downstream PCB manufacturing tools.
- `2` - Extension 2 adds file attributes `%TF.<attr>*%"` to the header and footer of Gerber files.

Example: `w = PCBServices.SunstoneWriter`; `w.ExtensionLevel = 2`

Data Types: `double`

Filename — Name of all files containing Gerber design
'untitled' (default) | character vector

Name of all files containing Gerber design, specified as a character vector.

Example: `w = PCBServices.SunstoneWriter`; `w.Filename = 'pcb_design'`.

Data Types: `char` | `string`

Files — Define stack of PCB files
character vector

Define stack of PCB files, specified as a character vector. This definition includes:

- Multiples files describing one PCB.
- A "file" as a memory object containing buffers that describe or hold the file content before the file is written.
- Cell vector of `Gerber.FileFunction` objects, one per file.

Data Types: `cell` | `char` | `string`

IncludeRootFolderInZip — Include top-level folder in zip archive
1 | 0

Include top-level folder in zip archive, specified as `1` or `0`.

Example: `w = PCBServices.SunstoneWriter`; `w.IncludeRootFolderInZip = 0`

Data Types: `logical`

`PostWriteFcn` — Function to invoke after a successful write operation
function handle (default)

Function to invoke after a successful write operation, specified as a function handle. In this case, it is the `sendTo` function. This property makes sure that the location of the Gerber files and the website of the manufacturing service is open after a successful write function.

Example: `w = PCBServices.SunstoneWriter`; `w.PostWriteFcn = @(obj)sendTo(obj)`

Data Types: `function_handle`

`SameExtensionForGerberFiles` — Use `.gbr` to be file extension for all Gerber files
`0` | `1`

Use `.gbr` to be file extension for all Gerber files, specified as `0` or `1`.

Example: `w = PCBServices.SunstoneWriter`; `w.SameExtensionForGerberFiles = 1`

Data Types: `logical`

`UseExcellon` — Generate Excellon drill files
`1` | `0`

Generate Excellon drill files, specified as `0` or `1`.

Example: `w = PCBServices.SunstoneWriter`; `w.UseExcellon = 1`, generates Gerber format drill files with `'x2'` extension.

Data Types: `logical`

Examples

Generate Gerber Format Files for Default PCB Component

Create a default PCB component.

```
p = pcbComponent;
```

Use 2 Cinch SMA connectors and the Mayhew Labs PCB viewer.

```
W = PCBServices.MayhewWriter;
C1 = PCBConnectors.SMA_Cinch;
C2 = PCBConnectors.SMA_Cinch;
```

Generate the Gerber-format files.

```
[A,g] = gerberWrite(p,W,{C1,C2})
```

```
A = 
  PCBWriter with properties:

                      Design: [1×1 struct]
                      Writer: [1×1 PCBServices.MayhewWriter]
                   Connector: {[1×1 PCBConnectors.SMA_Cinch]  [1×1 PCBConnectors.SMA_Cinch]}
         UseDefaultConnector: 0
    ComponentBoundaryLineWidth: 8
```

```
         ComponentNameFontSize: []
            DesignInfoFontSize: []
                          Font: 'Arial'
                     PCBMargin: 5.0000e-04
                    Soldermask: 'both'
                   Solderpaste: 1

   See info for details

g = 
'C:\Users\vgopalak\OneDrive - MathWorks\Documents\MATLAB\Examples\rfpcb-ex06685827\untitled'
```

See Also

PCBWriter | PCBConnectors | gerberWrite

Introduced in R2021b

PCBWriter

Create PCB board definitions from 2-D PCB designs

Description

Use the `PCBWriter` object to create a printed circuit board (PCB) design files based on multilayer 2-D PCB design. A set of manufacturing files known as Gerber files describes a PCB. A Gerber file uses an ASCII vector format for 2-D binary images.

Creation

Syntax

```
b = PCBWriter(pcbcomponentObject)
b = PCBWriter(pcbcomponentObject,rfConnector)
b = PCBWriter(pcbcomponentObject,writer)
b = PCBWriter(pcbcomponentObject,rfConnector,writer)
```

Description

`b = PCBWriter(pcbcomponentObject)` creates a `PCBWriter` object that generates Gerber-format PCB design files based on a 2-D PCB design geometry using PCB stack.

`b = PCBWriter(pcbcomponentObject,rfConnector)` creates a customized PCB file using specified `rfConnector` type.

`b = PCBWriter(pcbcomponentObject,writer)` creates a customized PCB file using a specified PCB service, `writer`.

`b = PCBWriter(pcbcomponentObject,rfConnector,writer)` creates customised PCB file using specified PCB service and PCB connector type.

Input Arguments

`pcbcomponentObject` — Single feed PCB
`pcbComponent` object

Single feed PCB, specified as a `pcbComponent` object.

Example: `p1 = pcbComponent` creates a PCB component object, `p1` . `a = PCBWriter(p1)` uses `p1` to create a `PCBWriter` object `a`.

`writer` — PCB service to view PCB design
`PCBServices` object

PCB service to view PCB design, specified as a `PCBServices` object.

Example: `s = PCBServices.MayhewWriter; a = PCBWriter(p1,s)` uses Mayhew Labs PCB service to view the PCB design.

`rfConnector` — RF connector type
`PCBConnectors` object

RF connector type for PCB feedpoint, specified as a `PCBConnectors` object.

Example: `c = PCBConnectors.SMA_Cinch; a = PCBWriter(p1,c)` uses SMA_Cinch RF connector at feedpoint.

Properties

`UseDefaultConnector` — Use default connector
`1` (default) | `0`

Use default connector, specified as `0` or `1`.

Example: `a.UseDefaultConnector = 1`, where `a` is a `PCBWriter` object.

Data Types: `logical`

`ComponentBoundaryLineWidth` — Line widths drawn around components on silk screens
`8` (default) | positive scalar

Line widths drawn around components on silk screens, specified as a positive scalar in mils.

Example: `a.ComponentBoundaryLineWidth = 10`, where `a` is a `PCBWriter` object.

Data Types: `double`

`ComponentNameFontSize` — Font size to label components on silk screen
positive scalar

Font size to label components on silk screen, specified as a positive scalar in points.

Example: `a.ComponentNameFontSize = 12`, where `a` is a `PCBWriter` object.

Data Types: `double`

`DesignInfoFontSize` — Font size for design informzation added outside board profile
positive scalar

Design information text font size added outside board profile, specified as a positive scalar.

Example: `a.DesignInfoFontSize = 12`, where `a` is a `PCBWriter` object.

Data Types: `double`

`Font` — Font used for component name and design info
`'Arial'` (default) | character vector

Font used for component name and design info, specified as a character vector.

Example: `a.Font = 'TimesNewRoman'`, where `a` is a `PCBWriter` object.

Data Types: `char` | `string`

`PCBMargin` — Copper free margin around board
`0.5e-3` (default) | positive scalar

Copper free margin around board, specified as a positive scalar in meters.

Example: a.PCBMargin = 0.7e-3, where a is a PCBWriter object.

Data Types: double

SolderMask — Add solder mask to top and bottom of PCB
'both' (default) | 'top' | 'bottom' | 'none'

Add solder mask to top and bottom of PCB, specified as 'both', 'top', 'bottom' or 'none'.

Example: a.SolderMask = 'top', where a is a PCBWriter object.

Data Types: char | string

SolderPaste — Generate solder paste files
1 (default) | 0

Generate solder paste files as a part of PCB stack, specified as 1 or 0.

Example: a.SolderPaste = 0, where a is a PCBWriter object.

Data Types: logical

Object Functions

gerberWrite   Generate Gerber files

Examples

Create PCB Component Design File Using Mayhew Manufacturing Services

Create a coplanar waveguide.

```
cpw = coplanarWaveguide
```

```
cpw = 
  coplanarWaveguide with properties:

             Length: 0.0231
              Width: 0.0039
            Spacing: 2.0000e-04
         ViaSpacing: [0.0011 0.0070]
        ViaDiameter: 5.0000e-04
             Height: 0.0016
    GroundPlaneWidth: 0.0300
          Substrate: [1×1 dielectric]
          Conductor: [1×1 metal]
```

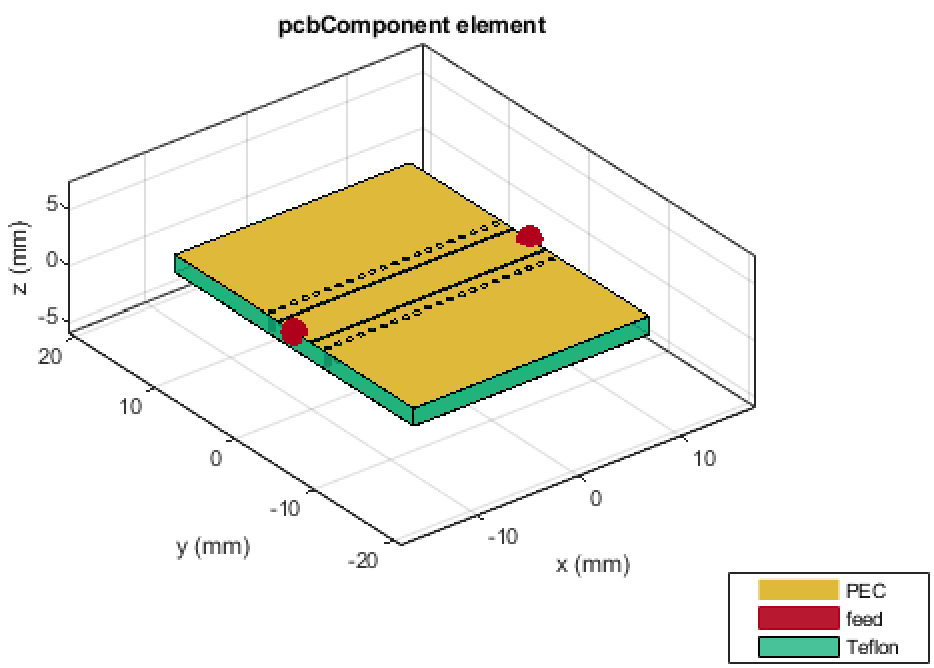
Use this waveguide to create a pcbComponent object.

```
p = pcbComponent(cpw);
p.Name = 'Coplanar Waveguide'
```

```
p = 
  pcbComponent with properties:

             Name: 'Coplanar Waveguide'
         Revision: 'v1.0'
```

```
        BoardShape: [1×1 antenna.Rectangle]
    BoardThickness: 0.0016
            Layers: {[1×1 antenna.Polygon]  [1×1 dielectric]  [1×1 antenna.Rectangle]}
     FeedLocations: [2×4 double]
      FeedDiameter: 0.0019
      ViaLocations: [42×4 double]
       ViaDiameter: 5.0000e-04
      FeedViaModel: 'strip'
         Conductor: [1×1 metal]
              Tilt: 0
          TiltAxis: [0 0 1]
              Load: [1×1 lumpedElement]
```

```
show(p)
```



Use an SMA_Cinch as an RF connector and Mayhew Writer as a 3-D viewer.

```
c = PCBConnectors.SMA_Cinch
```

```
c =
  SMA_Cinch with properties:

                  Type: 'SMA'
                   Mfg: 'Cinch'
                  Part: '142-0711-202'
            Annotation: 'SMA'
             Impedance: 50
```

```
              Datasheet: 'https://belfuse.com/resources/Johnson/drawings/dr-142-0711-202.pdf'
               Purchase: 'https://www.digikey.com/product-detail/en/cinch-connectivity-solutions
              TotalSize: [0.0071 0.0071]
          GroundPadSize: [0.0024 0.0024]
      SignalPadDiameter: 0.0017
        PinHoleDiameter: 0.0013
          IsolationRing: 0.0041
    VerticalGroundStrips: 1

   Cinch 142-0711-202 (Example Purchase)
```

```
s = PCBServices.MayhewWriter
```

```
s =
  MayhewWriter with properties:

              BoardProfileFile: 'legend'
         BoardProfileLineWidth: 1
                CoordPrecision: [2 6]
                    CoordUnits: 'in'
             CreateArchiveFile: 0
                DefaultViaDiam: 3.0000e-04
            DrawArcsUsingLines: 1
                ExtensionLevel: 1
                      Filename: 'untitled'
                         Files: {}
         IncludeRootFolderInZip: 0
                  PostWriteFcn: @(obj)sendTo(obj)
    SameExtensionForGerberFiles: 0
                   UseExcellon: 1
```

Create a PCB component design file.

```
PW = PCBWriter(p,s,c)
```

```
PW =
  PCBWriter with properties:

                       Design: [1×1 struct]
                       Writer: [1×1 PCBServices.MayhewWriter]
                    Connector: {[1×1 PCBConnectors.SMA_Cinch]  [1×1 PCBConnectors.SMA_Cinch]}
          UseDefaultConnector: 0
    ComponentBoundaryLineWidth: 8
        ComponentNameFontSize: []
           DesignInfoFontSize: []
                         Font: 'Arial'
                    PCBMargin: 5.0000e-04
                   Soldermask: 'both'
                  Solderpaste: 1

   See info for details
```

See Also

PCBServices | PCBConnectors

Introduced in R2021b

stackUp

Create PCB stackup definition

Description

Use the `stackUp` object to create a printed circuit board (PCB) stackup definition to import Gerber files. A Gerber file is a set of manufacturing files used to describe a PCB. A Gerber file uses an ASCII vector format for 2-D binary images.

Creation

Syntax

`s = stackUp`

Description

`s = stackUp` creates a default PCB stackup object with five layers. Specify Gerber files as inputs to the second and fourth layers. Specify dielectric material objects as inputs to layers one, three, and five.

Properties

`NumLayers` — Number of layers in stackup
5 (default) | positive scalar

This property is read-only.

Number of layers in the stackup, returned as a positive scalar.

`Layer1` — First layer in stackup
`'Air'` (default) | `dielectric` object

First layer in the stackup definition object, specified as a `dielectric` object.

Example: `s = stackUp; d = dielectric('RO4725JXR'); s.Layer1 = d;`

`Layer2` — Second layer in stackup
character vector | string scalar

Second layer in the stackup definition object, specified as a character vector or string. The file should be saved as a GTL, GBL, or GBR file.

Example: `s = stackUp; s.Layer2 = 'antenna_design_file.gtl';`

Note The Gerber file must be imported to the MATLAB® workspace before setting this property.

Layer3 — Third layer in stackup
'FR4' (default) | dielectric object

Third layer in the stackup definition object, specified as a `dielectric` object.

Example: `s = stackUp; d = dielectric('RO4725JXR'); s.Layer3 = d;`

Layer4 — Fourth layer in stackup
character vector | string scalar

Fourth layer in the stackup definition object, specified as a character vector or string. The file should be saved as a GTL, GBL, or GBR file.

Example: `s = stackUp; s.Layer4 = 'antenna_design_file.gbl';`

Note The Gerber file must be imported to the MATLAB workspace before setting this property.

Layer5 — Fifth layer in stackup
'Air' (default) | dielectric object

Fifth layer in the stackup definition object, specified as a `dielectric` object.

Example: `s = stackUp; d = dielectric('RO4725JXR'); s.Layer5 = d;`

Examples

Import Gerber Files Using Stackup Definition

Create a PCB stack up definition object using default properties.

```
S = stackUp;
```

Set the thickness of the dielectric Air in layer 1 to 0.1 mm.

```
S.Layer1.Thickness = 0.1e-3;
```

Import a top layer Gerber file to layer 2.

```
S.Layer2 = 'interdigital_Capacitor.gtl';
```

Create a `PCBReader` object using the `stackUp` object, `S`.

```
p = PCBReader('StackUp',S);
```

To update the Gerber file, convert the `PCBReader` object to a `pcbComponent` object.

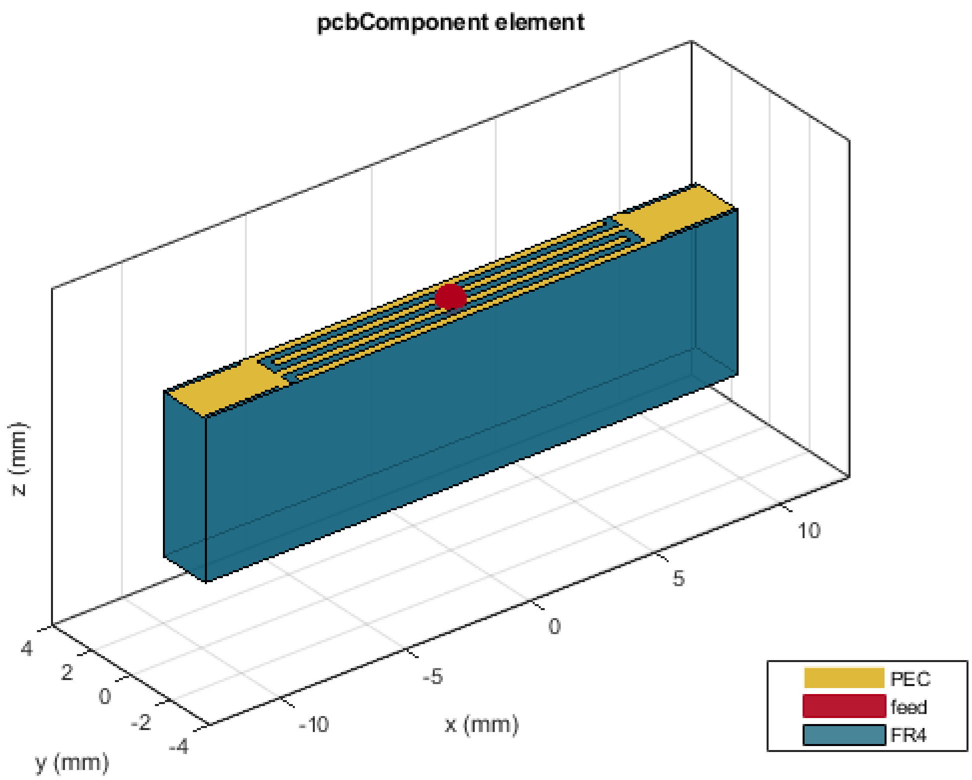
```
pcbcapacitor = pcbComponent(p);
pcbcapacitor.FeedDiameter = 0.001
```

```
pcbcapacitor = 
  pcbComponent with properties:

              Name: 'interdigital_Capacitor'
          Revision: 'v1.0'
        BoardShape: [1×1 antenna.Rectangle]
```

```
    BoardThickness: 0.0062
            Layers: {[1×1 dielectric]  [1×1 antenna.Polygon]  [1×1 dielectric]  [1×1 dielectric]
     FeedLocations: [0 0 2]
      FeedDiameter: 1.0000e-03
      ViaLocations: []
       ViaDiameter: []
      FeedViaModel: 'square'
         Conductor: [1×1 metal]
              Tilt: 0
          TiltAxis: [0 0 1]
              Load: [1×1 lumpedElement]
```

View the PCB component in the Gerber file.

```
show(pcbcapacitor)
```



See Also

PCBReader | gerberRead | DielectricCatalog | dielectric

Introduced in R2021b

pcbComponent

Create single or multifeed PCB component

Description

Use the `pcbComponent` object to create a multiport PCB component consisting of metal and dielectric layers.

Creation

Syntax

```
pcb = pcbComponent
pcb = pcbComponent(Name=Value)
```

Description

`pcb = pcbComponent` creates a two-port PCB component.

`pcb = pcbComponent(Name=Value)` sets "Properties" on page 1-224 using one or more name-value arguments. For example, `pcb = pcbComponent(Name=PCBWilkinson)` creates a PCB component named `'PCBWilkinson'` Properties not specified retain their default values.

Properties

`Name` — Name of PCB component
`'MyPCB'` (default) | character vector | string scalar

Name of the PCB component, specified a character vector or string scalar.

Example: `component = pcbComponent(Name='PCBsplitter')`

Data Types: `char` | `string`

`Revision` — Revision details
`'v1.0'` (default) | character vector | string scalar

Design revision details of the PCB component, specified as a character vector or string scalar.

Example: `component = pcbComponent(Revision='v2.0')`

Data Types: `char` | `string`

`BoardShape` — Shape of PCB
`traceRectangular` (default) | object

Shape of the PC board, specified as a shape object. You can specify any one of the shapes from "Custom Geometry and PCB Fabrication".

Example: `trace = tracerectangular; component = pcbComponent(BoardShape=trace)` creates a rectangle shaped trace on a PCB.

Data Types: `char` | `string`

`BoardThickness` — Height of PCB component
`0.0016` (default) | positive scalar

Height of the PCB component, specified as a positive scalar in meters.

Example: `component = pcbComponent(BoardThickness=0.0026)`

Data Types: `double`

`Layers` — Metal and dielectric layers
`{[1×1 traceRectangular] [1×1 dielectric] [1×1 traceRectangular]}` (default) | cell array of metal and dielectric layers

Metal and dielectric layers, specified a cell array of metal and dielectric layers. You can specify one metal shape or one dielectric substrate per layer starting with the top layer and proceeding downward.

Data Types: `cell`

`FeedLocations` — Feed locations on PCB component
`[2×4 double]` (default) | *N*-by-3 array | *N*-by-4 array

Feed locations on the PCB component in Cartesian coordinates, specified as either an *N*-by-3 or *N*-by-4 array with *N* representing the number for ports on the PCB component. You can place the feed inside the board or at the edge of the board. The arrays translate into the following:

- *N*-by-3 – [*x*, *y*, *Layer*]
- *N*-by-4 – [*x*, *y*, *SigLayer*, *GndLayer*]

Example: `component = pcbComponent(FeedLocations=[-0.0187 0 1 2])`

Data Types: `double`

`FeedDiameter` — Diameter of center pin of feed connector
`0.0025` (default) | positive scalar

Diameter of center pin of the feed connector, specified as a positive scalar in meters.

Example: `component = pcbComponent(FeedDiameter=2.000e-04)`

Data Types: `double`

`ViaLocations` — Electrical short locations on PCB component
real vector of size *M*-by-4 array

Electrical short locations on the PCB component in Cartesian coordinates, specified as a real vector of size *M*-by-4 array. The array translates into the following:

- *M*-by-4 – [*x*, *y*, *SigLayer*, *GndLayer*]

Example: `component = pcbComponent(ViaLocations=[0 -0.025 1 2])`

Data Types: `double`

`ViaDiameter` — Diameter of electrical shorting pin used between metal layers
positive scalar

Diameter of electrical shorting pin used between metal layers, specified as a positive scalar in meters.

Example: `component = pcbComponent(ViaDiameter=1.0e-3)`

Data Types: `double`

`FeedViaModel` — Model for approximating feed and via
`'strip'` (default) | `'square'` | `'hexagon'` | `'octagon'`

Model for approximating the feed and via, specified as one of the following:

- `'strip'` – A rectangular strip approximation to the feed and via cylinder. This approximation is the simplest and results in a small mesh.
- `'square'` – A four-sided polyhedron approximation to the feed and via cylinder.
- `'hexagon'` – A six-sided polyhedron approximation to the feed and via cylinder.
- `'octagon'` – A eight-sided polyhedron approximation to the feed and via cylinder.

Example: `component = pcbComponent(FeedViaModel='octagon')`

Data Types: `char` | `string`

`Conductor` — Type of metal material
`'PEC'` (default) | `metal` object

Type of the metal used as a conductor, specified as a `metal` object. You can choose any metal from the `MetalCatalog` or specify a metal of your choice. For more information, see `metal`. For more information on metal conductor meshing, see "Method of Moments Solver for Metal and Dielectric Structures".

Example: `m = metal('Copper'); component = pcbComponent(Conductor=m)`

Load — Lumped elements
[1x1 LumpedElement] (default) | lumped element object handle

Lumped elements added to the PCB component feed, specified as a lumped element object handle. For more information, see `lumpedElement` Antenna Toolbox™.

Example: `Load = lumpedelement`. `lumpedelement` is the object handle for the load created using `lumpedElement`.

`Tilt` — Tilt angle of PCB component
`0` (default) | scalar | vector

Tilt angle of the PCB component, specified as a scalar or vector with each element unit in degrees.

Example: `Tilt=90`

Example: `pcb.Tilt = 90`

Example: `Tilt=[90 90]`,`TiltAxis=[0 1 0;0 1 1]` tilts the PCB component at 90 degrees about the two axes defined by the vectors.

Data Types: `double`

TiltAxis — Tilt axis of PCB component

[1 0 0] (default) | three-element vector of Cartesian coordinates | two three-element vectors of Cartesian coordinates | 'X' | 'Y' | 'Z'

Tilt axis of the PCB component, specified as:

- Three-element vector of Cartesian coordinates in meters. In this case, each coordinate in the vector starts at the origin and lies along the specified points on the X-, Y-, and Z-axes.
- Two points in space, each specified as three-element vectors of Cartesian coordinates. In this case, the PCB component rotates around the line joining the two points in space.
- A string input describing simple rotations around one of the principal axes, 'X', 'Y', or 'Z'.

Example: `TiltAxis=[0 1 0]`

Example: `TiltAxis=[0 0 0;0 1 0]`

Example: `pcb.TiltAxis = 'Z'`

Data Types: `double`

Object Functions

current	Calculate and plot current distribution
charge	Calculate and plot charge distribution
feedCurrent	Calculate current at feed port
gerberWrite	Generate Gerber files
layout	Plot all metal layers and board shape
mesh	Change and view mesh properties of metal or dielectric in PCB component
meshconfig	Change mesh mode of PCB component or shape structure
sparameters	Calculate S-parameters for RF PCB objects
show	Display PCB component structure or PCB shape

Examples

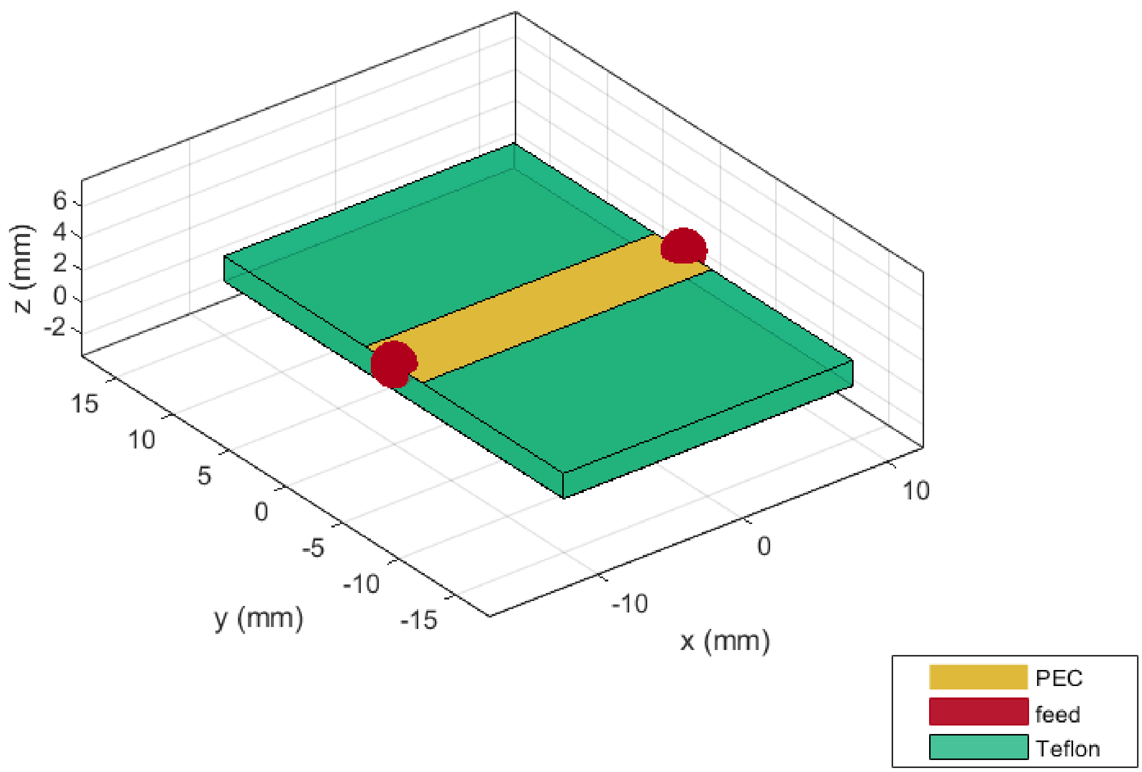
Create Default PCB Component and Plot S-Parameters

Create a PCB component using default properties.

```
pcb = pcbComponent;
```

View the PCB component.

```
show(pcb)
```

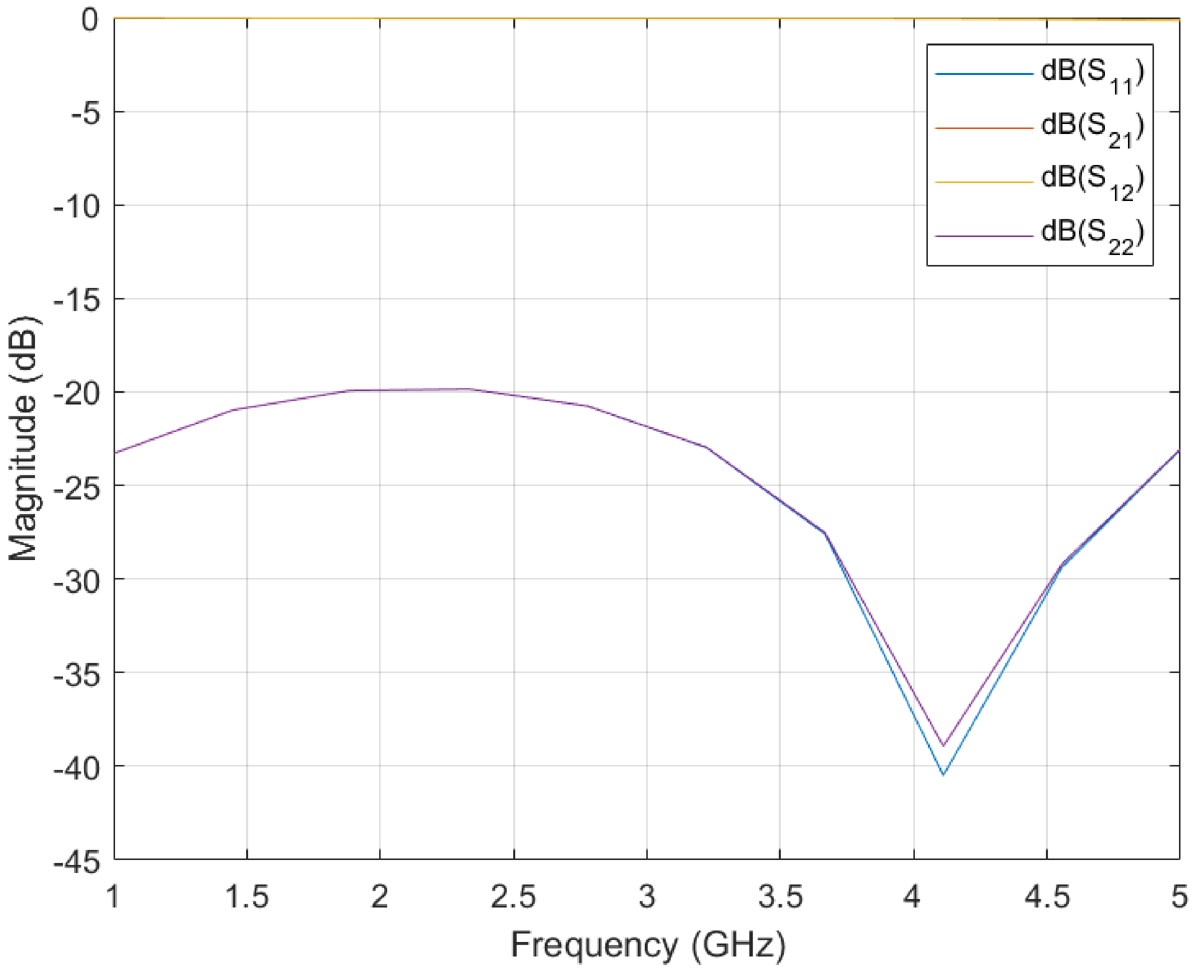


Calculate S-parameters over 10 frequencies from 1-5 GHz.

```
s=sparameters(pcb,linspace(1e9,5e9,10));
```

Plot the S-parameters.

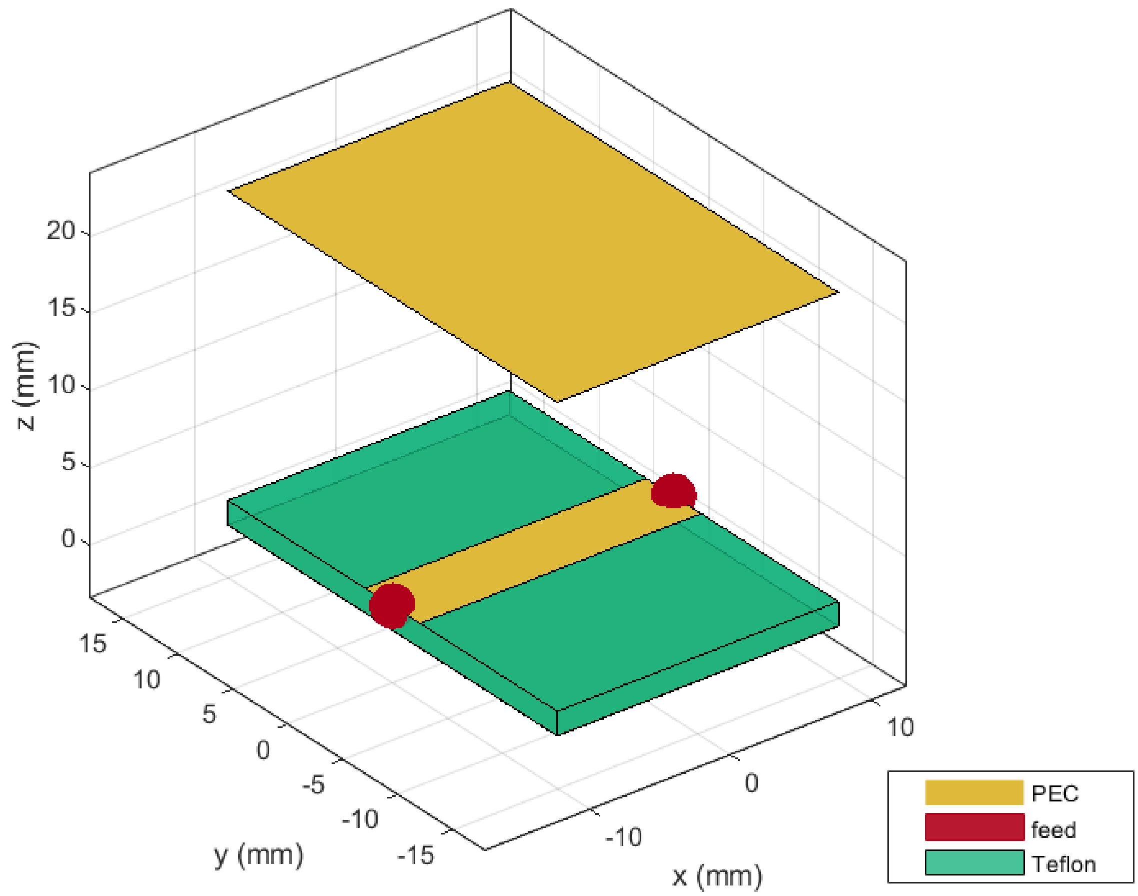
```
rfplot(s)
```



Create PCB Component with Lid on Top

Create a PCB component with a lid at a distance of 2 cm above the component.

```
p = pcbComponent;
pcblid = antenna.Rectangle(Length=p.Layers{1}.Length,Width=p.Layers{3}.Width);
dAir = dielectric('Air');
dAir.Thickness = 2e-2;
p.BoardThickness = p.BoardThickness + dAir.Thickness;
p.Layers = {pcblid,dAir,p.Layers{1},p.Layers{2},p.Layers{3}};
p.FeedLocations(:,3:4) = [3 5;3 5];
show(p)
```



Calculate the S-parameters over the 10 frequencies from 1-5 GHz.

```
s = sparameters(p,linspace(1e9,5e9,10));
```

See Also

gerberRead | gerberWrite | PCBReader | PCBWriter | PCBServices

Introduced in R2021b

pcbElement

Create RF Toolbox circuit element

Description

Use the `pcbElement` object to create an RF Toolbox circuit element.

Creation

Syntax

```
circuit_element = pcbElement(rfpcbobject)
circuit_element = pcbElement(rfpcbobject,Name=Value)
```

Description

`circuit_element = pcbElement(rfpcbobject)` creates a PCB element object from a PCB component. You can use this element in an RF Toolbox circuit.

`circuit_element = pcbElement(rfpcbobject,Name=Value)` sets properties using one or more name-value arguments.

Input Arguments

`rfpcbobject` — PCB component object
RF PCB object

PCB component object, specified as an RF PCB object. For a complete list of the PCB components, see "PCB Components Catalog".

Properties

`Behavioral` — Computes S-parameters using behavioral model
`'true'` (default) | `'false'`

Compute S-parameters of the PCB element using the behavioral model, specified as a logical true or false. When you specify true, the object calculates the S-parameters using the behavioral model. When you specify false, the object calculates the S-parameters using the full-wave solver. For components and shapes that support the behavioral model, see "Behavioral Models" and `sparameters`.

Data Types: `logical`

Examples

Calculate S-Parameters of Two Capacitors in Circuit

Create a circuit using default properties.

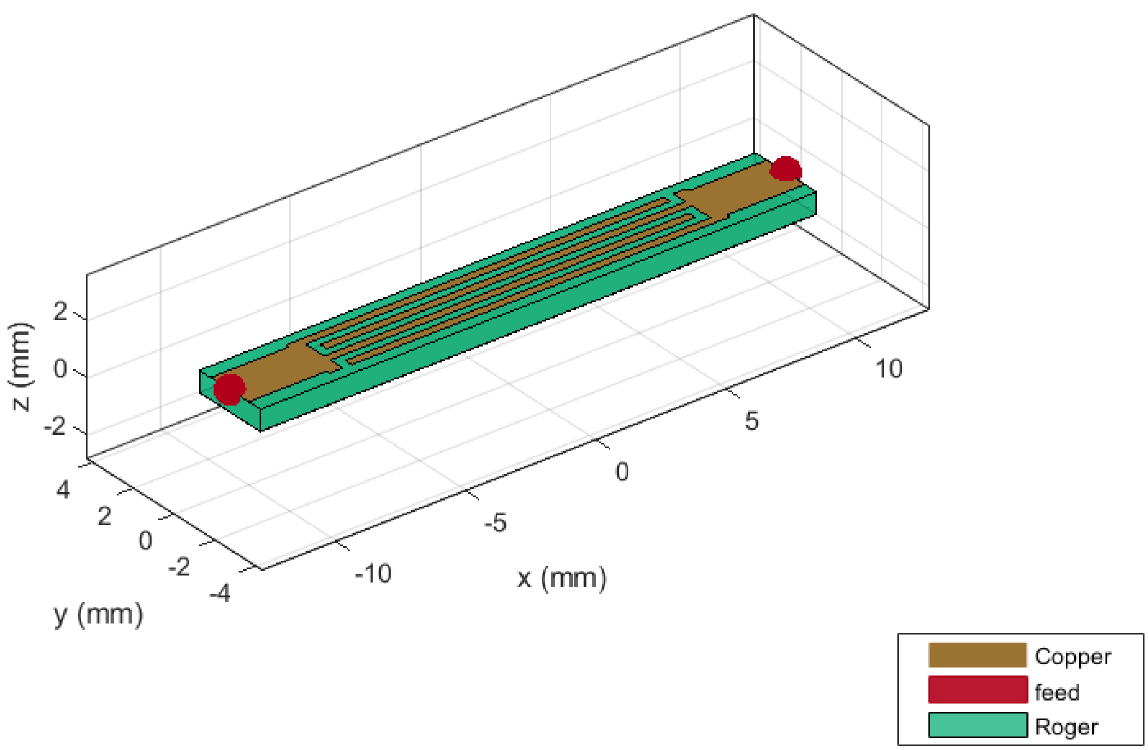
```
ckt = circuit;
```

Create two interdigital capacitors, one using default properties and one with three fingers.

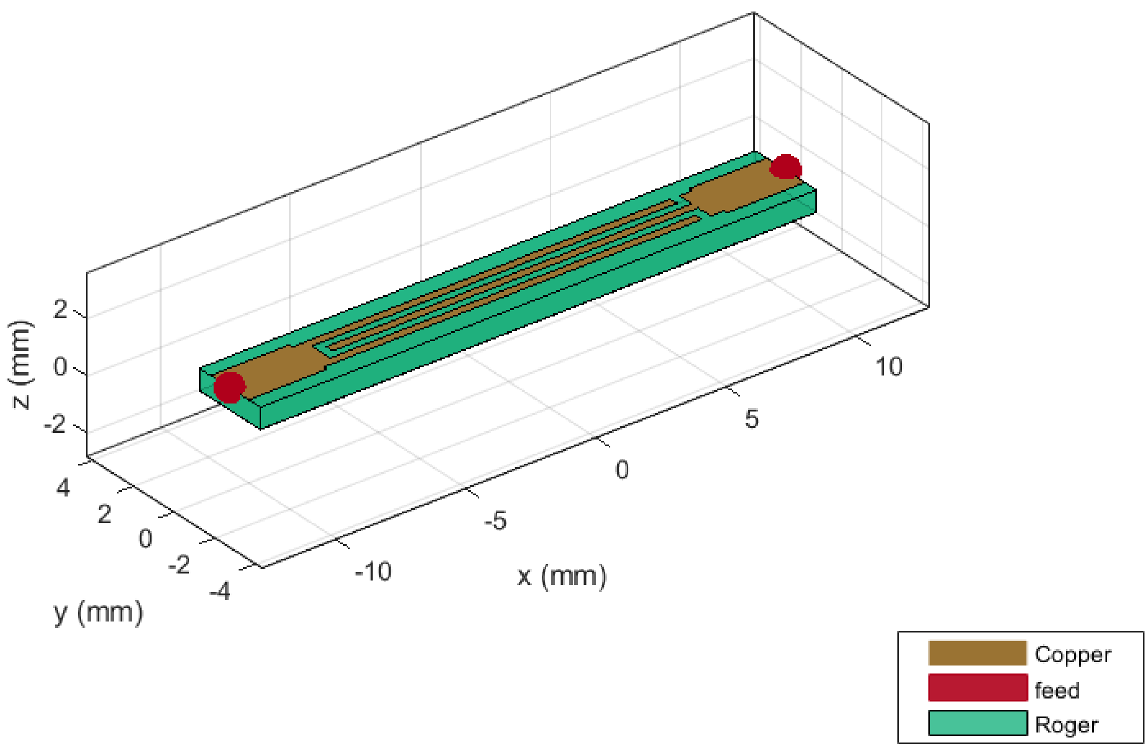
```
c1 = interdigitalCapacitor;
c2 = interdigitalCapacitor('NumFingers',3);
```

View both `c1` and `c2`.

```
show(c1)
```




```
figure;
show(c2)
```



Convert `c2` to a PCB element with the `Behavioral` property set to false.

```
p = pcbElement(c2,'Behavioral',false);
```

Add both capacitors to the circuit object.

```
add(ckt,[1 2 0 0],c1) % default pcbElement created automatically
add(ckt,[2 3 0 0],p)
setports(ckt,[1 0],[3 0])
```

Calculate the S-parameters.

```
S = sparameters(ckt,8e9)
```

```
S = 
  sparameters: S-parameters object

       NumPorts: 2
    Frequencies: 8.0000e+09
     Parameters: [2×2 double]
      Impedance: 50
```

```
rfparam(obj,i,j) returns S-parameter Sij
```

References

[1] Pozar, David M. *Microwave Engineering*. 4th ed. Hoboken, NJ: Wiley, 2012.

See Also

`sparameters`

Topics

"Behavioral Models"

Introduced in R2021b

2

Functions

show

Display PCB component structure or PCB shape

Syntax

```
show(pcbcomponent)
show(shape)
```

Description

`show(pcbcomponent)` displays the PCB component structure in the figure window.

`show(shape)` plots the shape as a filled region using patches.

Examples

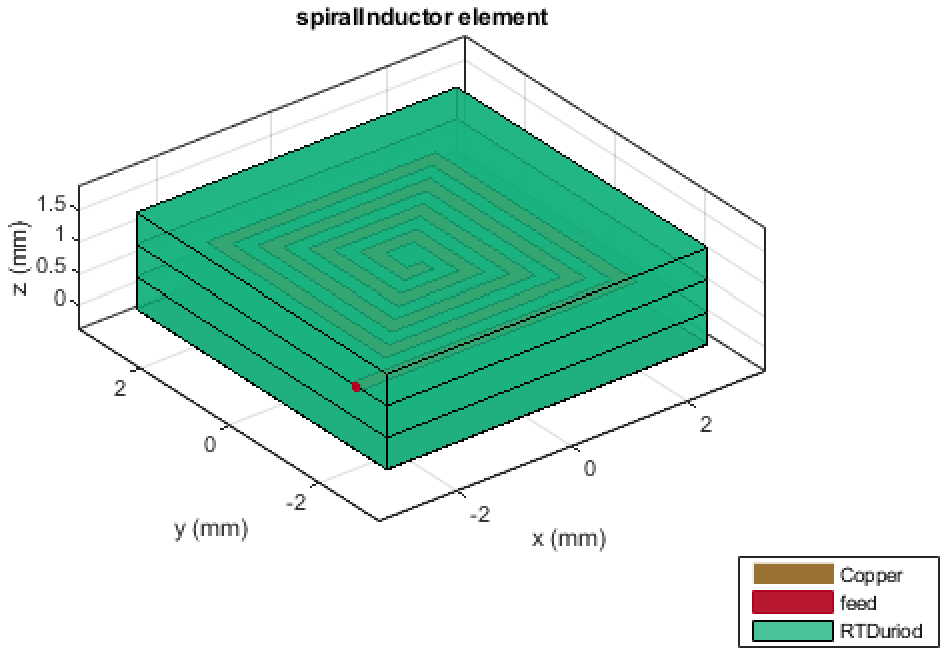
Create Default Spiral Inductor

Create and view a default spiral inductor.

```
inductor = spiralInductor
```

```
inductor = 
  spiralInductor with properties:

          SpiralShape: 'Square'
        InnerDiameter: 5.0000e-04
                Width: 2.5000e-04
              Spacing: 2.5000e-04
             NumTurns: 4
               Height: 0.0010
    GroundPlaneLength: 0.0056
     GroundPlaneWidth: 0.0056
            Substrate: [1x1 dielectric]
            Conductor: [1x1 metal]
```

```
show(inductor)
```



Create Default Curved U-Bend

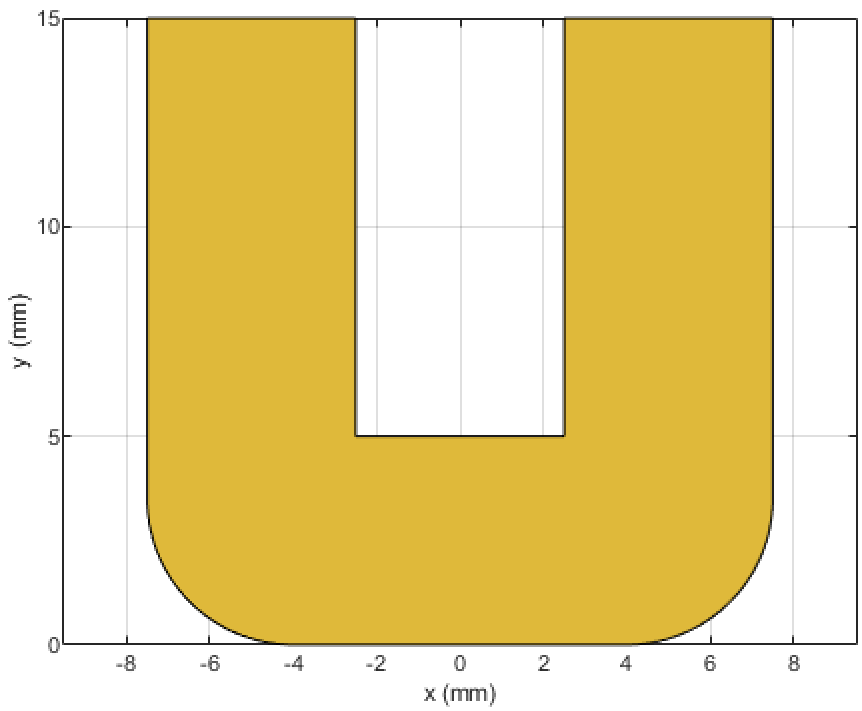
Create a curved U-bend with default properties.

```
curvedubend = ubendCurved
```

```
curvedubend = 
  ubendCurved with properties:

             Name: 'myCurvedubend'
    ReferencePoint: [0 0]
            Length: [0.0150 0.0050 0.0150]
             Width: [0.0050 0.0050 0.0050]
       CurveRadius: 0.0035
```

View the shape.

```
show(curvedubend)
```



Input Arguments

`pcbcomponent` — PCB component object

object handle

PCB component object, specified as a object handle.

Example: `microstrip = microstripLine; show(microstrip)`

`shape` — Shape created using custom elements and shape objects

object handle

Shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object handle.

Example: `shape = bendCurved; show(shape)`

See Also

Introduced in R2021b

current

Calculate and plot current distribution

Syntax

```
current(rfpcbobject,frequency)

i = current(rfpcbobject,frequency)
[i,p] = current(rfpcbobject,frequency)

current(rfpcbobject,frequency,'dielectric')
i = current(rfpcbobject,frequency,'dielectric')
i = current( ___ ,Name=Value)
```

Description

`current(rfpcbobject,frequency)` calculates and plots the absolute value of the current on the metal surface of a PCB component at the specified frequency.

`i = current(rfpcbobject,frequency)` calculates the *x*, *y*, *z* components of the current on the surface of a PCB component at a specified frequencies.

`[i,p] = current(rfpcbobject,frequency)` returns the current distribution and the points at which the current calculation was performed.

`current(rfpcbobject,frequency,'dielectric')` calculates and plots the absolute value of the current at the specified frequency on the dielectric surface of the PCB component.

`i = current(rfpcbobject,frequency,'dielectric')` calculates the *x*, *y*, *z* components of the current on the dielectric surface of a PCB component at the specified frequency.

`i = current( ___ ,Name=Value)` calculates the current on the surface of a PCB component using additional name-value arguments.

Examples

Calculate Current Distribution on Rat-Race Coupler

Create a rat-race coupler with default properties.

```
coupler = couplerRatrace;
```

Set the excitation voltage and the phase angle at the ports of the coupler.

```
v = voltagePort(4)
```

```
v = 
  voltagePort with properties:

        NumPorts: 4
```

```
      FeedVoltage: [1 0 0 0]
        FeedPhase: [0 0 0 0]
    PortImpedance: 50
```

```
v.FeedVoltage = [1 0 1 0]
```

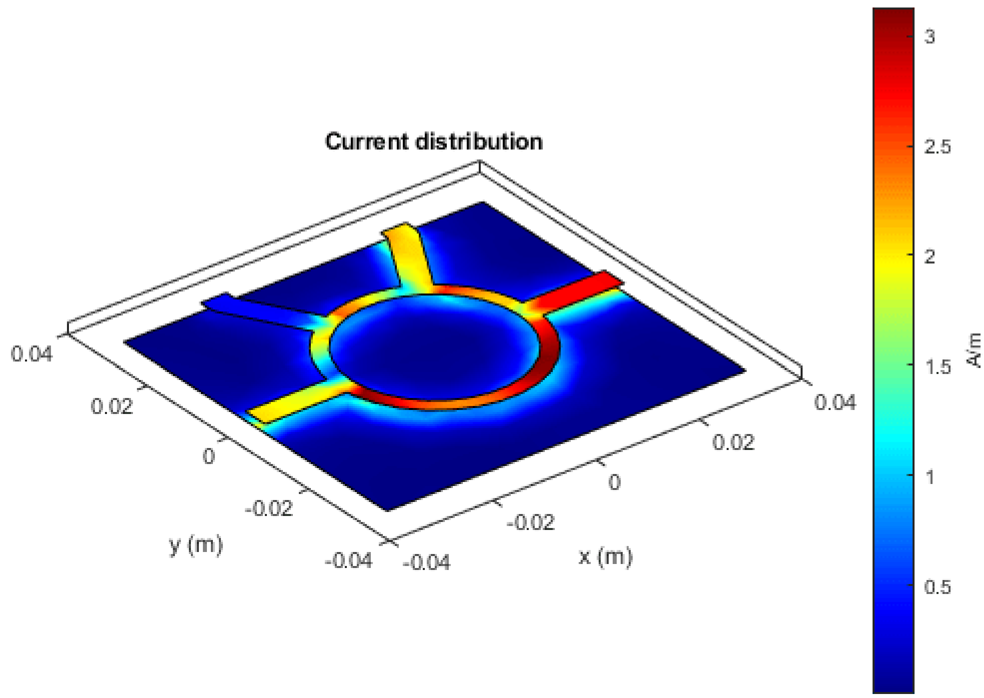
```
v = 
  voltagePort with properties:

         NumPorts: 4
      FeedVoltage: [1 0 1 0]
        FeedPhase: [0 0 0 0]
    PortImpedance: 50
```

```
v.FeedPhase = [90 0 270 0]
```

```
v = 
  voltagePort with properties:

         NumPorts: 4
      FeedVoltage: [1 0 1 0]
        FeedPhase: [90 0 270 0]
    PortImpedance: 50
```

Calculate and plot the current on the coupler at 3 GHz.

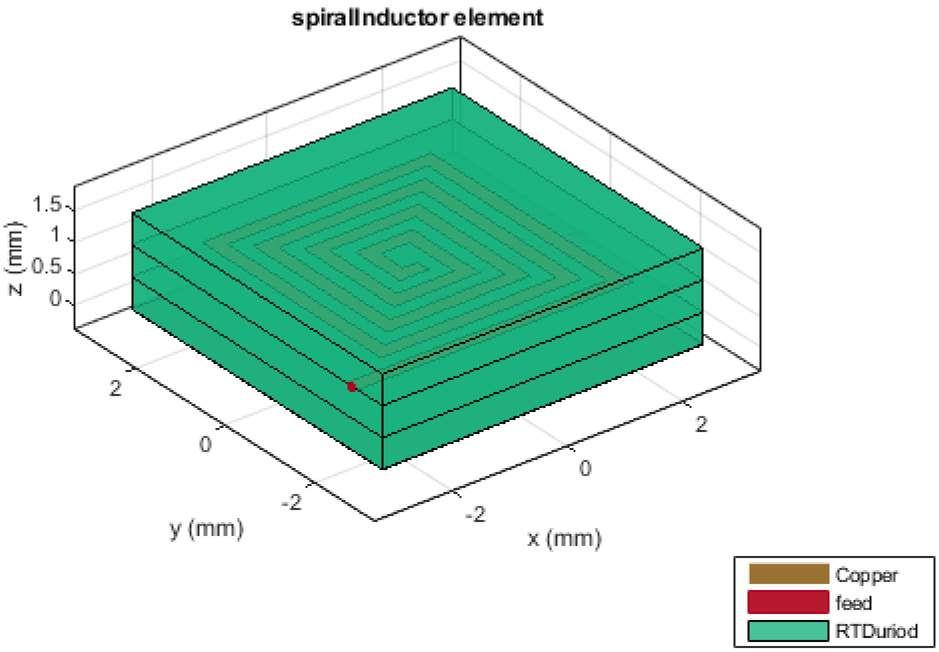
```
figure
current(coupler,3e9,Excitation=v)
```



Calculate Current Distribution on Spiral Inductor

Create a default spiral inductor.

```
inductor = spiralInductor;
show(inductor)
```



Calculate the current distribution on the inductor at 600 MHz.

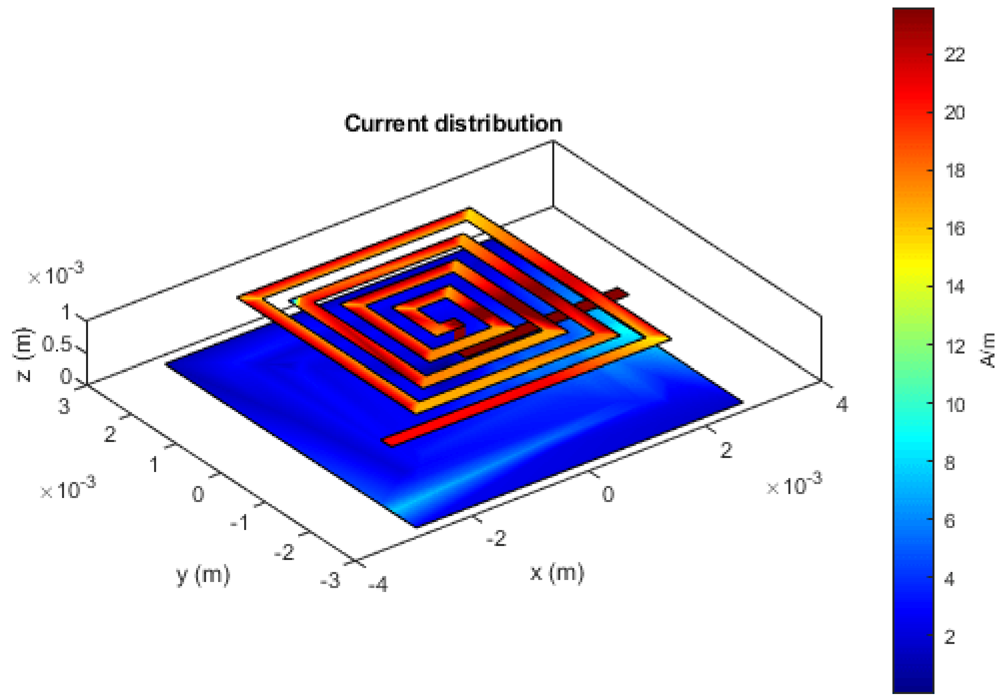
```
[i,p] = current(inductor,600e6)
```

```
i = 3×258 complex

  -0.0728 + 0.1411i  11.9623 -20.0648i  -0.0003 + 0.0033i   0.1377 - 0.2742i  -0.8881 + 1.3157i
   0.7718 - 1.2051i   3.0732 - 4.7207i   0.8737 - 1.2552i   0.4853 - 0.4545i  -0.0000 + 0.0016i
   0.0000 + 0.0000i   0.0000 + 0.0000i   0.0000 + 0.0000i   0.0000 + 0.0000i   0.0000 + 0.0000i
```

```
p = 3×258

    0.0024    0.0025    0.0025    0.0023   -0.0001   -0.0022   -0.0009   -0.0019    0.0021
   -0.0022   -0.0002    0.0017    0.0007    0.0027    0.0025   -0.0022   -0.0021   -0.0020   -0.0
         0         0         0         0         0         0         0         0         0
```

Plot the current distribution.

```
current(inductor,600e6)
```



Input Arguments

`rfpcbobject` — PCB component object
RF PCB object

PCB component object, specified as an RF PCB object. For a complete list of the PCB components, microstrip bends, and traces, see “PCB Components Catalog” and “Custom Geometry and PCB Fabrication”.

`frequency` — Frequency used to calculate current distribution
scalar

Frequency to calculate the current distribution, specified as a scalar in Hz.

Example: 70e6

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `scale=log10`

`scale` — Scale to visualize current distribution
linear (default) | string scalar | character vector

Scale to visualize the current distribution on the surface of the PCB component, specified as a string or a character vector. The string values are either `'linear'`, `'log'`, or `'log10'` or as a function. You can specify any mathematical function such as `log`, `log10`, `cos`, or `sin`.

Data Types: `char` | `function_handle`

`excitation` — Excitation using voltage source
function handle

Excitation using as voltage source, specified function handle from the `voltagePort` function.

Data Types: `char` | `function_handle`

Output Arguments

`i` — *x*, *y*, *z* components of current distribution
3-by-*n* complex matrix in A/m

x, *y*, *z* components of the current distribution, returned as a 3-by-*n* complex matrix in A/m. The value of the current is calculated on every triangle mesh on the surface of the PCB component.

`p` — Cartesian coordinates representing center of each triangle in mesh
3-by-*n* real matrix

Cartesian coordinates representing the center of each triangle in the mesh, returned as a 3-by-*n* real matrix.

See Also
`voltagePort` | `charge`

Introduced in R2021b

charge

Calculate and plot charge distribution

Syntax

```
charge(rfpcbobject,frequency)
```

```
c = charge(rfpcbobject,frequency)
[c,p] = charge(rfpcbobject,frequency)
```

```
charge(rfpcbobject,frequency,'dielectric')
c = charge(rfpcbobject,frequency,'dielectric')
c = charge( ___ ,Name=Value)
```

Description

`charge(rfpcbobject,frequency)` calculates and plots the absolute value of the charge in C/m on the metal surface of a PCB component at the specified frequencies.

`c = charge(rfpcbobject,frequency)` calculates a vector of charges in C/m on the metal surface of a PCB component, at the specified frequencies.

`[c,p] = charge(rfpcbobject,frequency)` returns the point at which the charge calculation was performed.

`charge(rfpcbobject,frequency,'dielectric')` calculates and plots the absolute value of the charge at the specified frequency on the dielectric surface of the PCB component.

`c = charge(rfpcbobject,frequency,'dielectric')` calculates the charge on the dielectric surface of a PCB component at the specified frequency.

`c = charge( ___ ,Name=Value)` calculates the charge on the surface of a PCB component using additional name-value arguments.

Examples

Calculate Charge Distribution on Rat-Race Coupler

Create a rat-race coupler with default properties.

```
coupler = couplerRatrace;
```

Set the feed voltage and phase at the coupler ports.

```
v = voltagePort(4)
```

```
v = 
  voltagePort with properties:

       NumPorts: 4
```

```
    FeedVoltage: [1 0 0 0]
      FeedPhase: [0 0 0 0]
```

```
v.FeedVoltage = [1 0 1 0]
```

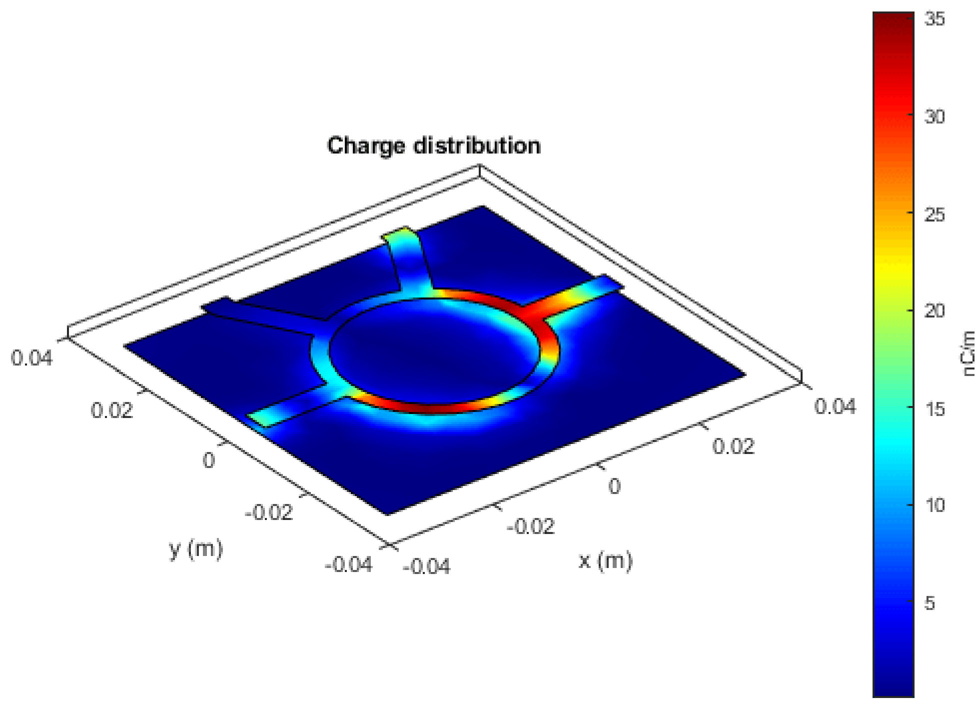
```
v = 
  voltagePort with properties:

       NumPorts: 4
    FeedVoltage: [1 0 1 0]
      FeedPhase: [0 0 0 0]
```

```
v.FeedPhase = [90 0 270 0]
```

```
v = 
  voltagePort with properties:

       NumPorts: 4
    FeedVoltage: [1 0 1 0]
      FeedPhase: [90 0 270 0]
```

Calculate and view the charge distribution of the coupler at 3 GHz.

```
figure
charge(coupler,3e9,Excitation=v)
```



Input Arguments

`rfpcbobject` — PCB component object
RF PCB object

PCB component object, specified as an RF PCB object. For a complete list of the PCB components and shapes, see "PCB Components Catalog" and "Custom Geometry and PCB Fabrication".

`frequency` — Frequency to calculate charge distribution
scalar

Frequency to calculate charge distribution in hertz, specified as a scalar.

Example: 70e6

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `scale='log10'`

`scale` — Scale to visualize charge distribution
string (default) | character vector

Scale to visualize the charge distribution on the surface of the PCB component, specified as a string or a character vector. The string values are either `'linear'`, `'log'`, or `'log10'` or as a function. You can specify any mathematical function such as `log`, `log10`, `cos`, or `sin`.

Data Types: `char` | `function_handle`

`excitation` — Excitation using voltage source
function handle

Excitation using as voltage source of N-ports to excite an N-port RF PCB component, specified a string or a function handle.

Data Types: `string` | `function_handle`

Output Arguments

c — Complex charges
1-by-*n* vector in C/m

Complex charges, returned as a 1-by-*n* vector in C/m. This value is calculated on every triangle mesh or every dielectric tetrahedron face on the surface of a PCB component.

p — Cartesian coordinates representing center of each triangle in mesh
3-by-*n* real matrix

Cartesian coordinates representing the center of each triangle in the mesh, returned as a 3-by-*n* real matrix.

See Also

`current`

Introduced in R2021b

feedCurrent

Calculate current at feed port

Syntax

```
feedCurrent(rfpcbobject,frequency)
feedCurrent( ___ ,Name=Value)
```

Description

`feedCurrent(rfpcbobject,frequency)` calculates the current in A/m at the feed port of a PCB component at the specified frequency. The feed current when multiplied by the PCB impedance gives the voltage across the PCB component.

`feedCurrent( ___ ,Name=Value)` calculates the feed current at the feed port of the PCB component using additional name-value arguments.

Examples

Calculate Feed Current of Rat-Race Coupler

Create a rat-race coupler with default properties.

```
coupler = couplerRatrace;
```

Set the feed voltage and phase at the ports of the rat-race coupler:

```
v = voltagePort(4);
v.FeedVoltage = [1 0 1 0];
v.FeedPhase = [90 0 270,0];
```

Calculate the feed current at 3 GHz.

```
If = feedCurrent(coupler,3e9)
```

```
If = 1×4 complex

   0.0003 - 0.0151i   0.0001 - 0.0178i  -0.0001 + 0.0045i  -0.0002 + 0.0199i
```

Input Arguments

`rfpcbobject` — PCB component object
RF PCB object

PCB component object, specified as an RF PCB object. For a complete list of the PCB components and shapes, see “PCB Components Catalog” and “Custom Geometry and PCB Fabrication”.

`frequency` — Frequency to calculate feed current
scalar | vector

Frequency to calculate the feed current, specified as a scalar integer in Hz or as a vector with each element specified in Hz.

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `excitation='2'`

`excitation` — Excitation using voltage source
`string` (default) | function handle

Excitation using as voltage source, specified function handle from the `voltagePort` function.

Data Types: `char` | `function_handle`

See Also

`current` | `voltagePort`

Introduced in R2021b

mesh

Change and view mesh properties of metal or dielectric in PCB component

Syntax

```
mesh(object)
mesh(shape)
mesh( ___ ,Name,Value)
meshdata = mesh( ___ ,Name,Value)
```

Description

`mesh(object)` plots the mesh used to analyze a PCB component.

`mesh(shape)` plots the mesh for the shapes.

`mesh( ___ ,Name,Value)` changes and plots the mesh structure of a PCB component, using additional options specified by the name-value pairs. You can also determine the number of unknowns from the number of basis functions in the output.

`meshdata = mesh( ___ ,Name,Value)` returns a mesh structure that specifies the properties used to analyze the PCB component.

Examples

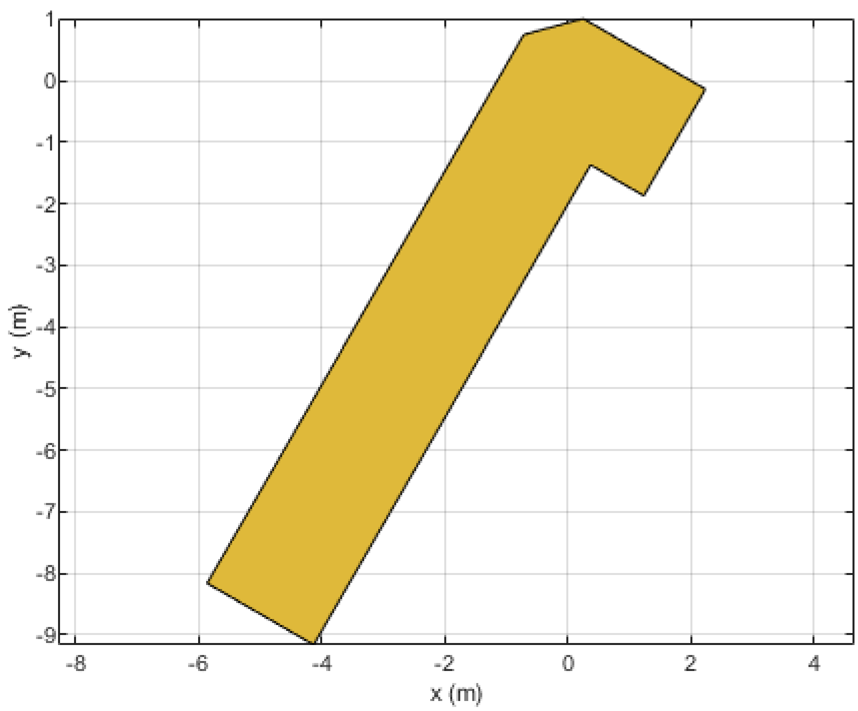
Mesh Rotated Mitered Bend Shape

Create a mitered bend shape of lengths of 10 m and 2 m, width of 2 m, and rotate it about the Z-axis by 60 degrees.

```
bend = bendMitered(Length=[10 2],Width=[2 2],MiterDiagonal=1);
bend = rotateZ(bend,60)
```

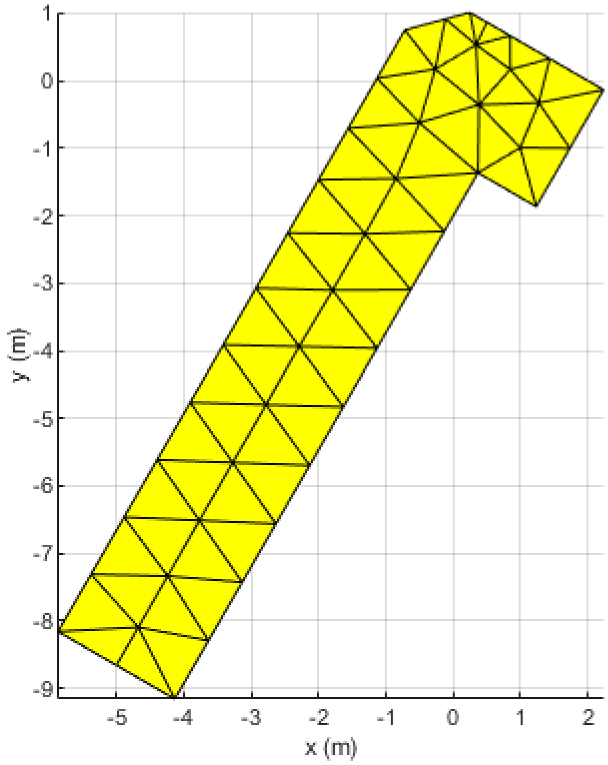
```
bend = 
  bendMitered with properties:

              Name: 'myMiteredbend'
    ReferencePoint: [0 0]
            Length: [10 2]
             Width: [2 2]
     MiterDiagonal: 1
```

```
show(bend)
```



Mesh the mitered bend shape at a maximum edge length of 1 m.

```
mesh(bend,MaxEdgeLength=1)
```



Input Arguments

`object` — PCB component
object handle

PCB component, specified as an object handle. For complete list of PCB components and shapes, see "PCB Components Catalog"

`shape` — Shape created using custom elements and shape objects
object handle

Shape created using custom elements and shape objects, specified as an object handle. For a complete list of custom shapes, see "Custom Geometry and PCB Fabrication".

Example: `c = bendCurved; mesh(c)`

Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` pair arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside single quotes (`''`). You can specify several name and value pair arguments in any order as `Name1, Value1, ..., NameN, ValueN`.

Example: 'MaxEdgeLength', 0.1

`MaxEdgeLength` — Maximum edge length of triangles in mesh
scalar

Maximum edge length of triangles in mesh, specified as a comma-separated pair consisting of 'MaxEdgeLength' and a scalar. All triangles in the mesh have sides less than or equal to the 'MaxEdgeLength' value.

Data Types: double

MinEdgeLength — Minimum edge length of triangles in mesh
scalar

Minimum edge length of triangles in mesh, specified as a comma-separated pair consisting of 'MinEdgeLength' and a scalar. All triangles in the mesh have sides greater than or equal to the 'MinEdgeLength'.

Note You can use this property only with the pcbComponent object.

Data Types: double

GrowthRate — Mesh growth rate
0.7 (default) | scalar

Mesh growth rate, specified as a comma-separated pair consisting of 'GrowthRate' and a scalar. The default value of 0.7 states that the growth rate of the mesh is 70 percent. Growth rate values lie between 0 and 1.

Note You can use this property only with the pcbComponent object.

Data Types: double

View — Customize mesh view of a PCB component
'all' (default) | 'metal' | 'dielectric surface' | 'dielectric volume'

Customize mesh view of a PCB component, specified as a comma-separated pair consisting of 'View' and 'all', 'metal','dielectric surface', or 'dielectric volume'.

You choose 'dielectric surface' to view the boundary triangle mesh of the dielectric. You choose 'dielectric volume' to view the tetrahedral volume mesh of the dielectric.

Data Types: char

See Also
meshconfig

Introduced in R2021b

getZ0

Calculate characteristic impedance of transmission line

Syntax

```
z0 = getZ0(txline)
z0 = getZ0(txline,freq)
```

Description

`z0 = getZ0(txline)` calculates the characteristic impedance `z0` of a transmission line.

`z0 = getZ0(txline,freq)` calculates the characteristic impedance at the specified frequency.

Examples

Calculate Characteristic Impedance of Microstrip Transmission Line

Create a microstrip transmission line with default properties.

```
mline = microstripLine
```

```
mline = 
  microstripLine with properties:

             Length: 0.0200
              Width: 0.0050
             Height: 0.0016
    GroundPlaneWidth: 0.0300
          Substrate: [1x1 dielectric]
          Conductor: [1x1 metal]
```

Calculate the characteristic impedance of the line.

```
Z0 = getZ0(mline)
```

```
Z0 = 49.6897 - 0.0003i
```

Input Arguments

`txline` — Transmission line
transmission line object (default)

Transmission line, specified as one of the following: `coplanarWaveguide`, `coupledMicrostripLine`, and `microstripLine`.

Example: `txline = microstripLine;getZ0(txline)`. Calculates the characteristic impedance of the microstrip transmission line object with handle `txline`.

Data Types: `char` | `string`

`freq` — Frequency
positive scalar

Frequency to calculate the characteristic impedance, specified as a positive scalar in hertz.

Output Arguments

`z0` — Characteristic impedance of transmission lines
complex scalar

Characteristic impedance of the transmission line, returned as a complex scalar.

Data Types: `double`

See Also

`sparameters`

Introduced in R2021b

inductance

Calculate inductance

Syntax

```
inductance(object,frequency)
l = inductance(object,frequency)
```

Description

`inductance(object,frequency)` calculates and plots the inductance of an inductor over the specified frequency.

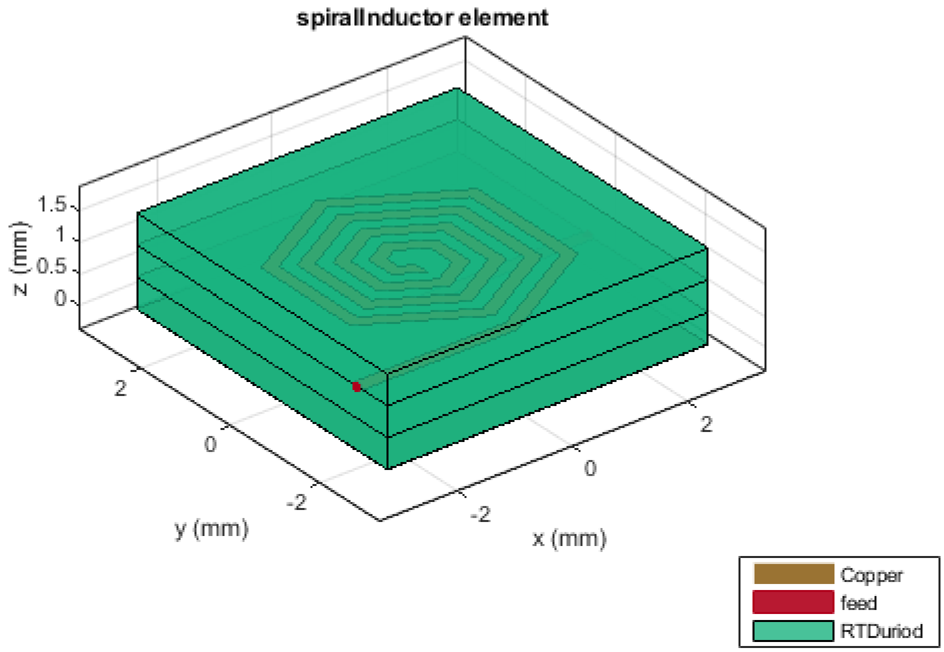
`l = inductance(object,frequency)` calculates the inductance of an inductor over the specified frequency.

Examples

Calculate Inductance of Spiral Inductor

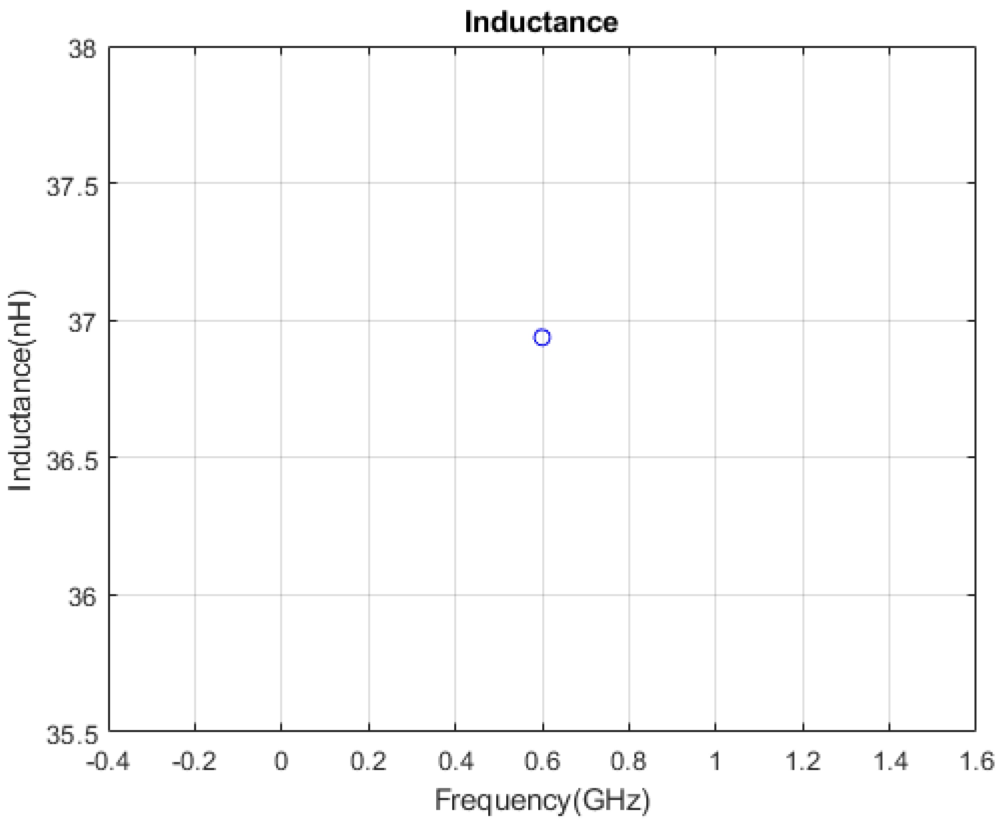
Create a hexagonal spiral inductor with default properties.

```
inductor = spiralInductor(SpiralShape='Hexagon');
show(inductor)
```



Calculate the inductance of the inductor at 600 MHz.

```
inductance(inductor,600e6)
```




```
l = inductance(inductor,600e6)
```

```
l = 3.6937e-08
```

Input Arguments

`object` — Spiral inductor
`spiralInductor` object

Spiral inductor, specified as an `spiralInductor` object.

Data Types: `char` | `string`

`frequency` — Frequency to calculate inductance
nonnegative scalar | nonnegative vector

Frequency to calculate inductance, specified as a nonnegative scalar or vector in hertz.

Data Types: `double`

Output Arguments

`l` — Inductance of inductor
scalar | vector

Inductance of the inductor, returned as a scalar or vector in henries.

Data Types: `double`

See Also
`capacitance`

Introduced in R2021b

capacitance

Calculate capacitance

Syntax

```
capacitance(object,frequency)
c = capacitance(object,frequency)
capacitance( ___ ,Name=Value)
```

Description

`capacitance(object,frequency)` calculates and plots the capacitance of a capacitor over the specified frequency.

`c = capacitance(object,frequency)` calculates the capacitance of a capacitor over the specified frequency.

`capacitance( ___ ,Name=Value)` calculates the capacitance with additional options specified using name-value arguments.

Examples

Calculate Capacitance of Interdigital Capacitor

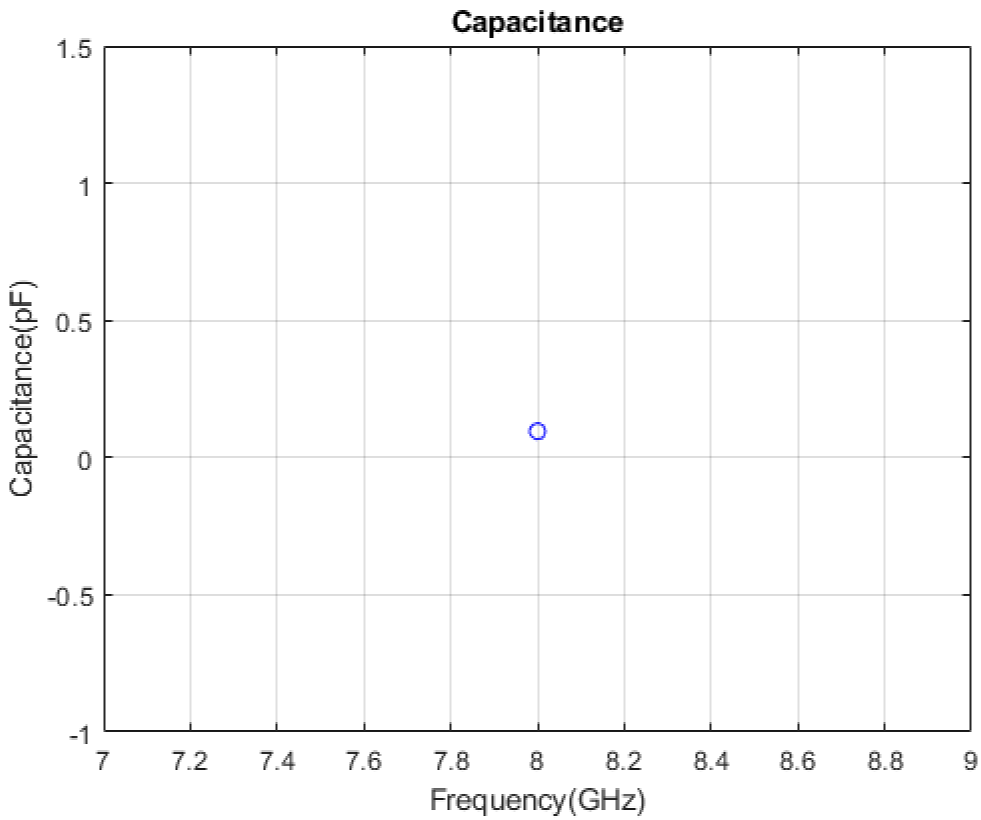
Create an interdigital capacitor using default properties.

```
capacitor = interdigitalCapacitor
```

```
capacitor = 
  interdigitalCapacitor with properties:

           NumFingers: 4
         FingerLength: 0.0137
          FingerWidth: 3.1600e-04
        FingerSpacing: 3.0000e-04
        FingerEdgeGap: 3.4100e-04
    TerminalStripWidth: 5.0000e-04
         PortLineWidth: 0.0019
        PortLineLength: 0.0030
               Height: 7.8700e-04
      GroundPlaneWidth: 0.0030
            Substrate: [1×1 dielectric]
            Conductor: [1×1 metal]
```

Calculate the capacitance of the capacitor at 8 GHz.

```
capacitance(capacitor,8e9,DeEmbed=1,IncludeParasitics=1)
```



Input Arguments

`object` — Interdigital capacitor
`interdigitalCapacitor` object

Interdigital capacitor, specified as an `interdigitalCapacitor` object.

Data Types: `char` | `string`

`frequency` — Frequency to calculate capacitance
nonnegative scalar | vector

Frequency to calculate the capacitance in hertz, specified as a nonnegative scalar or vector of nonnegative elements.

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `DeEmbed=1`

`DeEmbed` — Deembed feeder line
`1` (default) | `0`

Dembed feeder line, specified as `0` or `1`. When you specify `1`, the function deembeds the feeder line.

`IncludeParasitics` — Include parasitic effects
`1` (default) | `0`

Include parasitic effects, specified as `0` or `1`. When you specify `1`, the function includes the parasitic effects.

`Z0` — Input and output line impedance
`50` (default) | positive scalar

Input and output line impedance to feed the capacitor, specified as a positive scalar in ohms.

Output Arguments

`c` — Capacitance of capacitor
scalar | vector

Capacitance of the capacitor, returned as a scalar or vector in farads.

Data Types: `double`

See Also

`inductance`

Introduced in R2021b

qualityfactor

Calculates and plots Q-factor of capacitor

Syntax

```
qualityfactor(objectfrequency)
qf = qualityfactor(objectfrequency)
qualityfactor(object,frequency,Name,Value)
qf = qualityfactor(object,frequency,Name,Value)
```

Description

`qualityfactor(objectfrequency)` calculates and plots the Q-factor (quality factor) of the capacitor over the specified frequency values in the figure window.

`qf = qualityfactor(objectfrequency)` returns the Q-factor of the capacitor over the specified frequency values.

`qualityfactor(object,frequency,Name,Value)` sets properties using one or more name-value pairs.

`qf = qualityfactor(object,frequency,Name,Value)` sets properties using one or more name-value pairs.

Examples

Input Arguments

`object` — PCB capacitor component
object handle

PCB capacitor component, specified as an object handle.

Example:

Data Types: `char` | `string`

`frequency` — Frequency to calculate inductance in hertz
nonnegative scalar or vector

Frequency to calculate inductance in hertz, specified as a nonnegative scalar or vector.

Example:

Data Types: `double`

Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside quotes. You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

Example: `'DeEmbed',1`

`DeEmbed` — Deembed feeder line
`1` (default) | `0`

Dembed feeder line when set to 1, specified as `0` or `1`.

`Z0` — Input and output line impedance in ohms
`50` (default) | positive scalar

Input and output line impedance to feed the capacitor in ohms, specified as a positive scalar.

Output Arguments

`qf` — Quality factor of capacitor
scalar or vector

Quality factor of the capacitor, returned as a scalar or vector.

Data Types: `double`

See Also

Introduced in R2021b

qualityfactor

Calculates and plots Q-factor of inductor

Syntax

```
qualityfactor(objectfrequency)
qf = qualityfactor(objectfrequency)
```

Description

`qualityfactor(objectfrequency)` calculates and plots the Q-factor (quality factor) of the inductor over the specified frequency values in the figure window.

`qf = qualityfactor(objectfrequency)` returns the Q-factor of the inductor over the specified frequency values.

Examples

Input Arguments

`object` — PCB inductor component
object handle

PCB inductor component, specified as an object handle.

Example:

Data Types: `char` | `string`

`frequency` — Frequency to calculate inductance in hertz
nonnegative scalar or vector

Frequency to calculate inductance in hertz, specified as a nonnegative scalar or vector.

Example:

Data Types: `double`

Output Arguments

`qf` — Quality factor of inductor
scalar or vector

Quality factor of the inductor, returned as a scalar or vector.

Data Types: `double`

See Also

Introduced in R2021b

metal

Conductor material

Syntax

```
m = metal(material)
m = metal(Name=Value)
```

Description

`m = metal(material)` returns the metal used as a conductor in the PCB components. You can specify a material from the `MetalCatalog`. The default value for material is perfect electric conductor (PEC).

`m = metal(Name=Value)` returns the metal based on the properties specified by one or more .

Examples

Microstrip Line with Copper Conductor

Create a microstrip transmission line with copper conductor.

```
mline = microstripLine;
```

Create a copper metal conductor.

```
m = metal ('copper')
```

```
m = 
  metal with properties:

            Name: 'Copper'
    Conductivity: 59600000
       Thickness: 3.5560e-05

For more materials see catalog
```

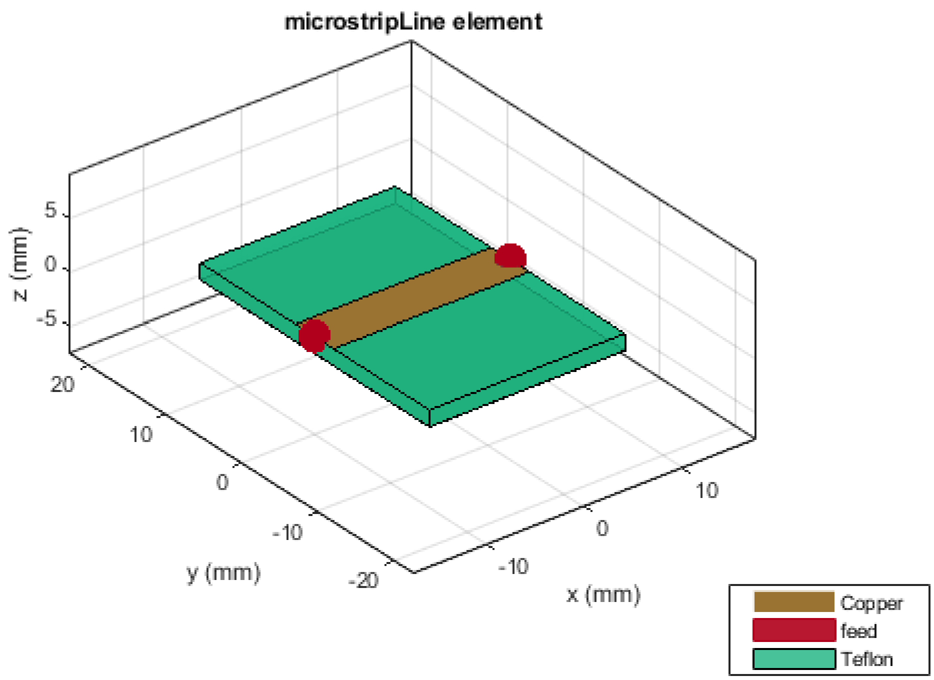
Change the microstrip transmission line conductor to copper.

```
mline.Conductor = m
```

```
mline = 
  microstripLine with properties:

             Length: 0.0200
              Width: 0.0050
             Height: 0.0016
    GroundPlaneWidth: 0.0300
           Substrate: [1x1 dielectric]
           Conductor: [1x1 metal]
```

View the microstrip transmission line.

```
show(mline)
```



Input Arguments

`material` — Material from metal catalog
'PEC' (default) | character vector

Material from the dielectric catalog, specified as a metal from the `MetalCatalog`. The default material is perfect electric conductor (PEC), which has infinite conductivity and zero thickness.

Example: 'Iron'

Data Types: `char`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: Name='Iron'

`Name` — Name of metal material
character vector

Name of the metal material you want to use as a conductor, specified as a character vector.

Example: `Name='Iron'`

Data Types: `char`

`Conductivity` — Conductivity of metal material
`Inf` (default) | scalar

Conductivity of the metal material, specified as a scalar in Siemens per meters(S/m). If you set `'Conductivity'` to `'Inf'`, you must set `'Thickness'` to `'0'`.

Example: `Conductivity=4.8e06`

Data Types: `double`

`Thickness` — Thickness of metal
`0` (default) | scalar

Thickness of the metal material along the default *z*-axis, specified as a scalar in meters.

Example: `Thickness=0.26e-6`

Data Types: `double`

Output Arguments

`m` — Conductor metal
metal object

Conductor metal, returned as a metal object.

See Also

`MetalCatalog`

Introduced in R2021a

MetalCatalog

Catalog of metals

Syntax

`mc = MetalCatalog`

Description

`mc = MetalCatalog` creates an object handle for the metal catalog.

- To open the metal catalog, use `open(mc)`
- To see the properties of a metal from the metal catalog, use `s = find(mc,name)`.

Examples

Use Metal Catalog to Design Coplanar Waveguide

Open the metal catalog.

```
mc = MetalCatalog;
open(mc)
```

	Name	Conductivity	Thickness	Units	Comments
1	PEC	Inf	0	m	
2	Copper	59.6000e+006	1.4000	mil	1 oz
3	Aluminium	37.7000e+006	30	mil	
4	Gold	41.1000e+006	0.2000	um	
5	Silver	63.0000e+006	0.2000	um	
6	Zinc	16.9000e+006	4	mil	
7	Tungsten	17.9000e+006	0.2000	um	
8	Lead	4.5500e+006	0.2000	um	
9	Iron	10.0000e+006	0.2000	um	
10	Steel	6.9900e+006	0.6800	mm	
11	[illegible]	[illegible]	[illegible]	[illegible]	

List the properties of the metal material `Aluminium`.

```
s = find(mc,'Aluminium')
```

```
s = struct with fields:
            Name: 'Aluminium'
    Conductivity: 37700000
       Thickness: 30
           Units: 'mil'
        Comments: ''
```

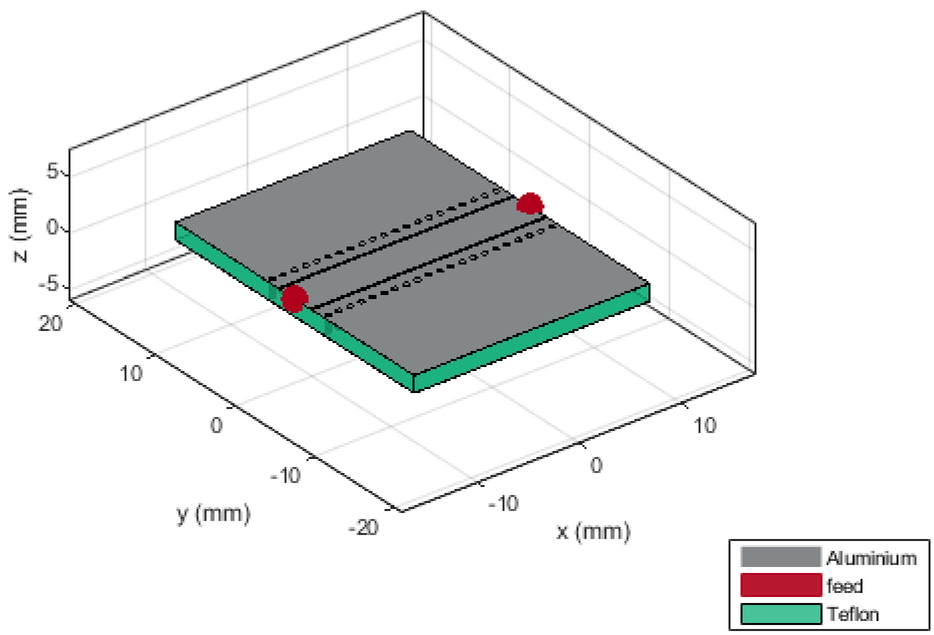
Use the above metal in a coplanar waveguide.

```
m = metal('Aluminium');
waveguide = coplanarWaveguide('Conductor',m)
```

```
waveguide = 
  coplanarWaveguide with properties:

           Length: 0.0231
            Width: 0.0039
          Spacing: 2.0000e-04
       ViaSpacing: [0.0011 0.0070]
      ViaDiameter: 5.0000e-04
           Height: 0.0016
   GroundPlaneWidth: 0.0300
        Substrate: [1×1 dielectric]
        Conductor: [1×1 metal]
```

View the waveguide using show function.

```
figure;
show(waveguide)
```



Input Arguments

name — Name of metal
'PEC' (default) | character vector

Name of the metal from the metal catalog, specified as a character vector.

Example: 'Copper'

Data Types: char

mc — Metal catalog
metal object

Metal catalog, specified as an object.

Data Types: char

Output Arguments

mc — Metal catalog
object

Metal catalog, returned as an object.

s — Parameters of metal
structure

Parameters of the specified metal from the metal catalog, returned as a structure.

See Also

metal

Introduced in R2021a

dielectric

Dielectric material for use as substrate

Syntax

```
d = dielectric(material)
d = dielectric(Name=Value)
```

Description

`d = dielectric(material)` returns dielectric materials for use as a substrate in PCB components.

`d = dielectric(Name=Value)` returns dielectric materials based on the properties specified by one or more `Name,Value` pair arguments.

Examples

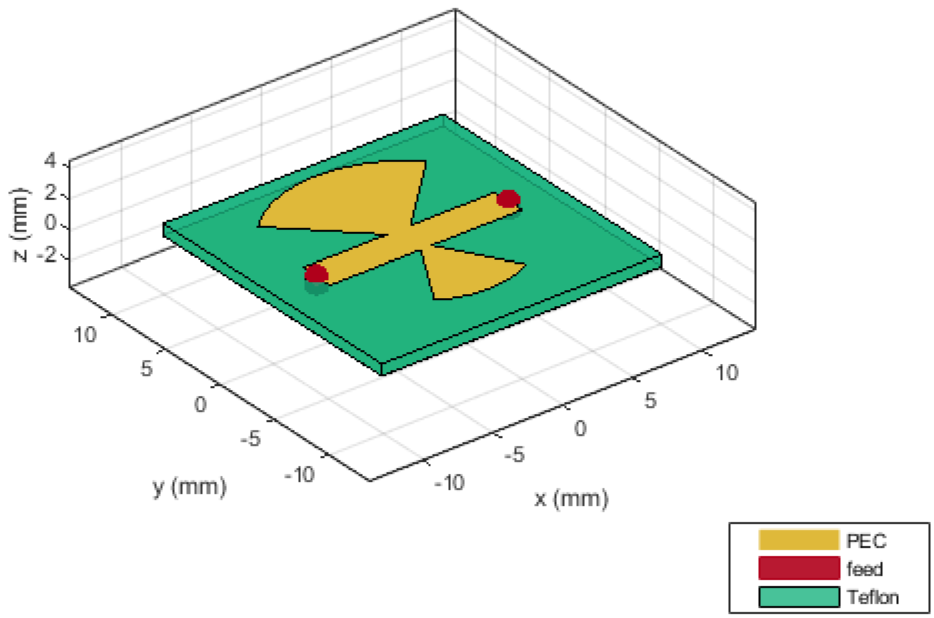
Create Double Shunt Radial Stub

Create shunt radial stub of type double.

```
stub = stubRadialShunt(StubType='double');
stub.OuterRadius = [0.0085 0.0065];
stub.InnerRadius = [0.0012 0.0008];
stub.Angle       = [90 60];
```

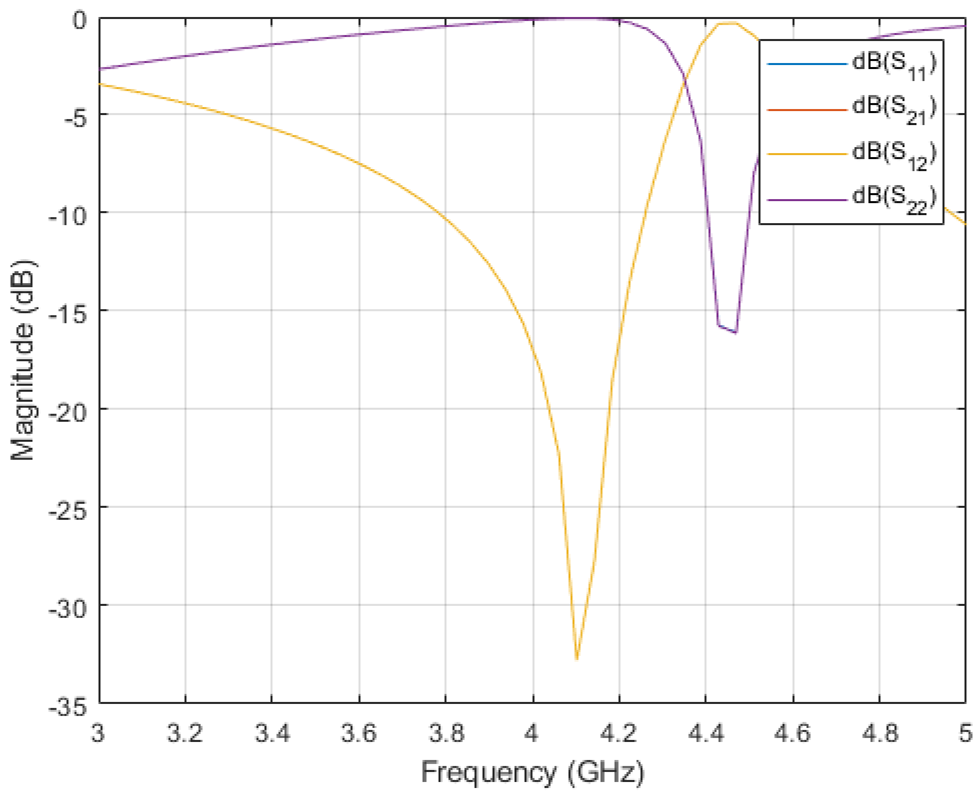
View shunt radial stub.

```
show(stub)
```



Plot s-parameters.

```
spar = sparameters(stub,linspace(3e9,5e9,50));
rfplot(spar)
```



Input Arguments

`material` — Material from dielectric catalog
`'Air'` (default)

Material from the dielectric catalog, specified as one of the values from the `DielectricCatalog`.

Example: `'FR4'`

Data Types: `char`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `Name=Air`

`Name` — Name of dielectric material
character vector

Name of the dielectric material you want to specify in the output, specified as a character vector.

Example: `Name='Taconic_TLC'`

Data Types: `char`

`EpsilonR` — Relative permittivity of dielectric material

`1` | vector

Relative permittivity of the dielectric material, specified as a vector.

Example: `EpsilonR=4.8000`

Data Types: `double`

`LossTangent` — Loss in dielectric material

`0` (default) | vector

Loss in the dielectric material, specified as a vector.

Example: `LossTangent=0.0260`

Data Types: `double`

`Thickness` — Thickness of dielectric material

`0.0060` (default) | vector in meters

Thickness of the dielectric material along default z-axis, specified a vector in meters. This property applies only when you call the function with no output arguments.

Example: `Thickness=0.05`

Data Types: `double`

Output Arguments

d — Dielectric material

object handle

Dielectric material, returned as an object handle. You can use the dielectric material object handle to add dielectric material as a substrate to any PCB component.

See Also

`DielectricCatalog`

Introduced in R2016a

DielectricCatalog

Catalog of dielectric materials

Syntax

`dc = DielectricCatalog`

Description

`dc = DielectricCatalog` creates an object handle for the dielectric catalog.

- To open the dielectric catalog, use `open(dc)`
- To know the properties of a dielectric material from the dielectric catalog, use `s = find(dc, name)`.

Examples

Use Dielectric Catalog to Design Coplanar Waveguide

Open the dielectric catalog.

```
dc = DielectricCatalog;
open(dc)
```

	Name	Relative_Permittivity	Loss_Tangent	Frequency	Comments
1	Air	1	0	1.0000e+009	
2	FR4	4.8000	0.0260	100.0000e+0...	
3	Teflon	2.1000	2.0000e-04	100.0000e+0...	
4	Foam	1.0300	1.5000e-04	50.0000e+006	
5	Polystyrene	2.5500	1.0000e-04	100.0000e+0...	
6	Plexiglas	2.5900	0.0068	10.0000e+009	
7	Fused quartz	3.7800	1.0000e-04	10.0000e+009	
8	E glass	6.2200	0.0023	100.0000e+0...	
9	RO4725JXR	2.5500	0.0022	2.5000e+009	
10	RO4730JXR	3	0.0023	2.5000e+009	
11	TMM3	[illegible]	[illegible]	[illegible]	

List the properties of the dielectric substrate `Foam`.

```
s = find(dc,'Foam')
```

```
s = struct with fields:
                     Name: 'Foam'
    Relative_Permittivity: 1.0300
             Loss_Tangent: 1.5000e-04
                Frequency: 50000000
                 Comments: ''
```

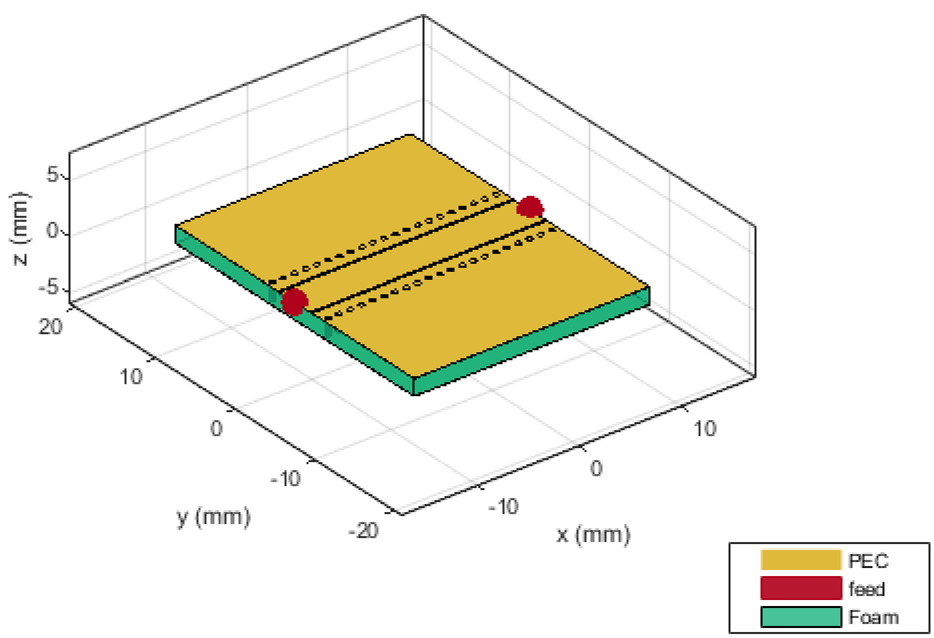
Use the substrate `Foam` in coplanar waveguide.

```
d = dielectric('Foam');
waveguide = coplanarWaveguide('Substrate',d)
```

```
waveguide = 
  coplanarWaveguide with properties:

             Length: 0.0231
              Width: 0.0039
            Spacing: 2.0000e-04
         ViaSpacing: [0.0011 0.0070]
        ViaDiameter: 5.0000e-04
             Height: 0.0016
    GroundPlaneWidth: 0.0300
          Substrate: [1×1 dielectric]
          Conductor: [1×1 metal]
```

View the waveguide.

```
figure;
show(waveguide)
```



Input Arguments

name — Name of dielectric material
'Air' (default) | character vector

Name of a dielectric material from the dielectric catalog, specified as a character vector.

Example: 'FR4'

Data Types: char

dc — Dielectric catalog
object handle

Dielectric catalog, specified as an object handle.

Data Types: char

Output Arguments

dc — Dielectric catalog
object handle

Dielectric catalog, returned as an object handle.

s — Parameters of dielectric material
structure

Parameters of a dielectric material from the dielectric catalog, returned as a structure.

See Also
dielectric

Introduced in R2016a

gerberRead

Create PCBReader object with specified Gerber and drill files

Syntax

```
P = gerberRead(T)
P = gerberRead([],B)
P = gerberRead(T,B)
P = gerberRead(T,B,D)
```

Description

P = gerberRead(T) creates a PCBReader object with the top layer Gerber file specified in T.

P = gerberRead([],B) creates a PCBReader object with the bottom layer Gerber file specified in B.

P = gerberRead(T,B) creates a PCBReader object with the specified top and bottom layer Gerber file names.

P = gerberRead(T,B,D) creates a PCBReader object with the specified top and bottom layer Gerber files and the drill file specified in D .

Examples

Import and View Top Layer Gerber File

Use the gerberRead function to import a top layer Gerber file.

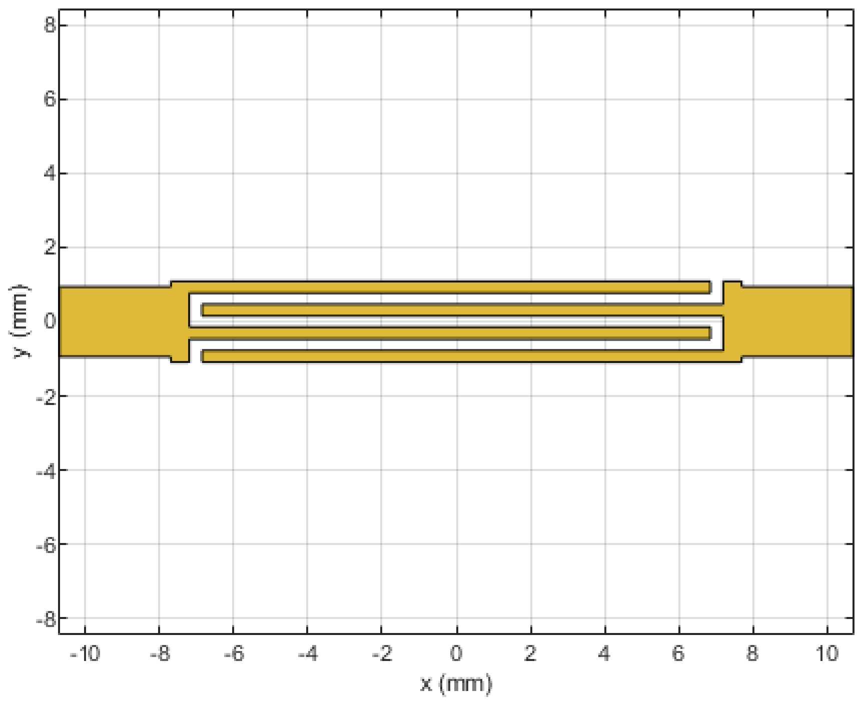
```
P = gerberRead('interdigital_capacitor.gtl');
```

Extract the metal layer from the file using the shapes function.

```
s = shapes(P);
```

View the top metal layer.

```
show(s)
```



Input Arguments

T — Top layer Gerber file
string scalar | character vector

Top layer Gerber file, specified as a character vector or string scalar. The file must be saved as a GTL file.

Example: `gerberRead('Filetop.gtl');`

B — Bottom layer Gerber file
string scalar | character vector

Bottom layer Gerber file, specified as a character vector or string scalar. The file must be saved as a GBL file.

Example: `gerberRead([],'FileBottom.gbl');`

D — Drill file
string scalar | character vector

Drill file, specified a character vector or string scalar. You can specify either a DRL or a TXT file.

Example: `gerberRead('Filetop.gtl','FileBottom.gbl','FileDrill.txt');`

Output Arguments

P — Files read
PCBReader object

Gerber and drill files read, returned as a PCBReader object.

See Also

PCBReader | PCBServices | PCBConnectors | PCBWriter

Introduced in R2021b

gerberWrite

Generate Gerber files

Syntax

```
gerberWrite(designObject)
gerberWrite(designObject,writer)
gerberWrite(designObject,writer,rfConnector)
[a,g] = gerberWrite(designObject,writer,rfConnector)
```

Description

`gerberWrite(designObject)` creates Gerber-format files based on multilayer 2.5D design from PCB component stack up.

`gerberWrite(designObject,writer)` creates a Gerber-format files based on multilayer 2.5D design from PCB component using specified PCB writer services.

`gerberWrite(designObject,writer,rfConnector)` creates Gerber-format files based on multilayer 2.5D design from PCB component using a writer object and an RF connector object.

`[a,g] = gerberWrite(designObject,writer,rfConnector)` returns the `PCBWriter` object , `a` and the path to the location of the Gerber files.

Examples

Generate Gerber Format Files for Default PCB Component

Create a default PCB component.

```
p = pcbComponent;
```

Use 2 Cinch SMA connectors and the Mayhew Labs PCB viewer.

```
W = PCBServices.MayhewWriter;
C1 = PCBConnectors.SMA_Cinch;
C2 = PCBConnectors.SMA_Cinch;
```

Generate the Gerber-format files.

```
[A,g] = gerberWrite(p,W,{C1,C2})
```

```
A = 
  PCBWriter with properties:

                      Design: [1×1 struct]
                      Writer: [1×1 PCBServices.MayhewWriter]
                   Connector: {[1×1 PCBConnectors.SMA_Cinch]  [1×1 PCBConnectors.SMA_Cinch]}
         UseDefaultConnector: 0
  ComponentBoundaryLineWidth: 8
       ComponentNameFontSize: []
```

```
            DesignInfoFontSize: []
                          Font: 'Arial'
                     PCBMargin: 5.0000e-04
                    Soldermask: 'both'
                   Solderpaste: 1

   See info for details

g = 
'C:\Users\vgopalak\OneDrive - MathWorks\Documents\MATLAB\Examples\rfpcb-ex06685827\untitled'
```

Input Arguments

`designObject` — PCB design geometry file
`PCBWriter` object

PCB design geometry file, specified as a `PCBWriter` object.

Example: `a = PCBWriter(p1)` creates a `PCBWriter` object, `a`. `gerberWrite(a)` creates a Gerber file using `a`.

`rfConnector` — RF connector type
`PCBConnector` object

RF connector type, specified as a `PCBConnectors` object.

Example: `c = PCBConnectors.SMA_Cinch; gerberWrite(p1,c)` uses SMA_Cinch RF connector at the feedpoint.

`writer` — PCB service
`PCBServices` object

PCB service, specified as a `PCBServices` object.

Example: `s = PCBServices.MayhewWriter; gerberWrite(p1,s)` uses Mayhew Labs PCB service to create and view the PCB design.

Output Arguments

a — PCB writer
`PCBWriter` object

PCB writer that generated the Gerber files, returned as a `PCBWriter` object.

g — Path to generated Gerber files folder
character vector

Path to generated Gerber files folder, returned as a character vector.

See Also
`PCBServices` | `PCBConnectors` | `PCBWriter`

Introduced in R2021b

add

Boolean unite operation on two RF PCB shapes

Syntax

`c = add(shape1,shape2)`

Description

`c = add(shape1,shape2)` unites `shape1` and `shape2` using the add operation. You can also use the `+` symbol to add the two shapes together.

Examples

Add Two RF PCB Shapes

Create a curved bend shape with a length of 5 m.

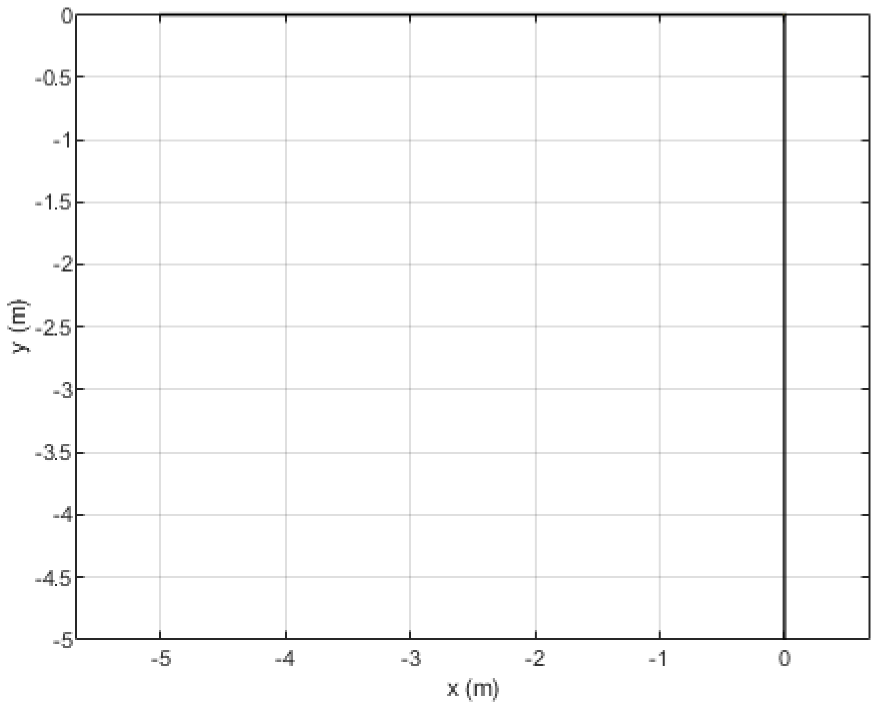
```
bend = bendCurved(Length=[5 5]);
```

Create an annular ring shape with the default inner radius of 5 m.

```
ring = ringAnnular;
```

Add the two shapes and display the result.

```
shapeSum = add(bend,ring);
show(shapeSum)
```



Add Two RF PCB Shapes Using + Operator

Create the default curve shape.

```
shape1 = curve;
```

Create a right angle U-bend shape with an adjusted size and position to complement the curve shape.

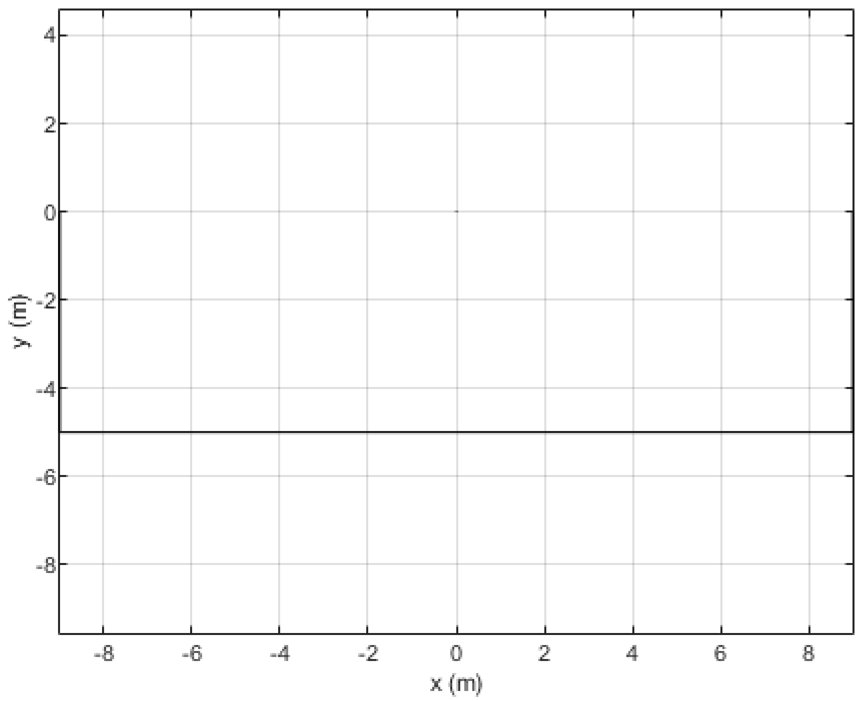
```
shape2 = ubendRightAngle(Length=[5 18 5],ReferencePoint=[0 -5]);
```

Add the two shapes using the + operator, and display the resulting `Polygon` object.

```
shapeSum = shape1+shape2
```

```
shapeSum = 
  Polygon with properties:

        Name: 'mypolygon'
    Vertices: [91x3 double]
```

```
show(shapeSum)
```



Input Arguments

shape1 — First shape
object

First shape created using custom elements and shape objects of RF PCB Toolbox™, specified as an object.

Example: `shape1 = bendCurved;` specifies the first shape as a `bendCurved` object.

shape2 — Second shape
object

Second shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape2 = ringAnnular;` specifies the second shape as a `ringAnnular` object.

See Also

`area` | `intersect` | `subtract` | `rotate` | `rotateX` | `rotateY` | `rotateZ` | `translate` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

and

Shape1 & Shape2 for RF PCB shapes

Syntax

```
c = and(shape1,shape2)
```

Description

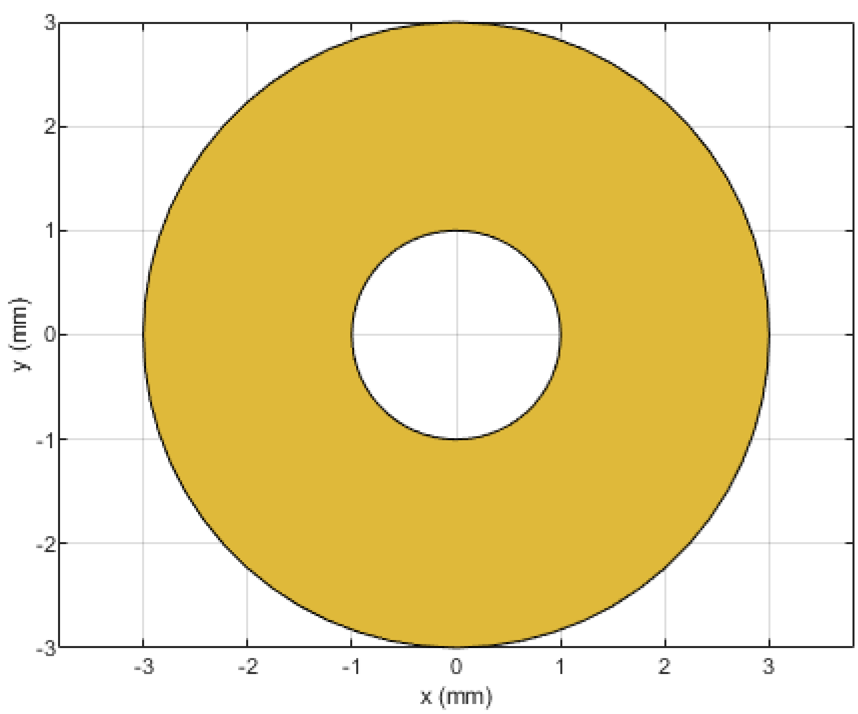
`c = and(shape1,shape2)` calls the syntax `shape1 & shape2` to intersect two shapes.

Examples

Intersect Two RF PCB Shapes

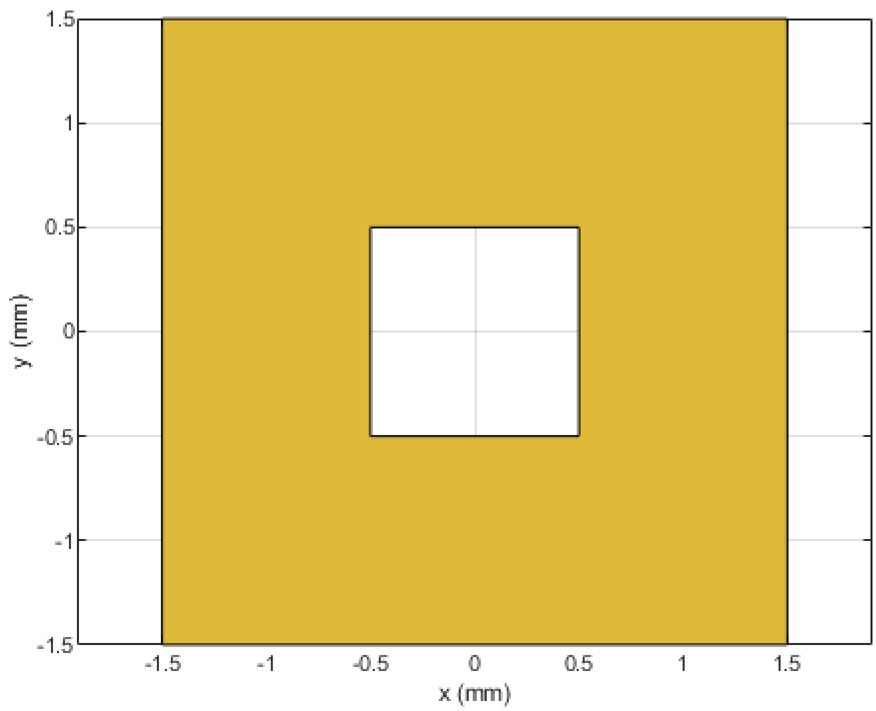
Create and display an annular ring.

```
shape1 = ringAnnular(InnerRadius=1e-3);
show(shape1)
```



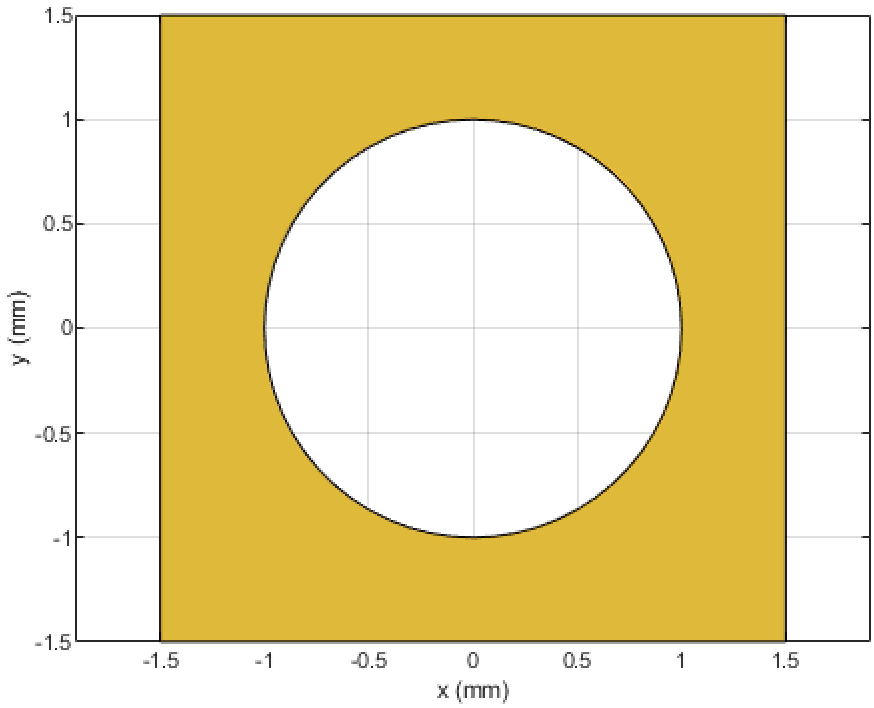
Create and display a square ring.

```
shape2 = ringSquare(InnerSide=1e-3);
show(shape2)
```



Display the intersection of the shapes.

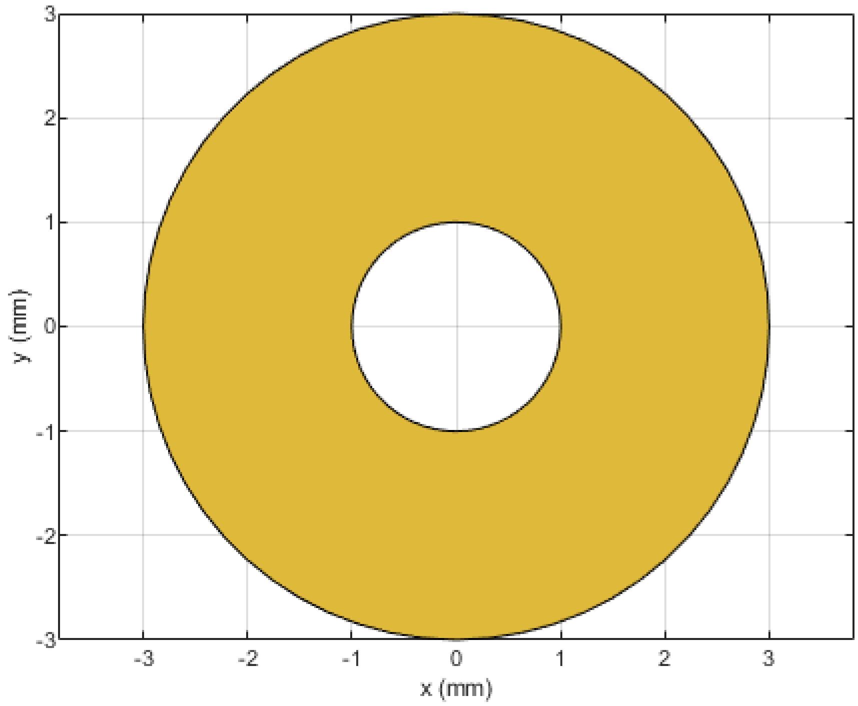
```
shapeIntersection = and(shape1,shape2);
show(shapeIntersection)
```



Intersect Two RF PCB Shapes Using & Operator

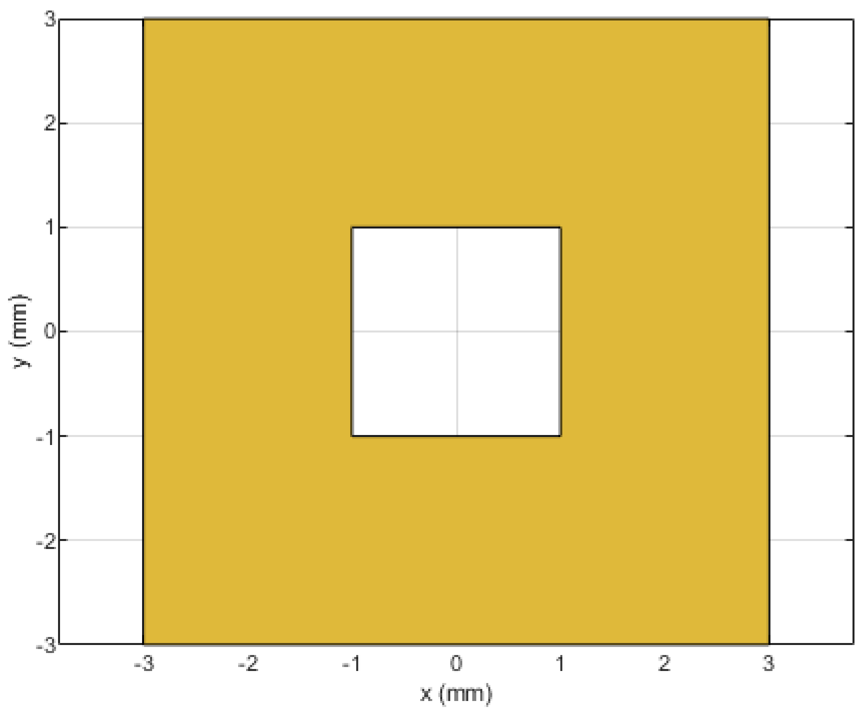
Create and display an annular ring.

```
shape1 = ringAnnular(InnerRadius=1e-3);
show(shape1)
```



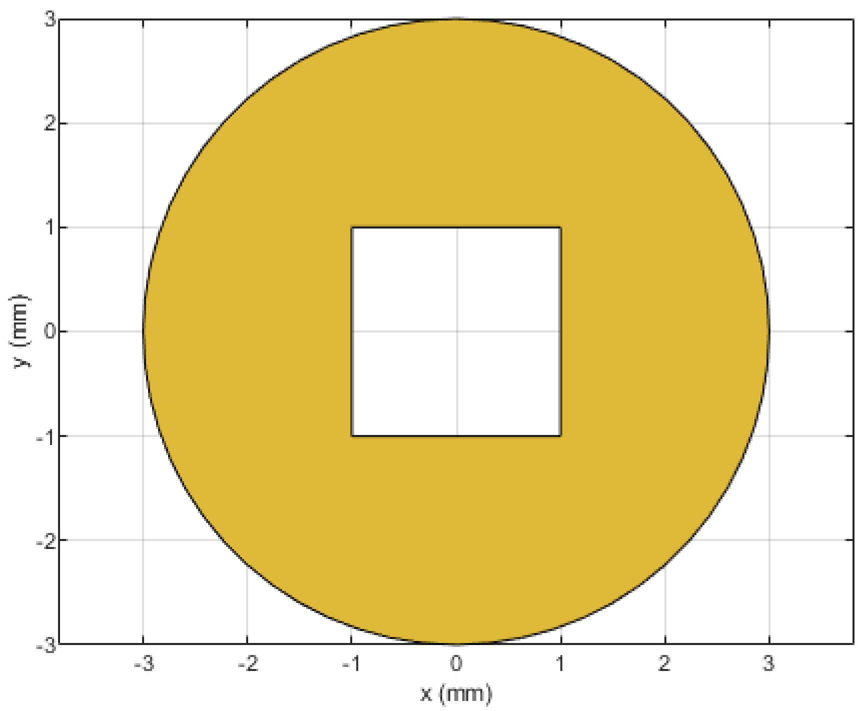
Create and display a square ring.

```
shape2 = ringSquare(InnerSide=2e-3,Width=4e-3);
show(shape2)
```



Display the intersection of the shapes.

```
shapeIntersection = shape1 & shape2;
show(shapeIntersection)
```



Input Arguments

shape1 — First shape
object

First shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape1 = bendCurved;` specifies the first shape as a `bendCurved` object.

shape2 — Second shape
object

Second shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape2 = ringAnnular;` specifies the second shape as a `ringAnnular` object.

See Also

`add` | `subtract` | `area` | `intersect` | `rotate` | `rotateX` | `rotateY` | `rotateZ` | `translate` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

area

Calculate area of RF PCB shape in square meters

Syntax

```
a = area(shape)
```

Description

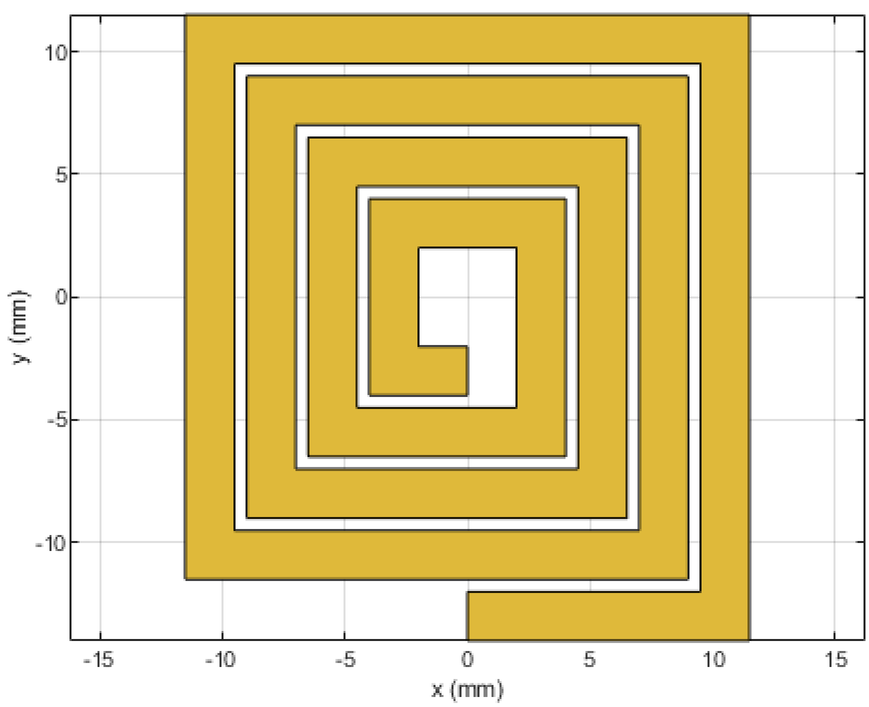
`a = area(shape)` calculates the area of the shape in units of square meters.

Examples

Calculate Area of Spiral Trace

Create and view a default spiral trace.

```
trace = traceSpiral;
show(trace)
```



Get the area of the spiral trace.

```
a = area(trace)
```

```
a = 4.5200e-04
```

Input Arguments

shape — RF PCB shape
object

RF PCB shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape = bendCurved;` specifies the shape as a `bendCurved` object.

See Also

`add` | `subtract` | `intersect` | `rotate` | `rotateX` | `rotateY` | `rotateZ` | `translate` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

intersect

Boolean intersection operation on two RF PCB shapes

Syntax

```
c = intersect(shape1,shape2)
```

Description

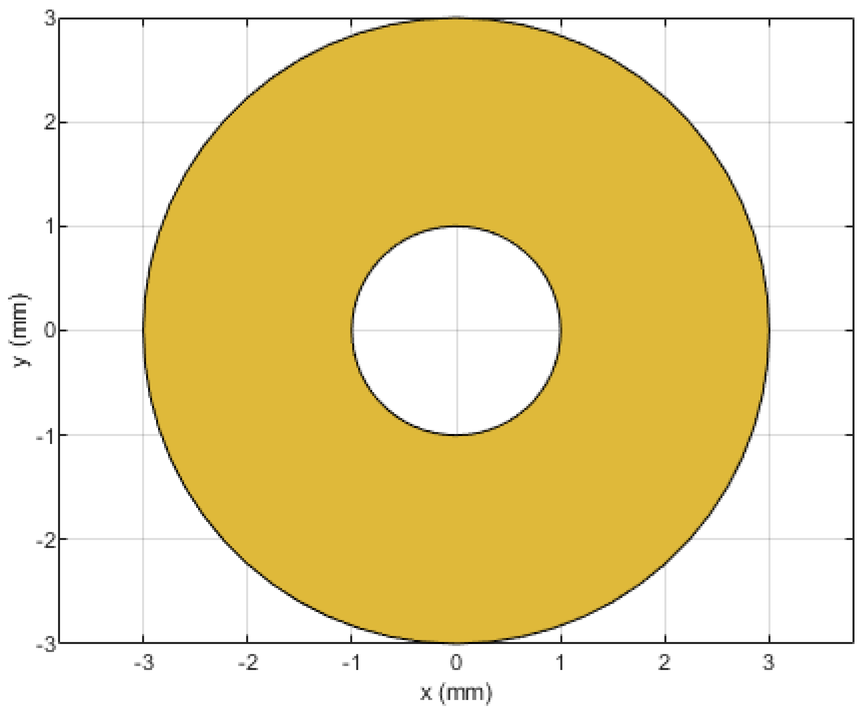
`c = intersect(shape1,shape2)` intersects `shape1` and `shape2` using the intersect operation. You can also use the & to intersect the two shapes.

Examples

Boolean Intersection of Two RF PCB Shapes

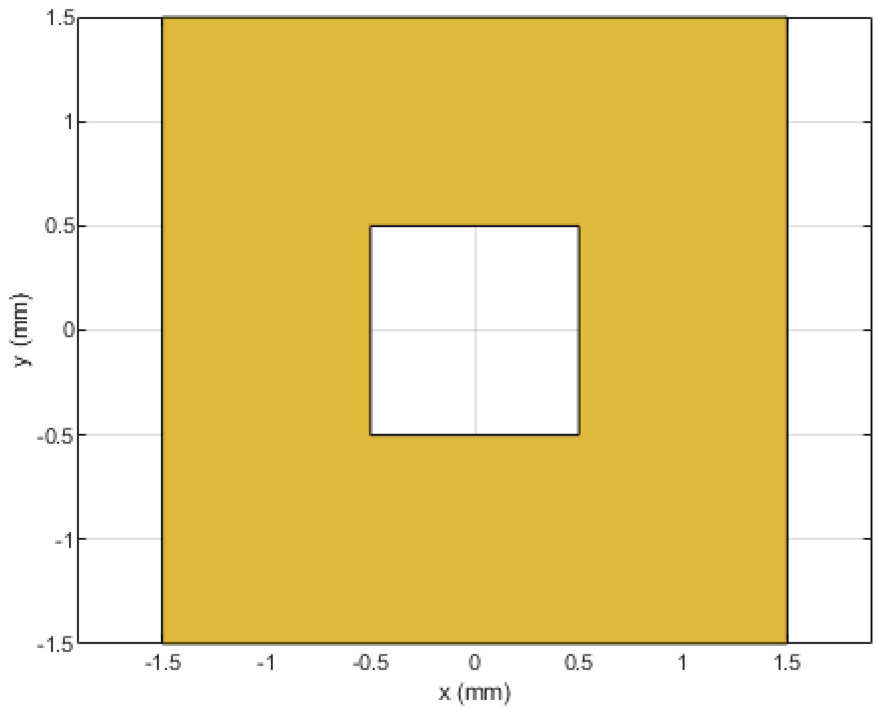
Create and display an annular ring.

```
shape1 = ringAnnular(InnerRadius=1e-3);
show(shape1)
```



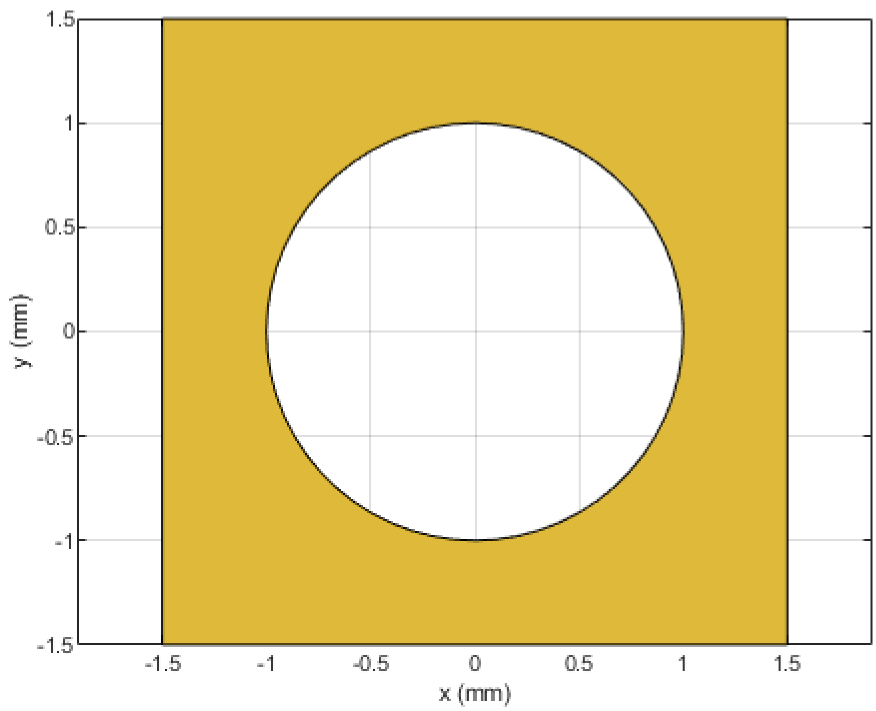
Create and display a square ring.

```
shape2 = ringSquare(InnerSide=1e-3);
show(shape2)
```



Find and display the intersection of the shapes.

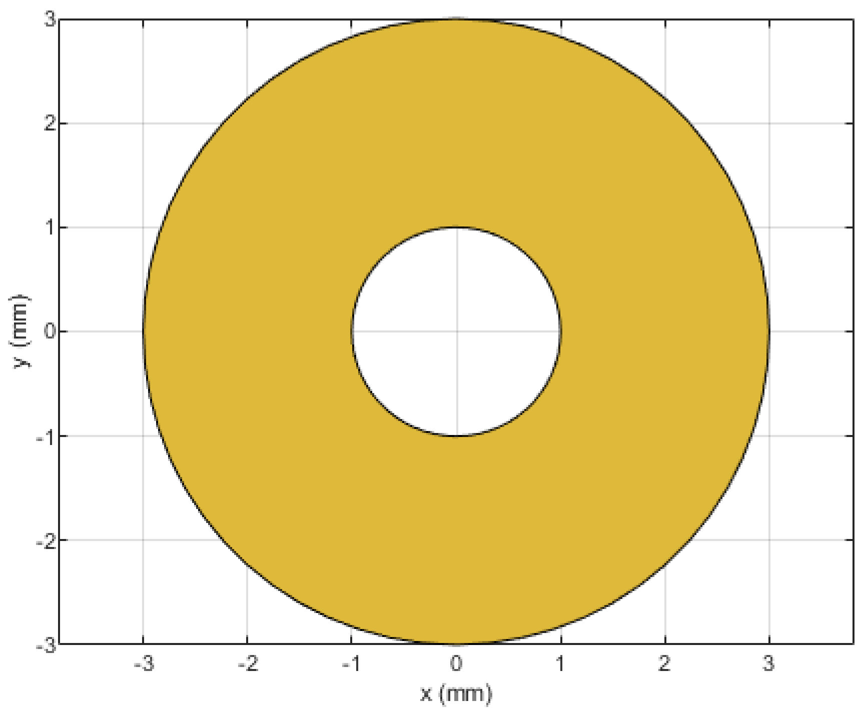
```
shapeIntersection = intersect(shape1,shape2);
show(shapeIntersection)
```



Intersect Two RF PCB Shapes Using & Operator

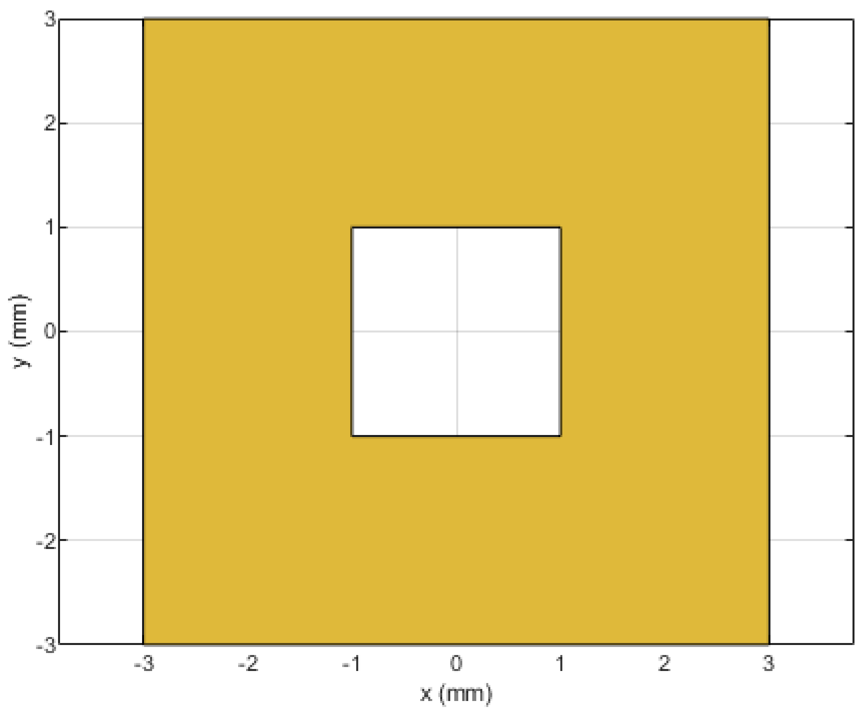
Create and display an annular ring.

```
shape1 = ringAnnular(InnerRadius=1e-3);
show(shape1)
```



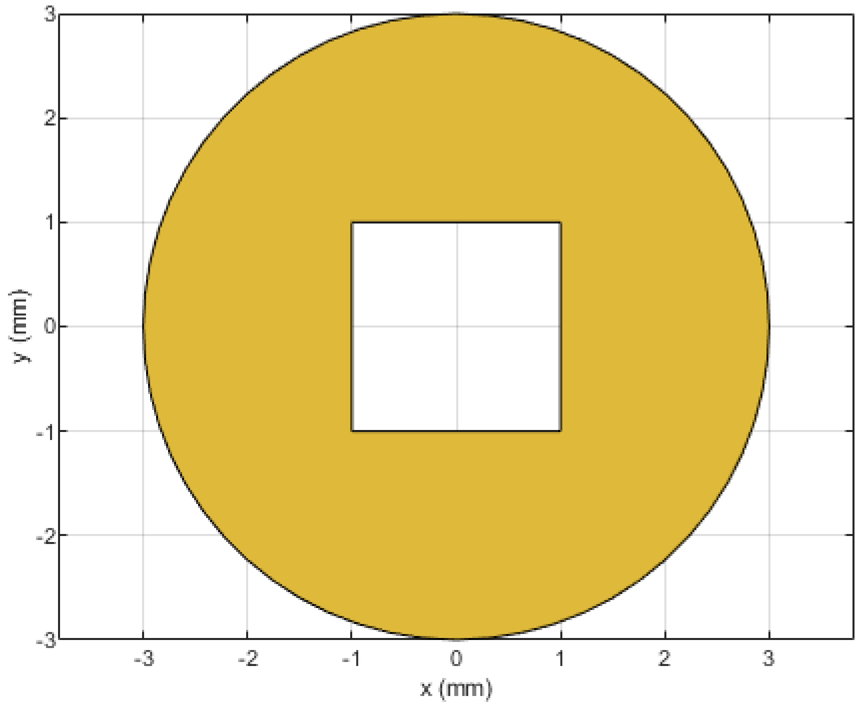
Create and display a square ring.

```
shape2 = ringSquare(InnerSide=2e-3,Width=4e-3);
show(shape2)
```



Display the intersection of the shapes.

```
shapeIntersection = shape1 & shape2;
show(shapeIntersection)
```



Input Arguments

shape1 — First shape
object

First shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape1 = bendCurved;` specifies the first shape as a `bendCurved` object.

shape2 — Second shape
object

Second shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape2 = ringAnnular;` specifies the second shape as a `ringAnnular` object.

See Also

`add` | `subtract` | `area` | `rotate` | `rotateX` | `rotateY` | `rotateZ` | `translate` | `show` | `mesh` | `plot`

Introduced in R2021b

minus

Shape1 - Shape2 for RF PCB shapes

Syntax

```
c = minus(shape1,shape2)
```

Description

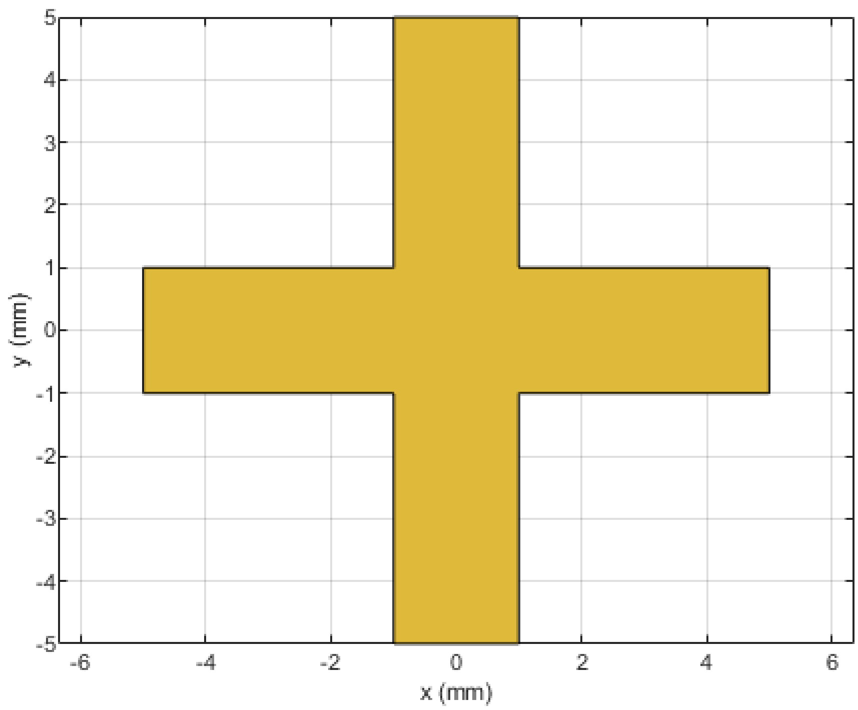
`c = minus(shape1,shape2)` calls the syntax `shape1 - shape2` to subtract two shapes.

Examples

Boolean Subtraction of Two RF PCB Shapes

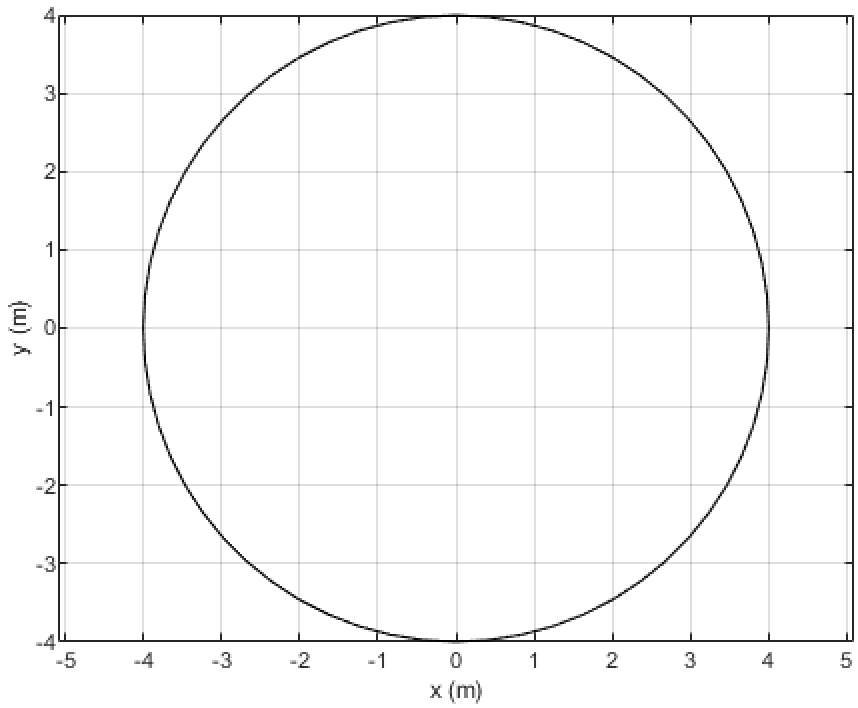
Create and display a cross trace shape.

```
trace = traceCross;
show(trace)
```



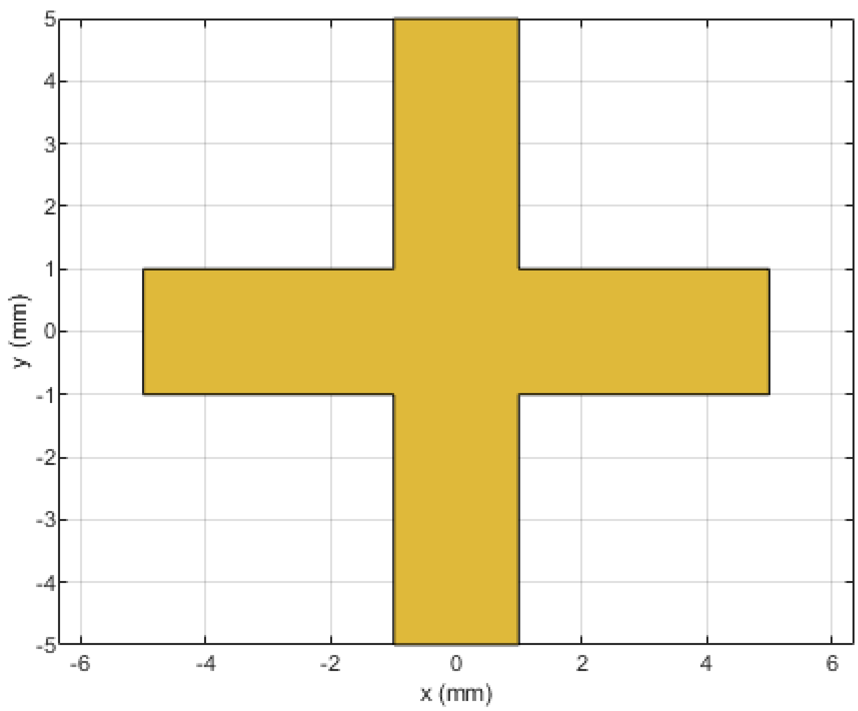
Create and display an annular ring shape with an inner radius of 4 m.

```
ring = ringAnnular(InnerRadius=4);
show(ring)
```



Subtract the annular ring from the cross trace and display the result.

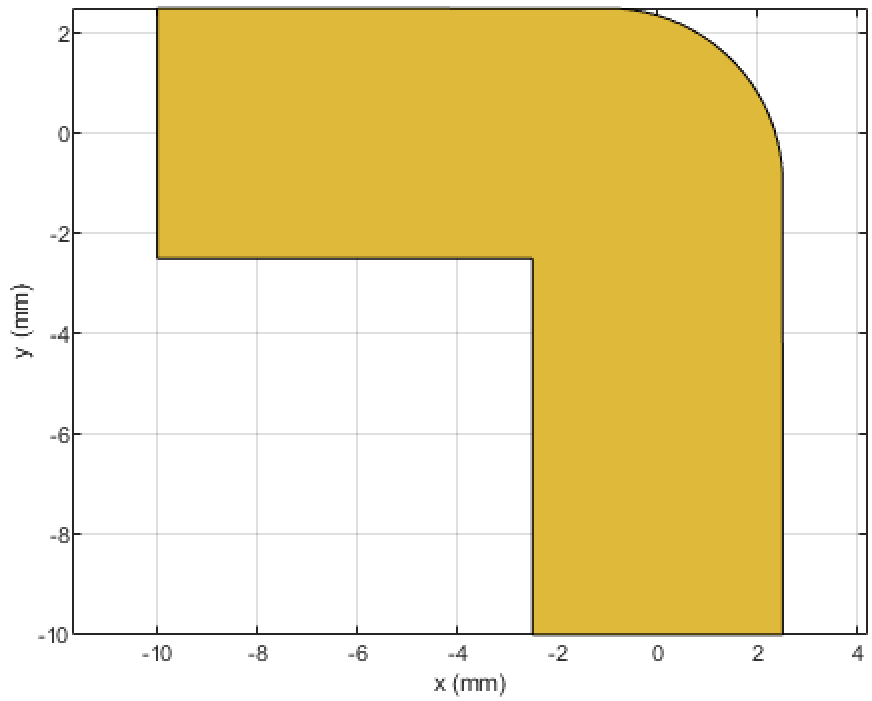
```
shapeDiff = minus(trace,ring);
show(shapeDiff)
```



Subtract Two RF PCB Shapes Using - Operator

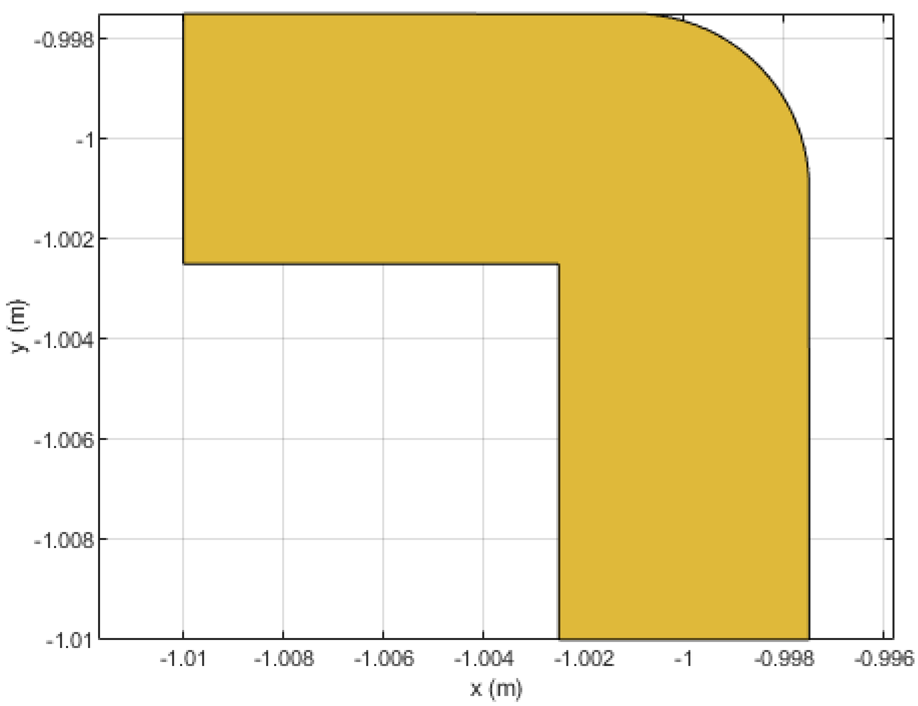
Create and display a curved bend shape.

```
bend1 = bendCurved;
show(bend1)
```



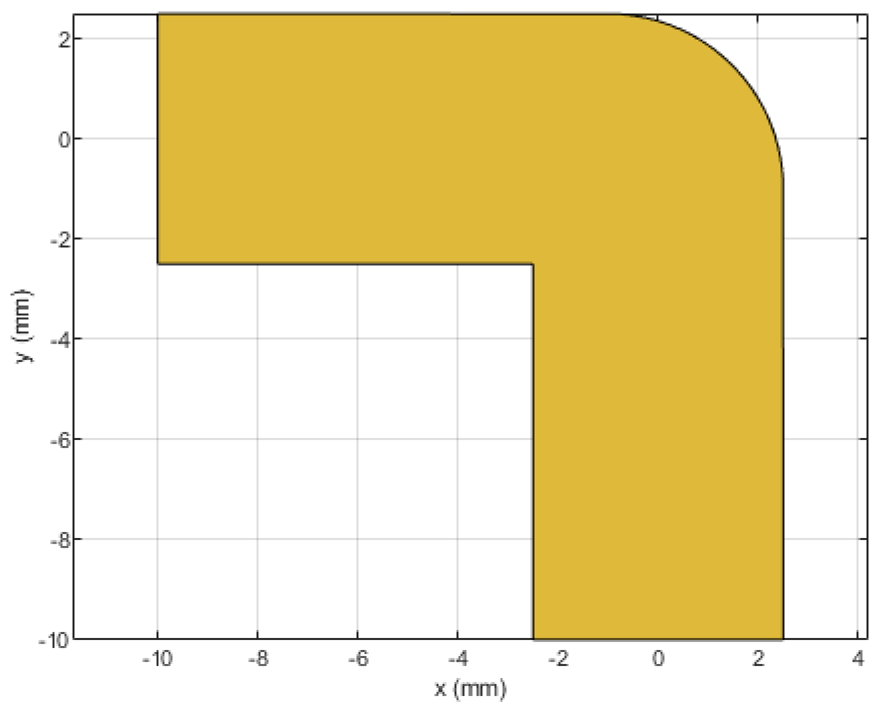
Create and display a curved bend shape with a spatial offset.

```
bend2 = bendCurved(ReferencePoint=[-1 -1]);
show(bend2)
```



Subtract the offset bend from the default bend and display the result.

```
shapeDiff = bend1 - bend2;
show(shapeDiff)
```



Input Arguments

shape1 — First shape
object

First shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape1 = bendCurved;` specifies the first shape as a `bendCurved` object.

shape2 — Second shape
object

Second shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape2 = ringAnnular;` specifies the second shape as a `ringAnnular` object.

See Also

`add` | `subtract` | `area` | `intersect` | `rotate` | `rotateX` | `rotateY` | `rotateZ` | `translate` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

plot

Plot boundary of RF PCB shape

Syntax

```
plot(shape)
plot(shape,Name,Value)
p = plot( ___ )
```

Description

`plot(shape)` plots the boundary of the shape.

`plot(shape,Name,Value)` specifies the line properties using one or more name-value arguments.

Example: `plot(shape,Color="r",LineWidth=2)` plots the boundary of the shape as a red line with a width of 2 pixels.

`p = plot( ___ )` returns the line object. Use `p` to modify properties of the line after it is created.

Examples

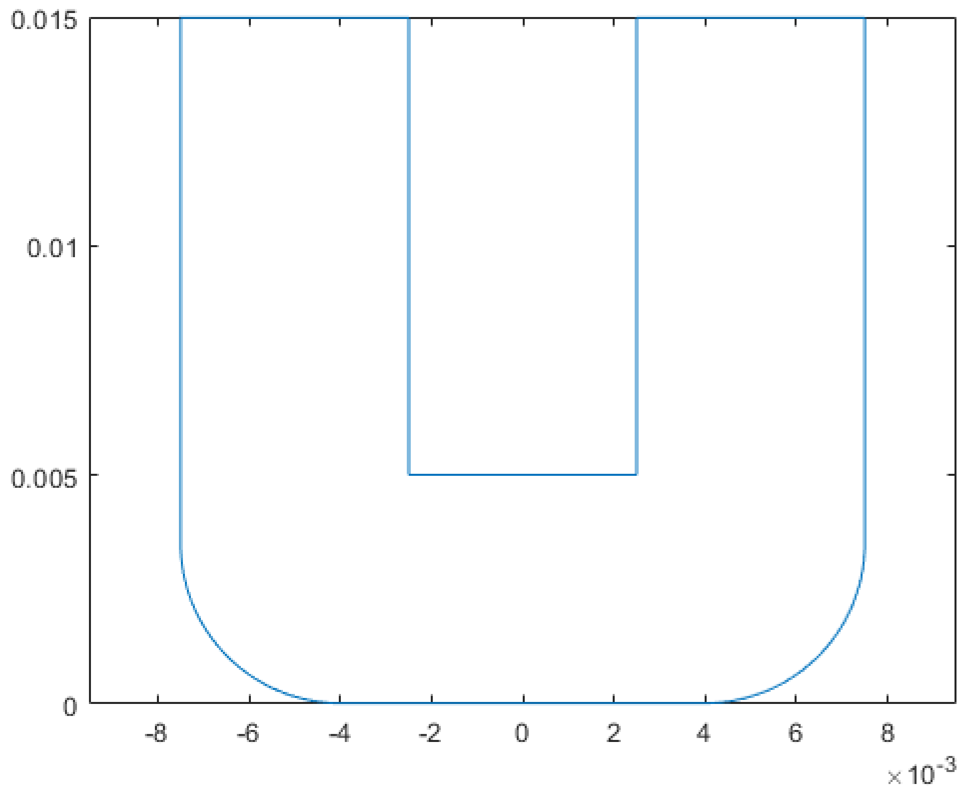
Plot Curved U-Bend

Create a curved U-bend shape.

```
ubend = ubendCurved;
```

Plot the shape.

```
plot(ubend)
```



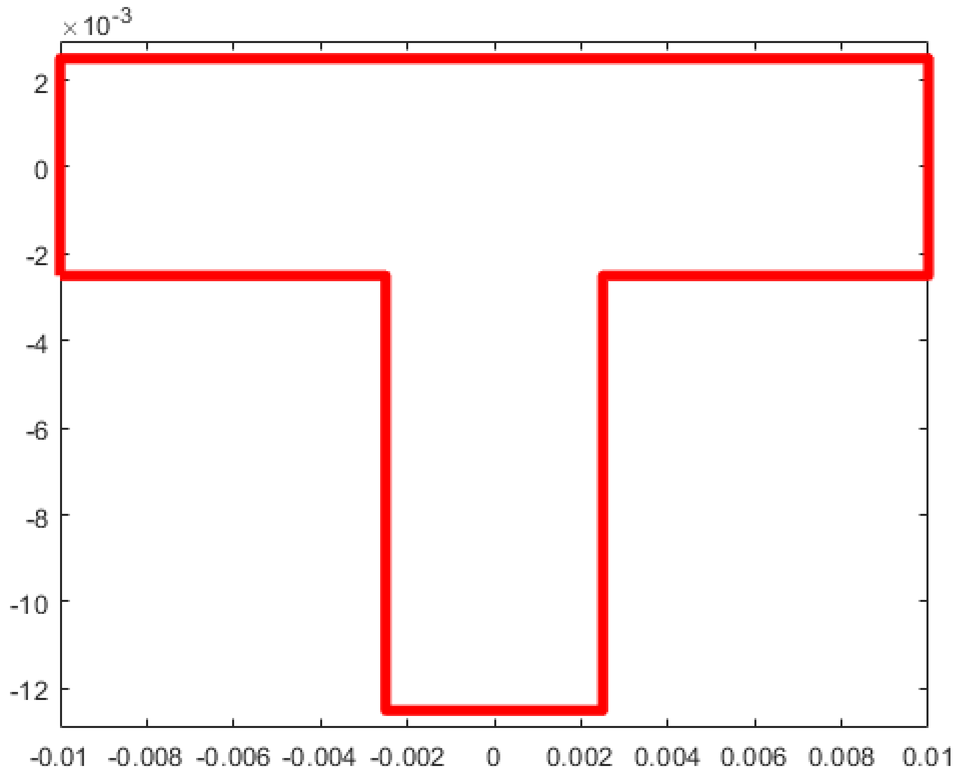
Plot Shape and Modify Line Properties

Create a tee trace shape.

```
trace = traceTee;
```

Plot the shape using a red line of width 4 pixels.

```
plot(trace,LineWidth=4,Color="r")
```



Input Arguments

shape — RF PCB shape
object

RF PCB shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape = bendCurved;` specifies the shape as a `bendCurved` object.

See Also

`show` | `mesh`

Introduced in R2021b

plus

Shape1 + Shape2 for RF PCB shapes

Syntax

`c = plus(shape1,shape2)`

Description

`c = plus(shape1,shape2)` calls the syntax `shape1 + shape2` to unite two shapes.

Examples

Boolean Unite of Two RF PCB Shapes

Create a curved bend shape with a length of 5 m

```
bend = bendCurved(Length=[5 5]);
```

Create an annular ring shape with the default inner radius of 5 m.

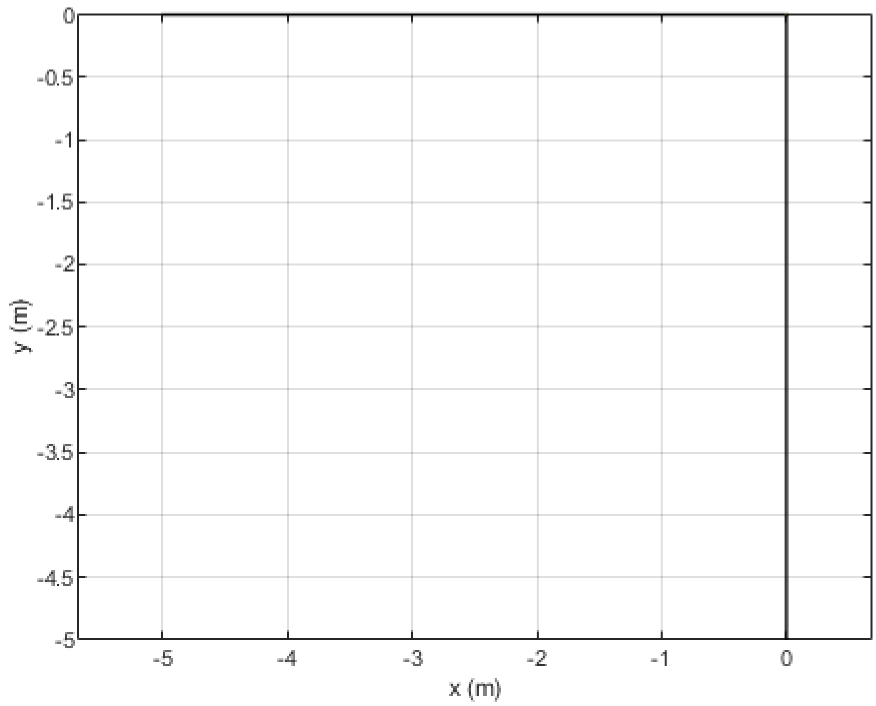
```
ring = ringAnnular;
```

Add the two shapes and display the result.

```
shapeSum = plus(bend,ring)
```

```
shapeSum = 
  Polygon with properties:

        Name: 'mypolygon'
    Vertices: [129x3 double]
```

```
show(shapeSum)
```



Add Two RF PCB Shapes Using + Operator

Create the default curve shape.

```
shape1 = curve;
```

Create a right angle U-bend shape with an adjusted size and position to complement the curve shape.

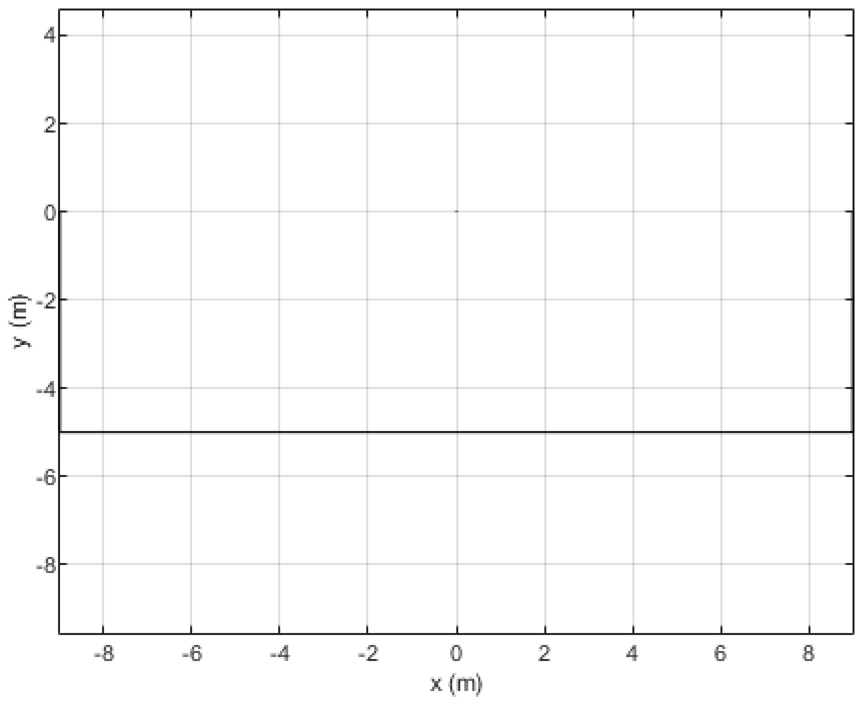
```
shape2 = ubendRightAngle(Length=[5 18 5],ReferencePoint=[0 -5]);
```

Add the two shapes using the + operator, and display the resulting `Polygon` object.

```
shapeSum = shape1+shape2
```

```
shapeSum = 
  Polygon with properties:

        Name: 'mypolygon'
    Vertices: [91x3 double]
```

```
show(shapeSum)
```



Input Arguments

shape1 — First shape
object

First shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape1 = bendCurved;` specifies the first shape as a `bendCurved` object.

shape2 — Second shape
object

Second shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape2 = ringAnnular;` specifies the second shape as a `ringAnnular` object.

See Also

`add` | `subtract` | `area` | `intersect` | `rotate` | `rotateX` | `rotateY` | `rotateZ` | `translate` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

rotate

Rotate RF PCB shape about defined axis

Syntax

`c = rotate(shape,angle,axis1,axis2)`

Description

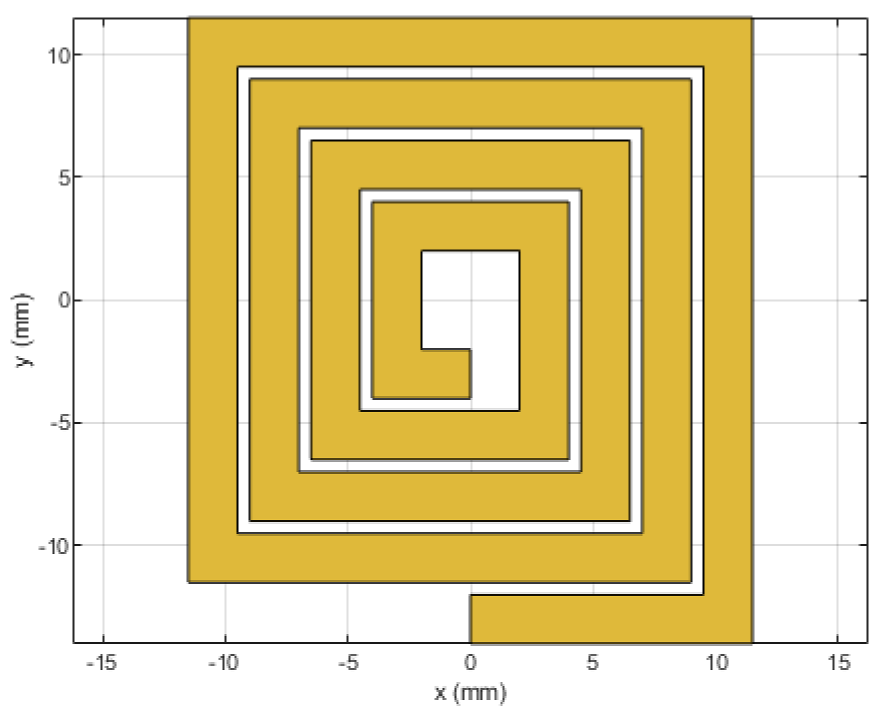
`c = rotate(shape,angle,axis1,axis2)` rotates a shape by a specified angle about an axes defined by two points `axis1` and `axis2`.

Examples

Rotate Spiral Trace About Axis

Create and display a spiral trace.

```
trace = traceSpiral;
show(trace)
```



Specify two points that define the axes of rotation.

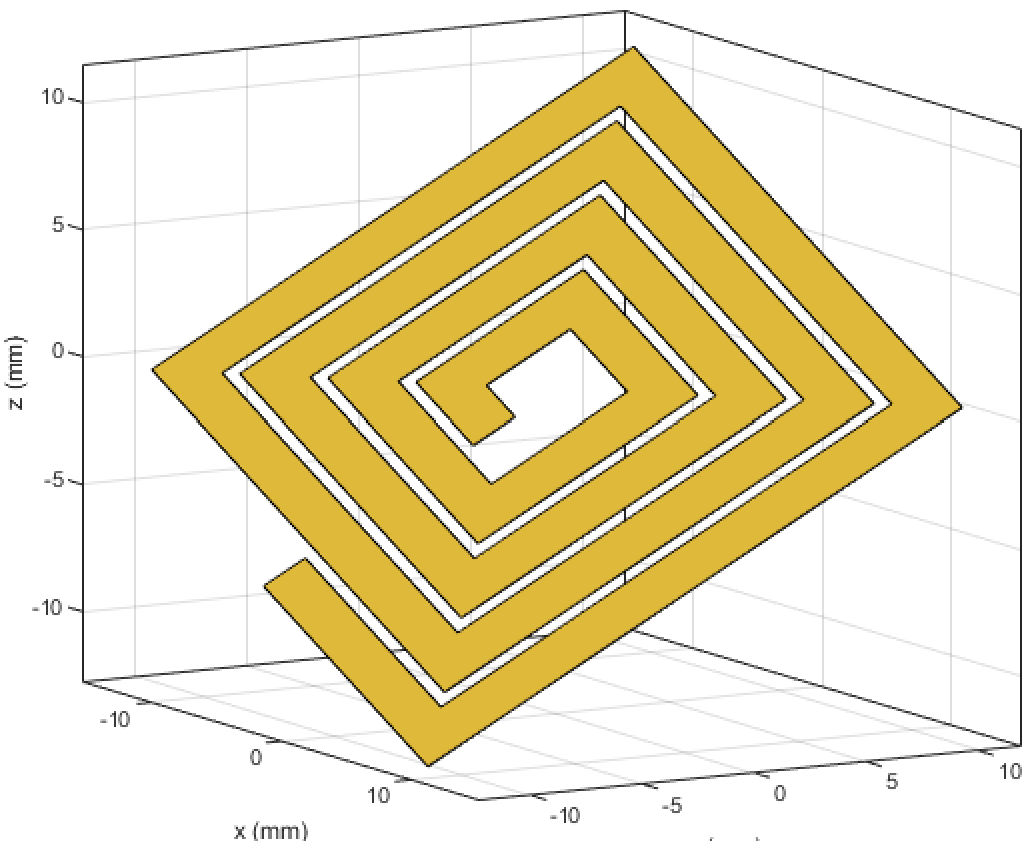
```
pt1 = [0 0 0];
pt2 = [1 1 0];
```

Rotate the spiral trace by 45 degrees about the axis.

```
traceRot = rotate(trace,45,pt1,pt2);
```

Display the rotated shape. Set the camera line of sight to display in 3-D space.

```
show(traceRot)
view(60,10)
```



Input Arguments

shape — RF PCB shape
object

RF PCB shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape = bendCurved;` specifies the shape as a `bendCurved` object.

`angle` — Angle of rotation
scalar

Angle of rotation, specified as a scalar in degrees.

Example: 45 rotates the shape around the axis by 45 degrees.

Data Types: `double`

`axis1` — One point on axis of rotation
three-element vector

One point on the axis of rotation, specified as a three-element vector of Cartesian coordinates in meters.

Example: `[0 0 0]`

Data Types: `double`

`axis2` — Second point on axis of rotation
three-element vector

Second point on the axis of rotation, specified as a three-element vectors of Cartesian coordinates in meters. `axis2` must be different than `axis1`.

Example: `[0 0 1]`

Data Types: `double`

See Also

`add` | `subtract` | `area` | `intersect` | `rotateX` | `rotateY` | `rotateZ` | `translate` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

rotateX

Rotate RF PCB shape about *x*-axis

Syntax

```
c = rotateX(shape,angle)
```

Description

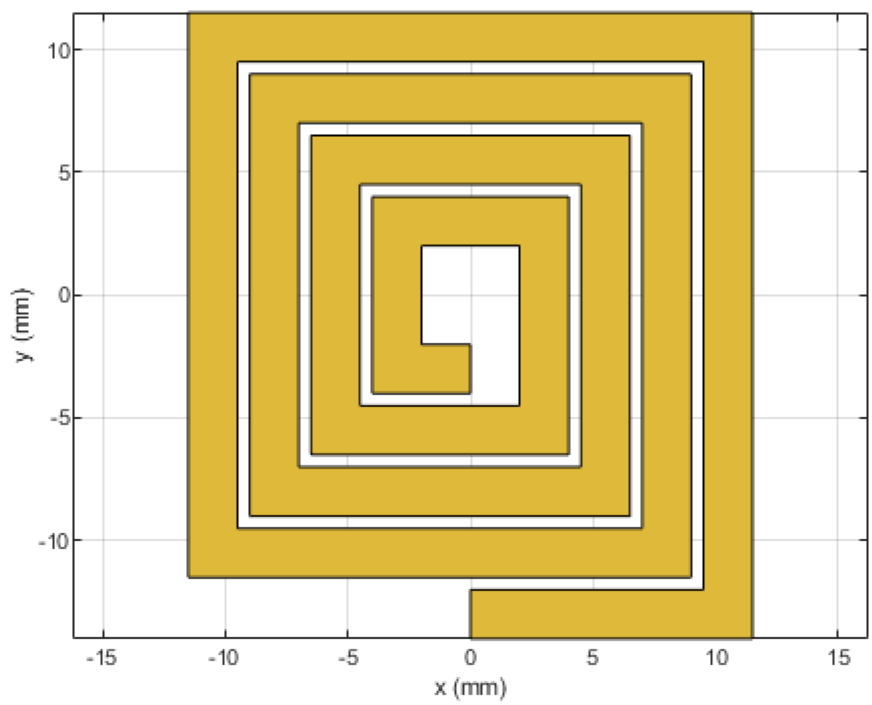
`c = rotateX(shape,angle)` rotates a shape by the specified angle about the *x*-axis.

Examples

Rotate Spiral Trace About X-Axis

Create and display a spiral trace.

```
trace = traceSpiral;
show(trace)
```

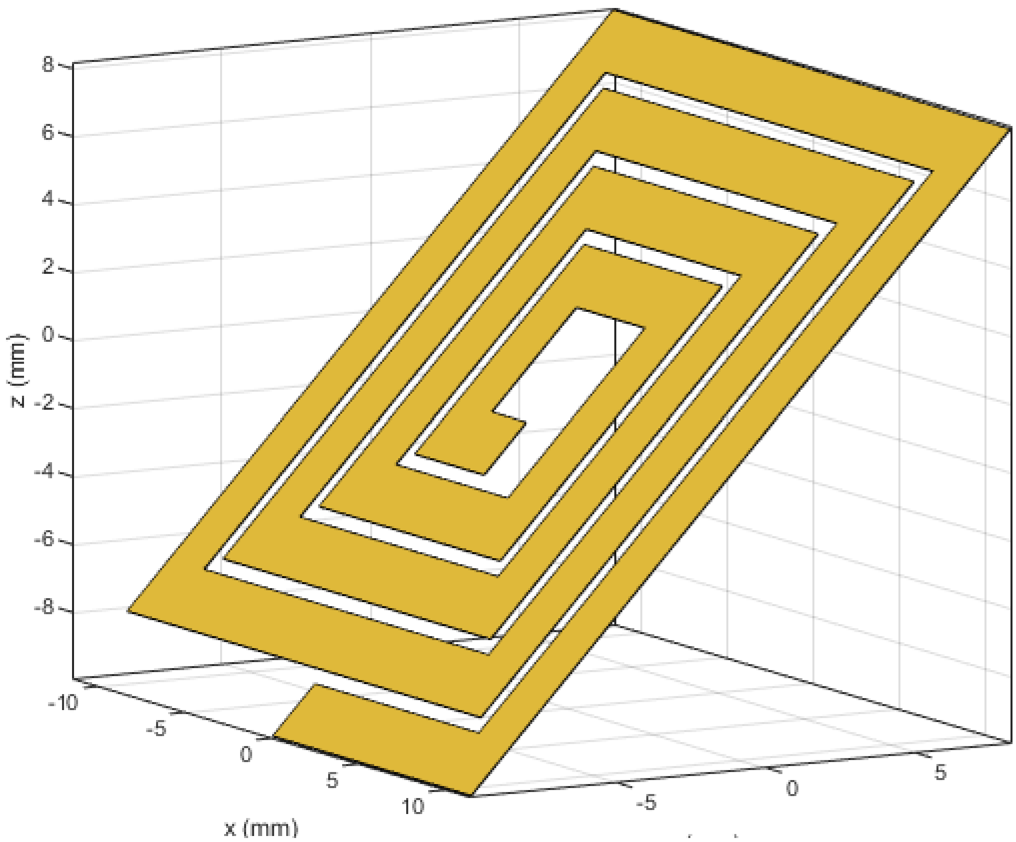


Rotate the spiral trace by 45 degrees about the *x*-axis.

```
traceRotX = rotateX(trace,45);
```

Display the rotated shape. Set the camera line of sight to display in 3-D space.

```
show(traceRotX)
view(60,10)
```



Input Arguments

shape — RF PCB shape
object

RF PCB shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape = bendCurved;` specifies the shape as a `bendCurved` object.

`angle` — Angle of rotation
scalar

Angle of rotation, specified as a scalar in degrees.

Example: `45` rotates the shape around the *x*-axis by 45 degrees.

Data Types: `double`

See Also
`add` | `subtract` | `area` | `intersect` | `rotate` | `rotateY` | `rotateZ` | `translate` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

rotateY

Rotate RF PCB shape about *y*-axis and angle

Syntax

```
c = rotateY(shape,angle)
```

Description

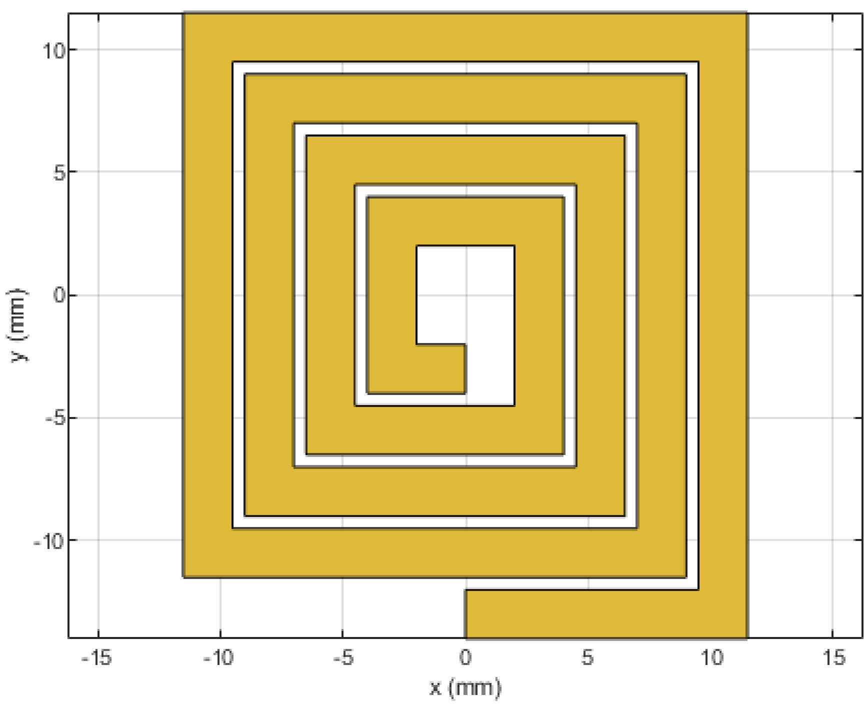
`c = rotateY(shape,angle)` rotates a shape by the specified angle about the *y*-axis.

Examples

Rotate Spiral Trace About Y-Axis

Create and display a spiral trace.

```
trace = traceSpiral;
show(trace)
```

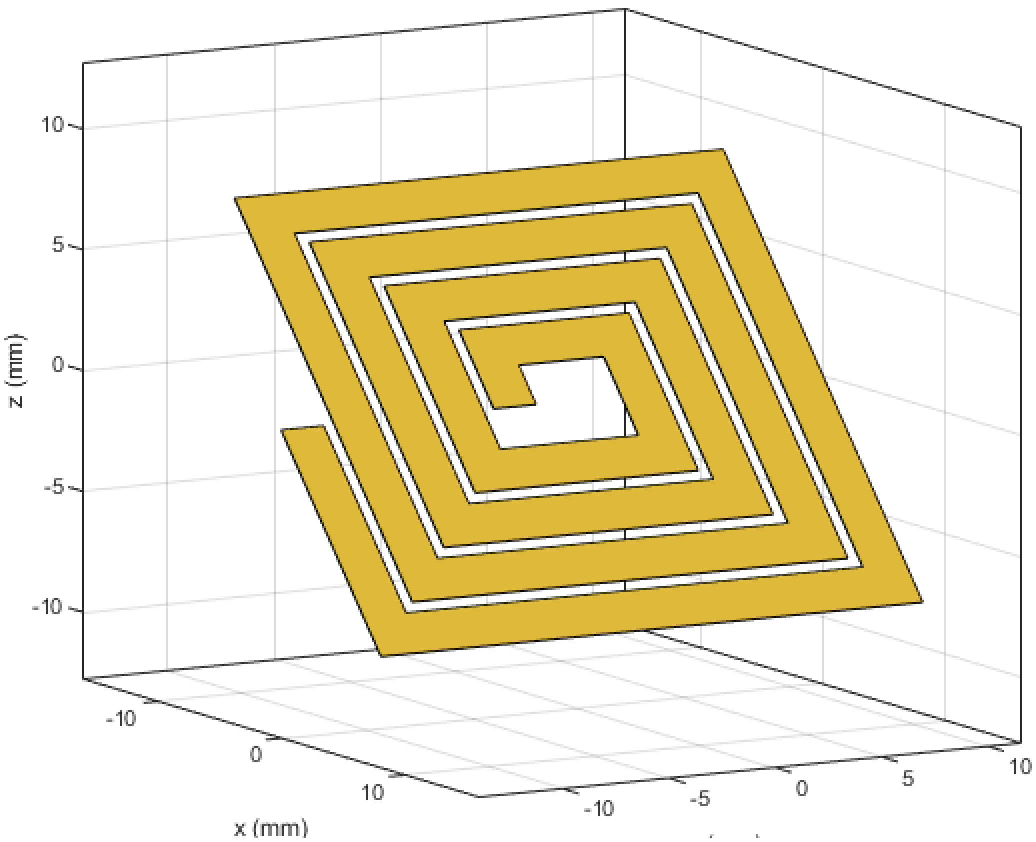


Rotate the spiral trace by 45 degrees about the *y*-axis.

```
traceRotY = rotateY(trace,45);
```

Display the rotated shape. Set the camera line of sight to display in 3-D space.

```
show(traceRotY)
view(60,10)
```



Input Arguments

shape — RF PCB shape
object

RF PCB shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape = bendCurved;` specifies the shape as a `bendCurved` object.

`angle` — Angle of rotation
scalar

Angle of rotation, specified as a scalar in degrees.

Example: 45 rotates the shape around the *y*-axis by 45 degrees.

Data Types: `double`

See Also

`add` | `subtract` | `area` | `intersect` | `rotate` | `rotateX` | `rotateZ` | `translate` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

rotateZ

Rotate RF PCB shape about *z*-axis

Syntax

`c = rotateZ(shape,angle)`

Description

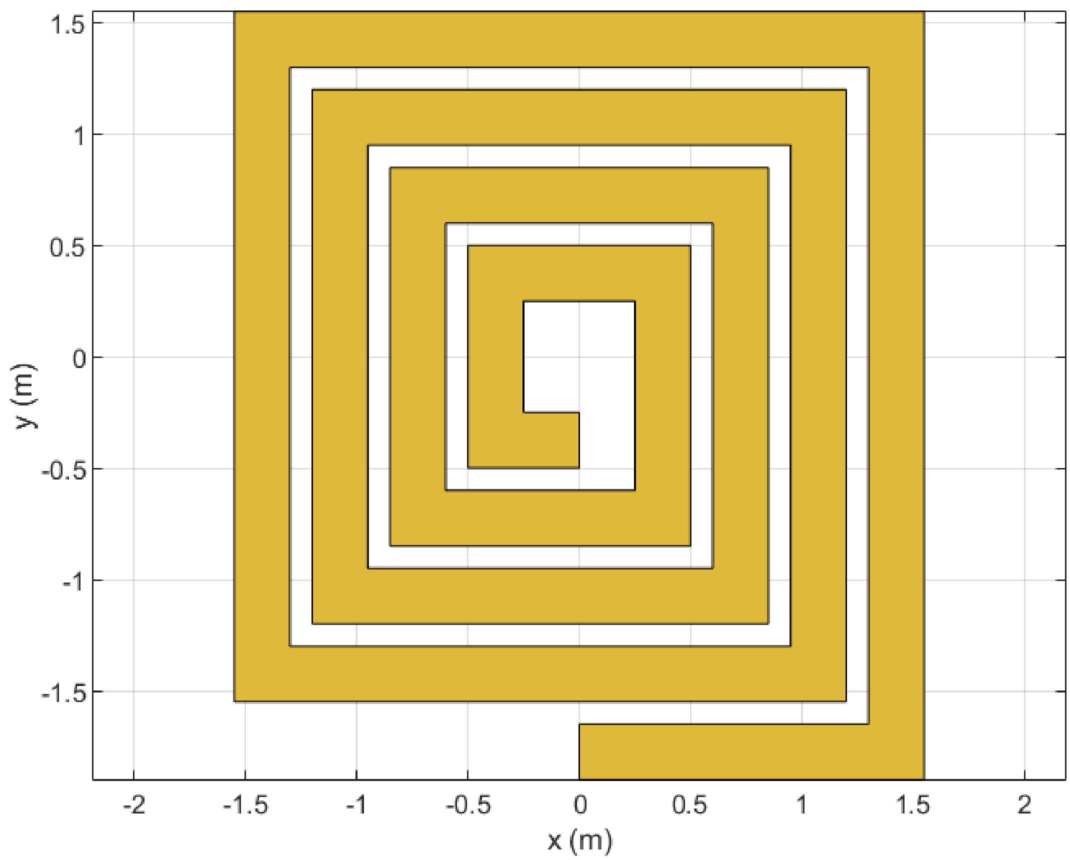
`c = rotateZ(shape,angle)` rotates a shape by the specified angle about the *z*-axis.

Examples

Rotate Spiral Trace About Z-Axis

Create and display a spiral trace.

```
trace = traceSpiral;
show(trace)
```

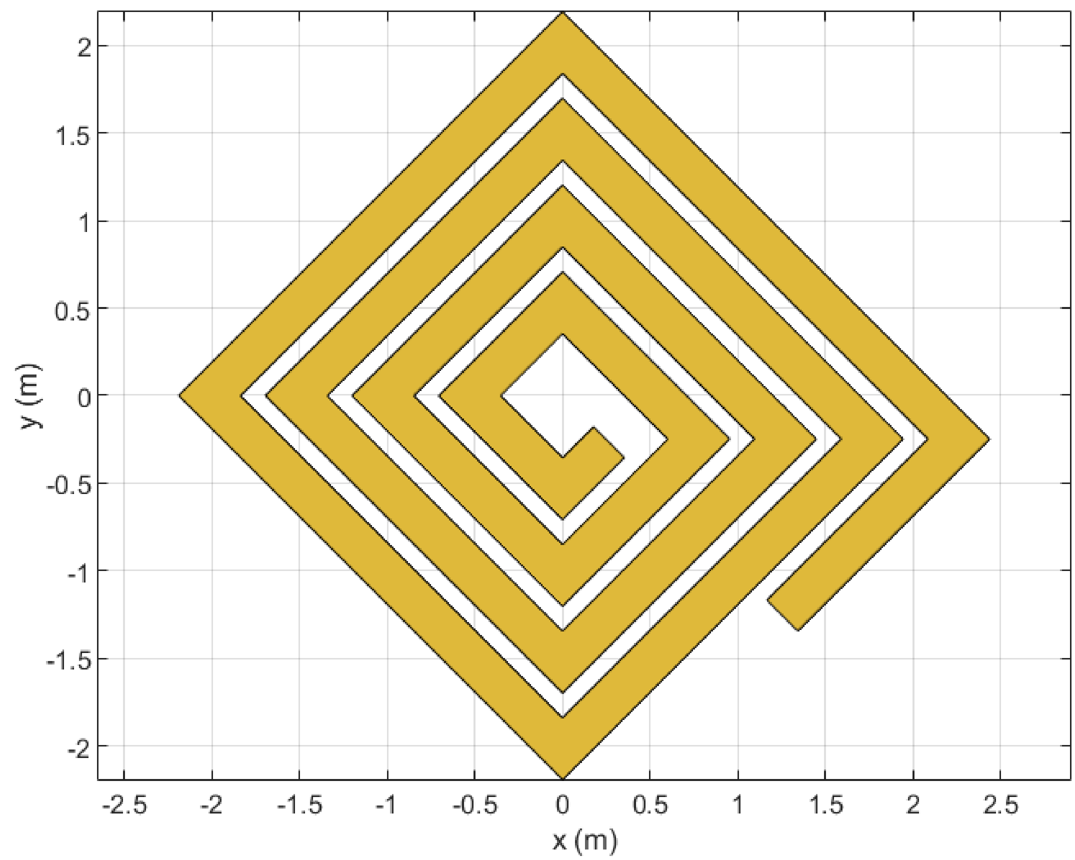


Rotate the spiral trace by 45 degrees about the *z*-axis.

```
traceRotZ = rotateZ(trace,45);
```

Display the rotated shape.

```
show(traceRotZ)
```



Input Arguments

shape — RF PCB shape
object

RF PCB shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape = bendCurved;` specifies the shape as a `bendCurved` object.

angle — Angle of rotation
scalar

Angle of rotation, specified as a scalar in degrees.

Example: 45 rotates the shape around the *z*-axis by 45 degrees.

Data Types: `double`

See Also

`add` | `subtract` | `area` | `intersect` | `rotate` | `rotateX` | `rotateY` | `translate` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

scale

Change size of RF PCB shape by fixed amount

Syntax

```
c = scale(shape,scaleFactor)
c = scale(shape,scaleFactor,RefPoint)
```

Description

`c = scale(shape,scaleFactor)` resizes the shape by a scaling factor.

`c = scale(shape,scaleFactor,RefPoint)` scales the shape by a constant factor with respect to the reference point. The reference point is ignored if the shape is symmetrical and scale is performed based on centroid. The reference point is considered if the shape is unsymmetrical and scale is performed based on specified reference point.

Examples

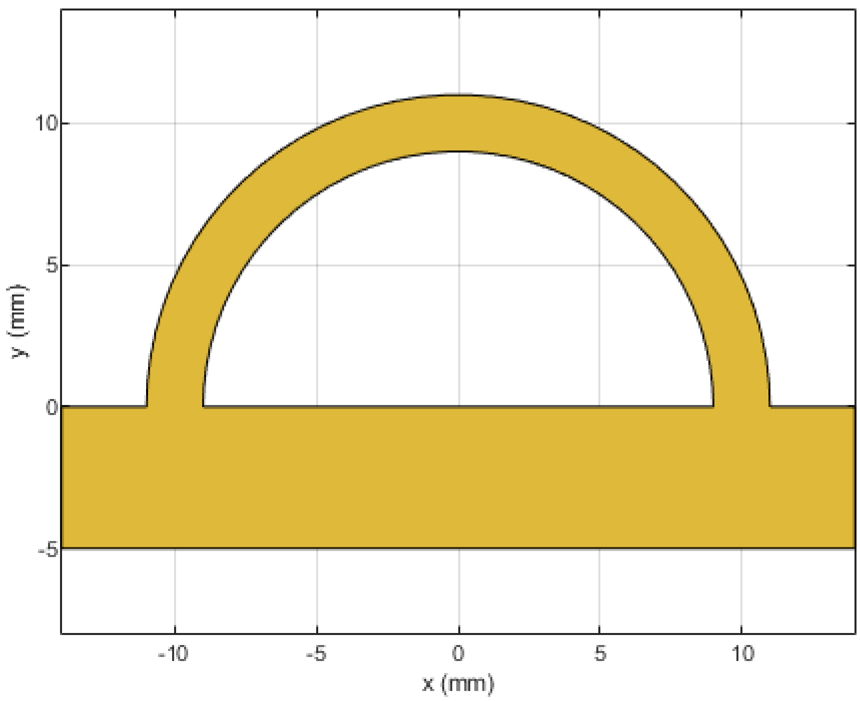
Resize Shape by Scale Factor

Create a shape consisting of a curve and a right angle U-bend.

```
shape1 = curve;
shape2 = ubendRightAngle(Length=[5 18 5]*1e-3,ReferencePoint=[0 -5]*1e-3);
shapeSum = shape1+shape2;
```

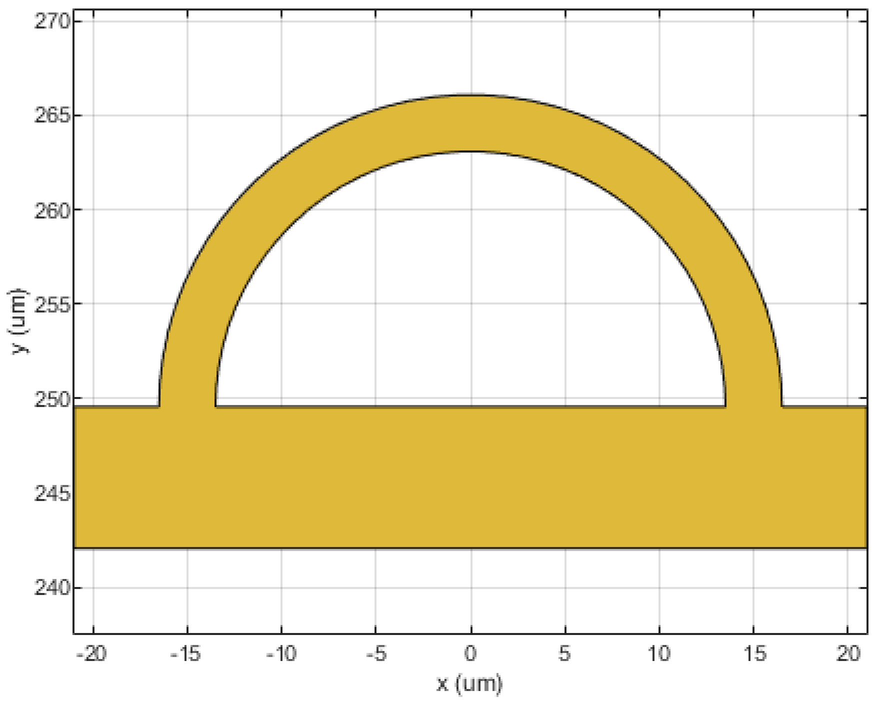
Display the shape.

```
show(shapeSum)
```



Specify a scale factor, then resize the shape. Display the result.

```
s = 1.5*1e-3;
shapeTrans = scale(shapeSum,s);
show(shapeTrans)
```



Input Arguments

shape — RF PCB shape
object

RF PCB shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape = bendCurved;` specifies the shape as a `bendCurved` object.

`scaleFactor` — Scaling factor
scalar

Scaling factor to change shape size, specified as a scalar.

Data Types: `double`

See Also

`add` | `subtract` | `area` | `intersect` | `rotate` | `rotateX` | `rotateY` | `rotateZ` | `show` | `mesh` | `plot`

Introduced in R2021b

subtract

Boolean subtraction operation on two RF PCB shapes

Syntax

```
c = subtract(shape1,shape2)
```

Description

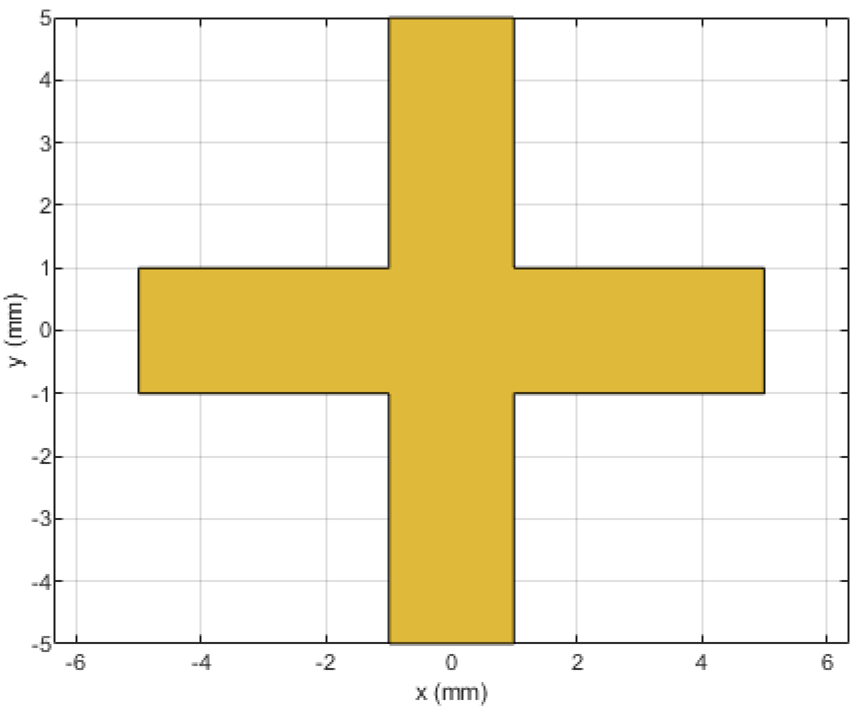
`c = subtract(shape1,shape2)` subtracts `shape1` and `shape2` using the subtract operation. You can also use the `-` symbol to subtract the two shapes.

Examples

Subtract Two RF PCB Shapes

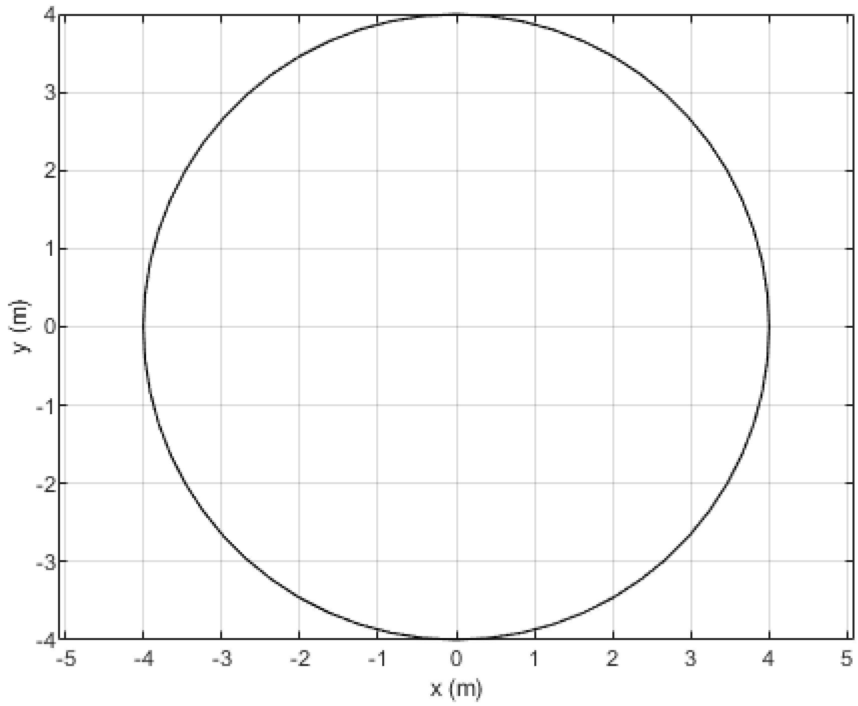
Create and display a cross trace shape.

```
trace = traceCross;
show(trace)
```



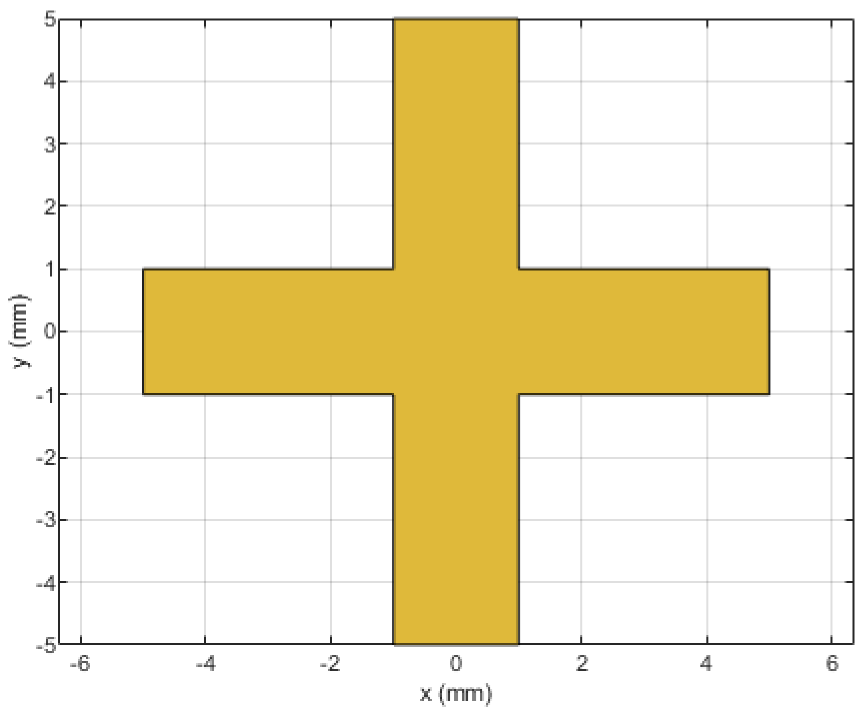
Create and display an annular ring shape with an inner radius of 4 m.

```
ring = ringAnnular(InnerRadius=4);
show(ring)
```



Subtract the annular ring from the cross trace and display the result.

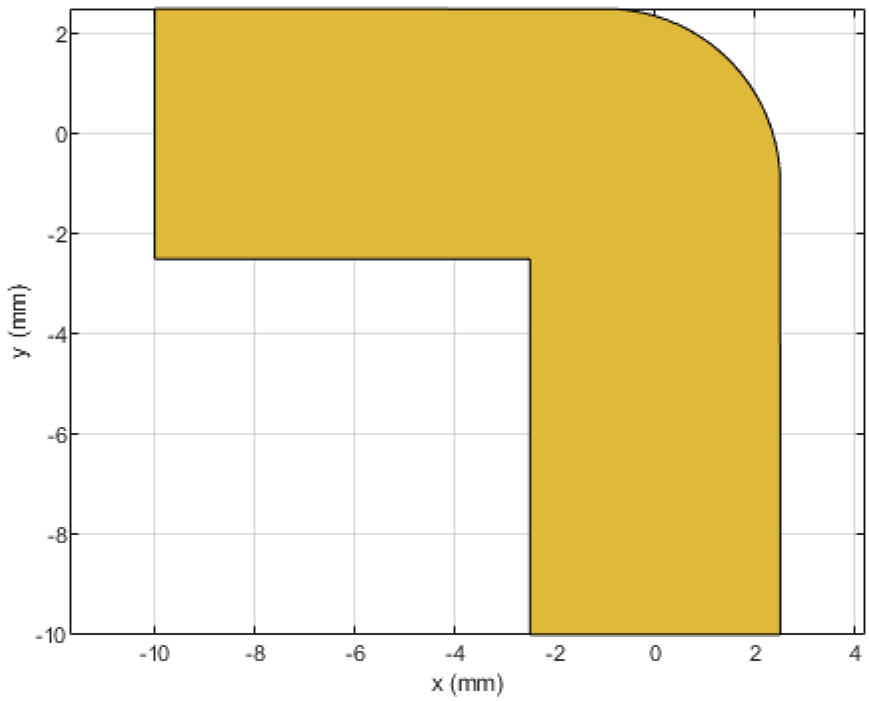
```
shapeDiff = subtract(trace,ring);
show(shapeDiff)
```



Subtract Two RF PCB Shapes Using - Operator

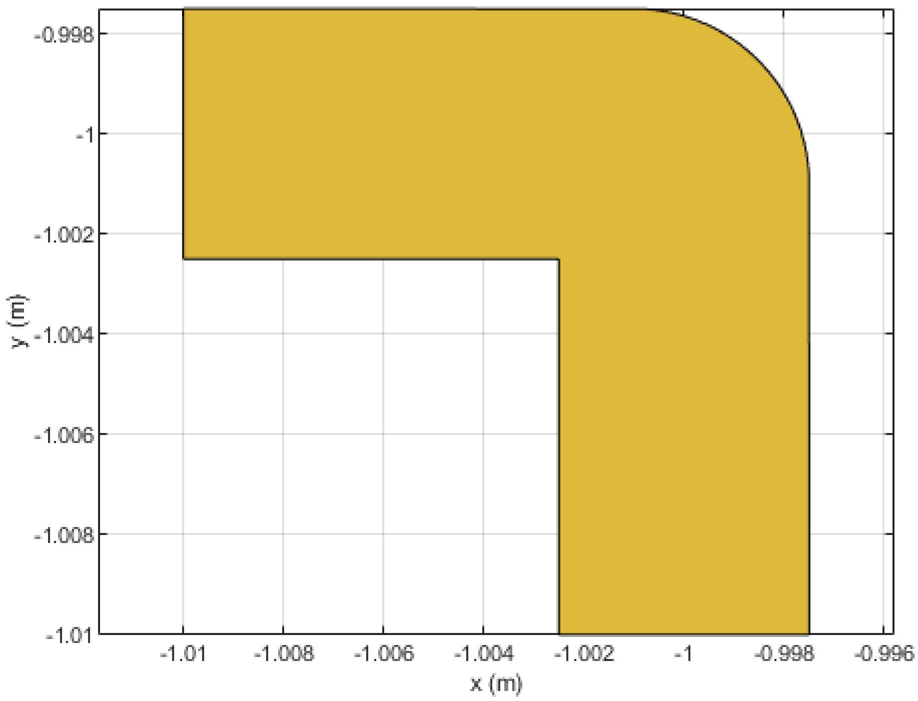
Create and display a curved bend shape.

```
bend1 = bendCurved;
show(bend1)
```



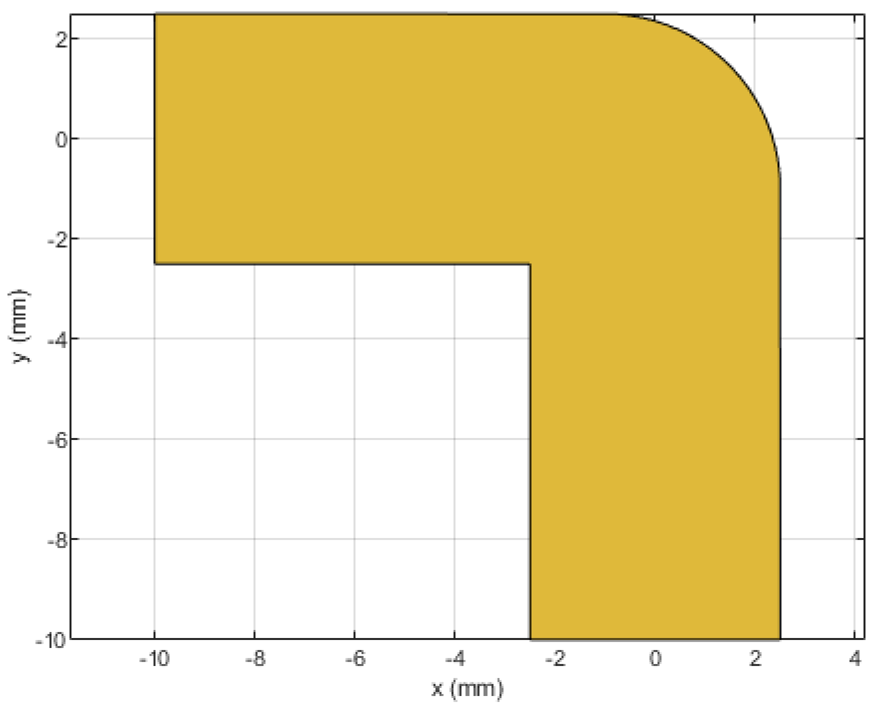
Create and display a curved bend shape with a spatial offset.

```
bend2 = bendCurved(ReferencePoint=[-1 -1]);
show(bend2)
```



Subtract the offset bend from the default bend and display the result.

```
shapeDiff = bend1 - bend2;
show(shapeDiff)
```



Input Arguments

shape1 — First shape
object

First shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape1 = bendCurved;` specifies the first shape as a `bendCurved` object.

shape2 — Second shape
object

Second shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape2 = ringAnnular;` specifies the second shape as a `ringAnnular` object.

See Also

`add` | `area` | `intersect` | `rotate` | `rotateX` | `rotateY` | `rotateZ` | `translate` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

translate

Move RF PCB shape to new location

Syntax

```
c = translate(shape,offset)
```

Description

`c = translate(shape,offset)` moves the shape to a new specified location using a translation vector.

Examples

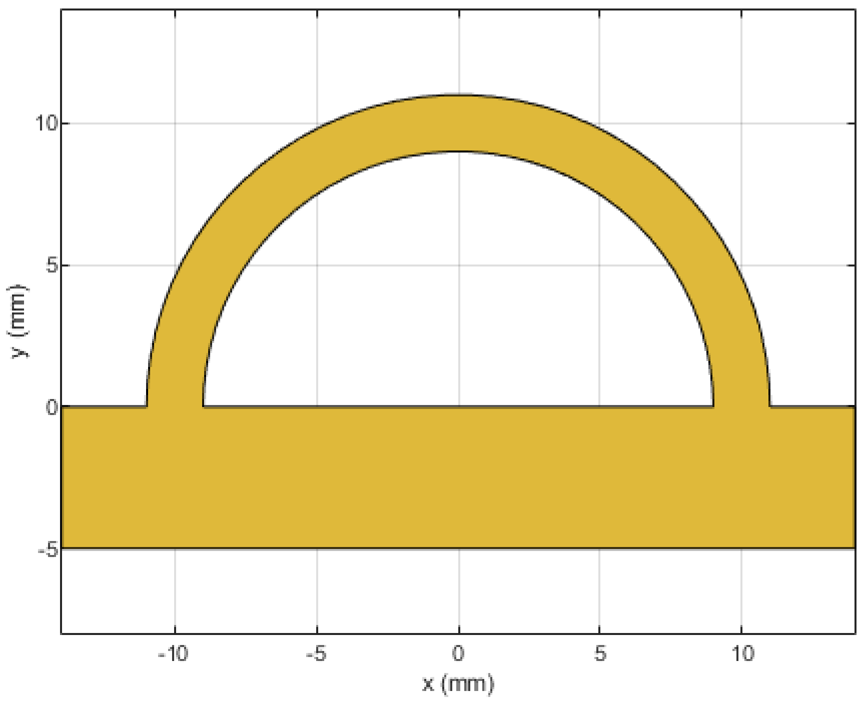
Translate Shape in XY Plane

Create a shape consisting of a curve and a right angle U-bend.

```
shape1 = curve;
shape2 = ubendRightAngle(Length=[5 18 5]*1e-3,ReferencePoint=[0 -5]*1e-3);
shapeSum = shape1+shape2;
```

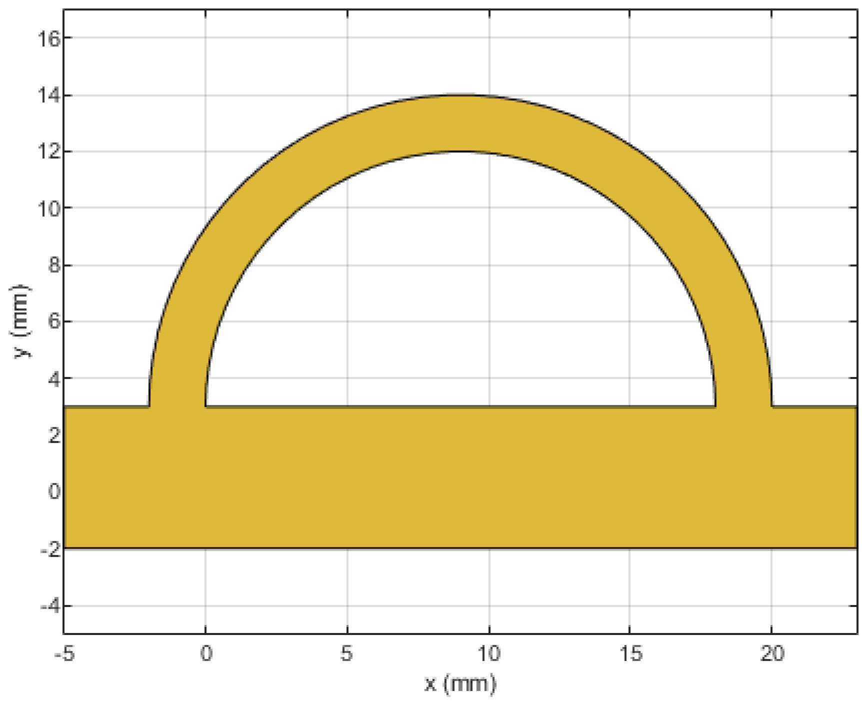
Display the shape.

```
show(shapeSum)
```



Specify a translation vector, then translate the shape in the X-Y plane. Display the result.

```
t = [9 3 0]*1e-3;
shapeTrans = translate(shapeSum,t);
show(shapeTrans)
```



Input Arguments

shape — RF PCB shape
object

RF PCB shape created using custom elements and shape objects of RF PCB Toolbox, specified as an object.

Example: `shape = bendCurved;` specifies the shape as a `bendCurved` object.

`offset` — Translation vector
vector

Translation vector, specified as a vector.

Data Types: `double`

See Also

`add` | `subtract` | `area` | `intersect` | `rotate` | `rotateX` | `rotateY` | `rotateZ` | `show` | `mesh` | `plot` | `scale`

Introduced in R2021b

meshconfig

Change mesh mode of PCB component or shape structure

Syntax

```
meshconfig(rfpcbobject,mode)
meshconfig(shape,mode)
m = meshconfig( ___ ,mode)
```

Description

`meshconfig(rfpcbobject,mode)` changes the meshing mode of the PCB component according to the text input mode.

`meshconfig(shape,mode)` changes the meshing mode of the PCB shape according to the text input mode.

`m = meshconfig( ___ ,mode)` returns the mesh structure after changing the meshing mode of the PCB component or shape according to the text input mode.

Examples

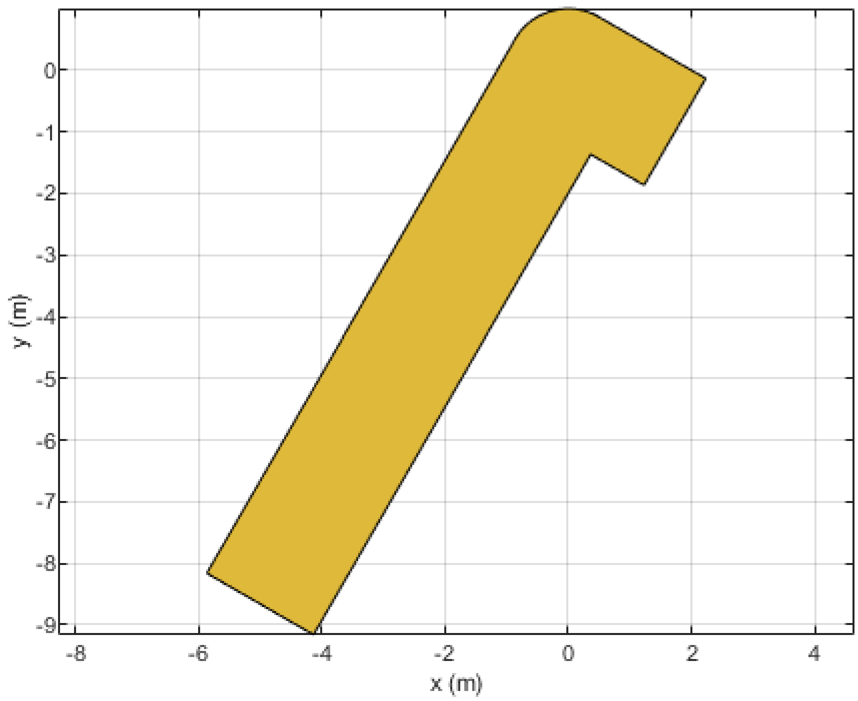
Mesh Rotated Curved Bend Shape

Create a curved bend shape of lengths of 10 m and 2 m, width of 2 m, and rotate it about the Z-axis by 60 degrees.

```
bend = bendCurved(Length=[10 2],Width=[2 2],CurveRadius=1)
```

```
bend = 
  bendCurved with properties:

             Name: 'myCurvedbend'
    ReferencePoint: [0 0]
           Length: [10 2]
            Width: [2 2]
      CurveRadius: 1
```

```
bend = rotateZ(bend,60);
show(bend)
```

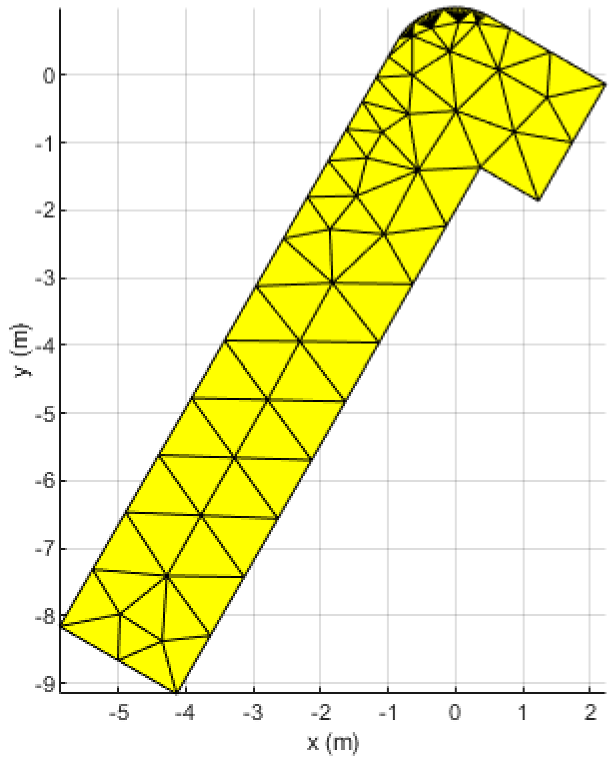


Mesh the curved bend shape at a maximum edge length of 1 m.

```
meshconfig(bend,"manual")
```

```
ans = struct with fields:
     NumTriangles: 0
    NumTetrahedra: 0
         NumBasis: []
    MaxEdgeLength: []
         MeshMode: 'manual'
```

```
mesh(bend,MaxEdgeLength=1)
```



Input Arguments

`rfpcbobject` — PCB component object
RF PCB object

PCB component object, specified as an RF PCB object. For a complete list of the PCB components, see "PCB Components Catalog".

`shape` — Shape created using custom elements and shape objects
object handle

Shape created using custom elements and shape objects, specified as an object handle. For a complete list of the custom shapes, see "Custom Geometry and PCB Fabrication".

Example: `c = bendCurved; mesh(c)`

`mode` — Meshing mode
`'auto'` (default) | `'manual'`

Meshing mode, specified as `'auto'` or `'manual'`.

Data Types: `char`

See Also

`mesh`

Introduced in R2021b

info

Display information about PCB component structure

Syntax

`info(rfpcbobject)`

Description

`info(rfpcbobject)` displays information about the PCB component. as a structure:

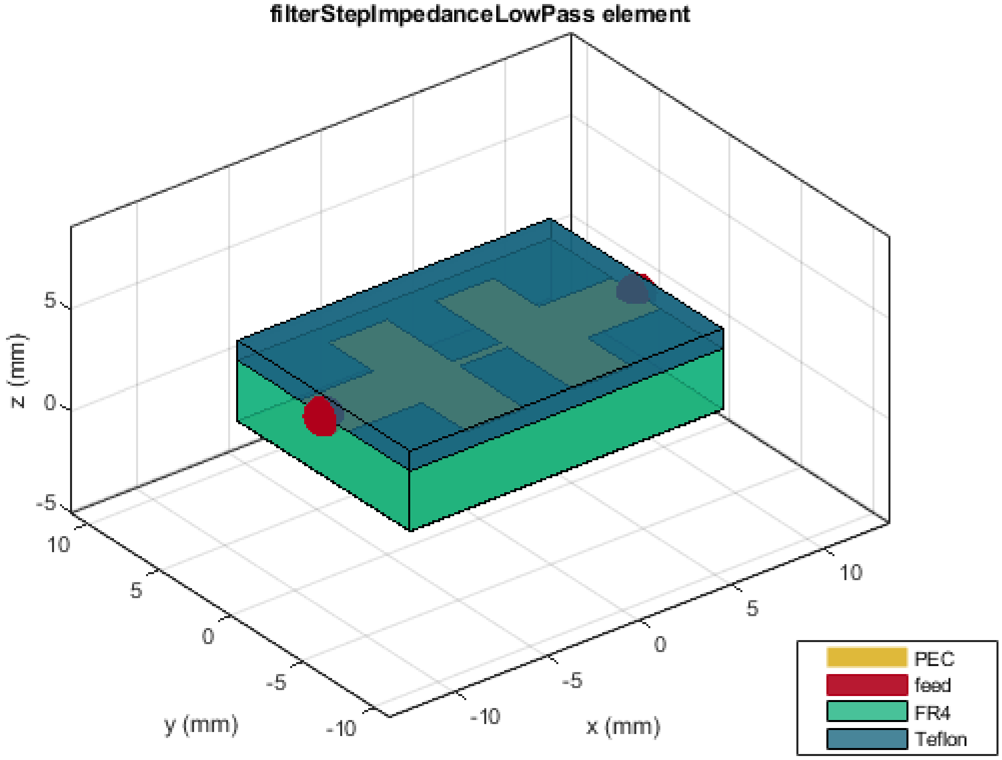
- `isSolved` – Logical specifying if an PCB component is solved.
- `isMeshed` – Logical specifying if an PCB component is meshed.
- `MeshingMode` – String specifying the meshing mode.
- `HasSubstrate` – Logical specifying if an PCB component uses a substrate.
- `HasLoad` – Logical specifying if an a PCB component has a load
- `PortFrequency` – Scalar or vector of frequencies used for port analysis.
- `FieldFrequency` – Scalar or vector of frequencies used for field analysis.
- `MemoryEstimate` – Approximate memory requirement for solving the antenna.

Examples

Create Stepped Impedance Lowpass Filter with Multilayer Dielectric Substrate

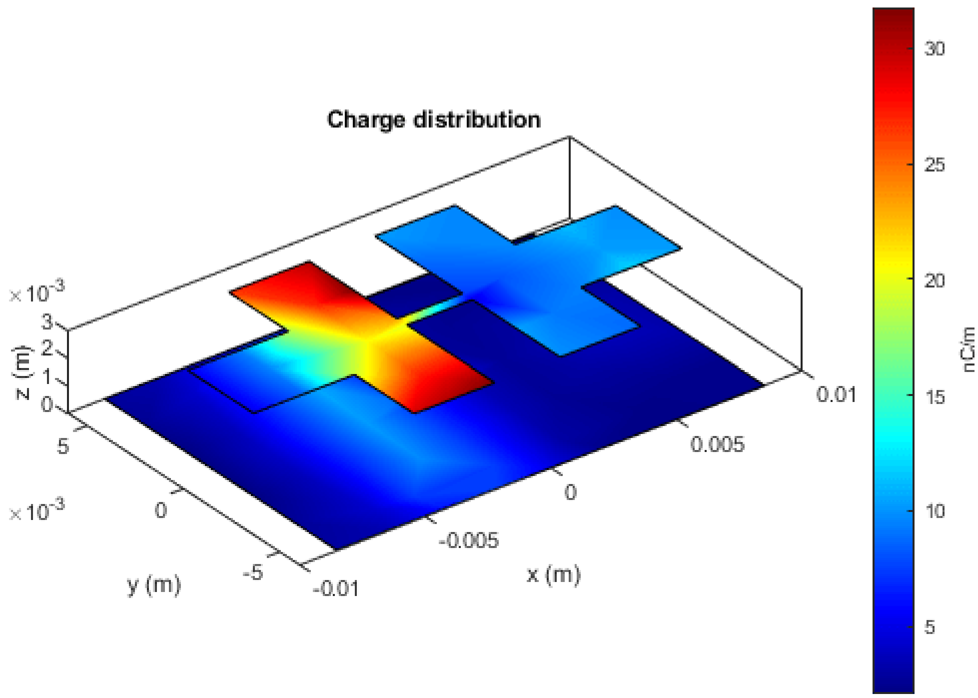
Create and view a stepped impedance lowpass filter with a multilayer dielectric substrate.

```
sub = dielectric("FR4","Teflon");
sub.Thickness =[0.003 0.001];
steppedfilter = filterStepImpedanceLowPass;
steppedfilter.Height = 0.003;
steppedfilter.Substrate = sub;
figure
show(steppedfilter)
```

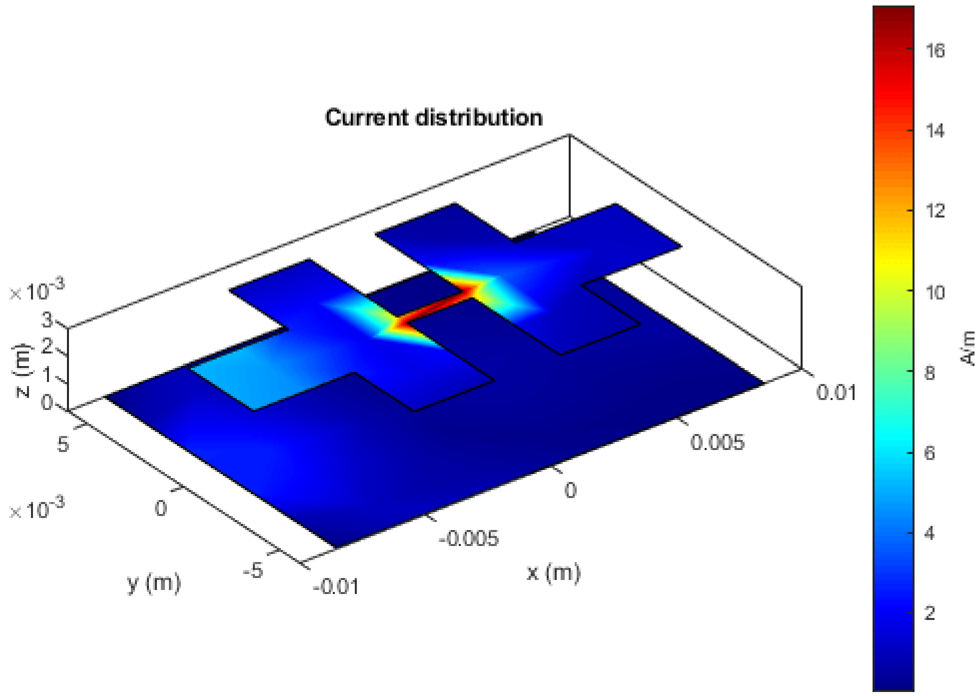


Plot the charge and current on the filter at 5 GHz.

```
figure
charge(steppedfilter,5e9)
```




```
figure
current(steppedfilter,5e9)
```




```
info(steppedfilter)
```

```
ans = struct with fields:
          IsSolved: "true"
          IsMeshed: "true"
       MeshingMode: "auto"
      HasSubstrate: "true"
           HasLoad: "false"
     PortFrequency: []
    MemoryEstimate: "790 MB"
```

Input Arguments

`rfpcbobject` — PCB component object
RF PCB object

PCB component object, specified as an RF PCB object. For a complete list of the PCB components, see "PCB Components Catalog".

See Also

`show`

Introduced in R2021b

cylinder2strip

Cylinder equivalent width approximation

Syntax

```
w = cylinder2strip(r)
```

Description

`w = cylinder2strip(r)` calculates the equivalent width of a strip approximation for a cylindrical cross section.

Examples

Calculate Cylinder to Strip Approximation

Calculate the width of the strip approximation to a cylinder of radius 20 mm.

```
w = cylinder2strip(20e-3)
```

```
w = 0.0800
```

Input Arguments

r — Cylindrical cross-section radius
scalar | vector

Cylindrical cross-section radius, specified as a scalar or vector in meters.

Example: 20e-3

Data Types: `double`

Output Arguments

w — Equivalent width of strip
scalar | vector

Equivalent width of strip, returned as a scalar or vector.

Data Types: `double`

See Also

Introduced in R2021b

voltagePort

Create voltage source with N-ports

Syntax

```
v = voltagePort(N)
v = voltagePort( ___ ,Name=Value)
```

Description

`v = voltagePort(N)` creates a voltage port source with *N* number of ports that you can use for excitation in an N-port PCB component.

`v = voltagePort( ___ ,Name=Value)` creates a voltage port source using additional name-value arguments.

Examples

Create Voltage Source

Create a voltage source with three ports.

```
v = voltagePort(3)
```

```
v = 
  voltagePort with properties:

         NumPorts: 3
      FeedVoltage: [1 0 0]
        FeedPhase: [0 0 0]
    PortImpedance: 50
```

Input Arguments

`N` — Number of ports
positive scalar

Number of ports, specified as a positive scalar.

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `FeedVoltage=1`

`NumPorts` — Number of ports
positive scalar

Number of ports, specified as a positive scalar.

`FeedVoltage` — Magnitude of voltage applied at each port
positive scalar | vector

Magnitude of the voltage applied at each port, specified as a positive scalar or vector.

`FeedPhase` — Phase shift applied to voltage at each port
positive scalar | vector

Phase shift applied to the voltage at each port in degrees, specified as a positive scalar or vector.

`PortImpedance` — Impedance to terminate each port
positive scalar | vector

Impedance to terminate each port in ohms, specified as a positive scalar or vector.

See Also
`current` | `feedCurrent` | `charge`

Introduced in R2021b

design

Design microstrip transmission line around specified frequency

Syntax

```
mline = design(mlineobj,frequency)
mline = design( ___ ,Name=Value)
```

Description

`mline = design(mlineobj,frequency)` designs a microstrip transmission line around the specified frequency.

`mline = design( ___ ,Name=Value)` designs a microstrip transmission line with additional options specified using name-value arguments.

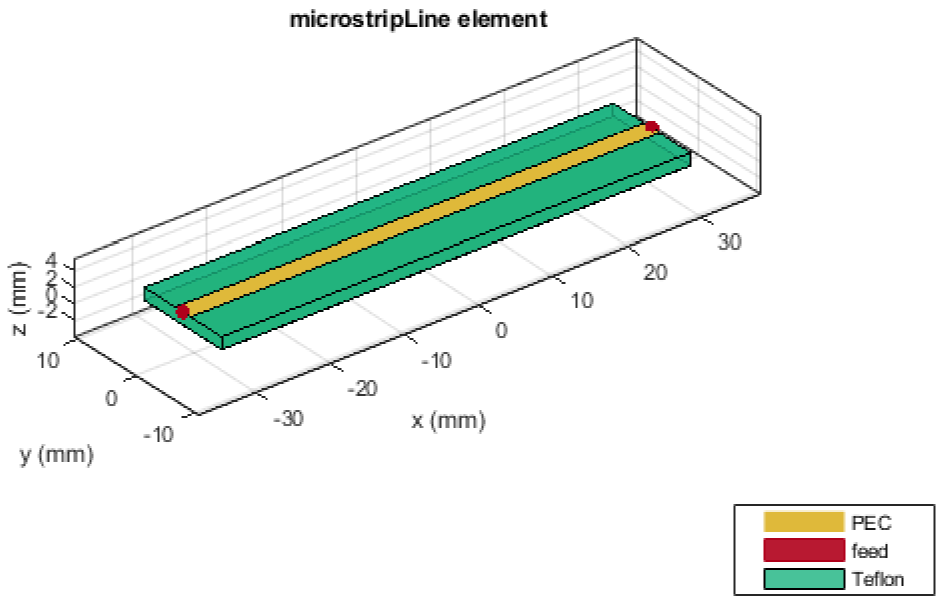
Note PCB components designed using the `design` function operate around the specified frequency with a 10-15% tolerance.

Examples

Design Microstrip Transmission Line Around 1.8 GHz

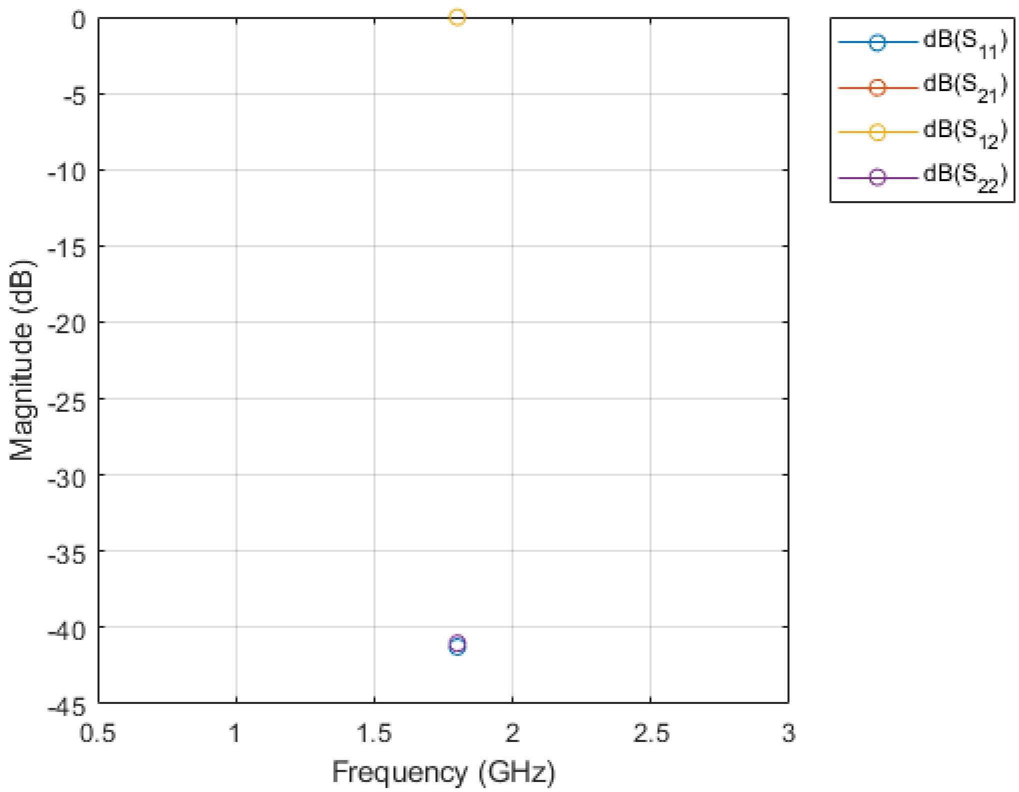
Design a microstrip line around 1.8 GHz and with a characteristic impedance of 75 ohms.

```
mline = design(microstripLine,1.8e9,Z0=75);
figure;
show(mline);
```



Plot the S-parameters of this line.

```
spar = sparameters(mline,1.8e9);
rfplot(spar)
```



Input Arguments

`mlineobj` — Microstrip transmission line

`microstripLine` object

Microstrip transmission line, specified as a `microstripLine` object.

Example: `mline = microstripLine; design(mline,2e9)` designs a microstrip transmission line around a frequency of 2 GHz.

`frequency` — Design frequency of transmission line

real positive scalar

Design frequency of the transmission line, specified as a real positive scalar in hertz.

Example: `55e6`

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `Z0=70`

Z0 — Characteristic impedance of microstrip transmission line
50 (default) | positive scalar

Characteristic impedance of the microstrip transmission line, specified as a positive scalar in ohms.

Data Types: double

LineLength — Length of line
0.5 (default) | positive scalar

Length of the line, specified as a positive scalar in Lambda.

Data Types: double

Output Arguments

mline — Microstrip transmission line operating around specified frequency
microstripLine object

Microstrip transmission line operating around the specified frequency, returned as a microstripLine object.

See Also

sparameters

Introduced in R2021b

design

Design symmetric strip transmission line around given frequency

Syntax

```
sline = design(slineobj,frequency)
sline = design( ___ ,Name,Value)
```

Description

`sline = design(slineobj,frequency)` designs an strip transmission line around the specified frequency with default 50 ohm reference impedance and `0.25` line length.

`sline = design( ___ ,Name,Value)` designs a strip transmission line with additional options specified by name-value pair arguments.

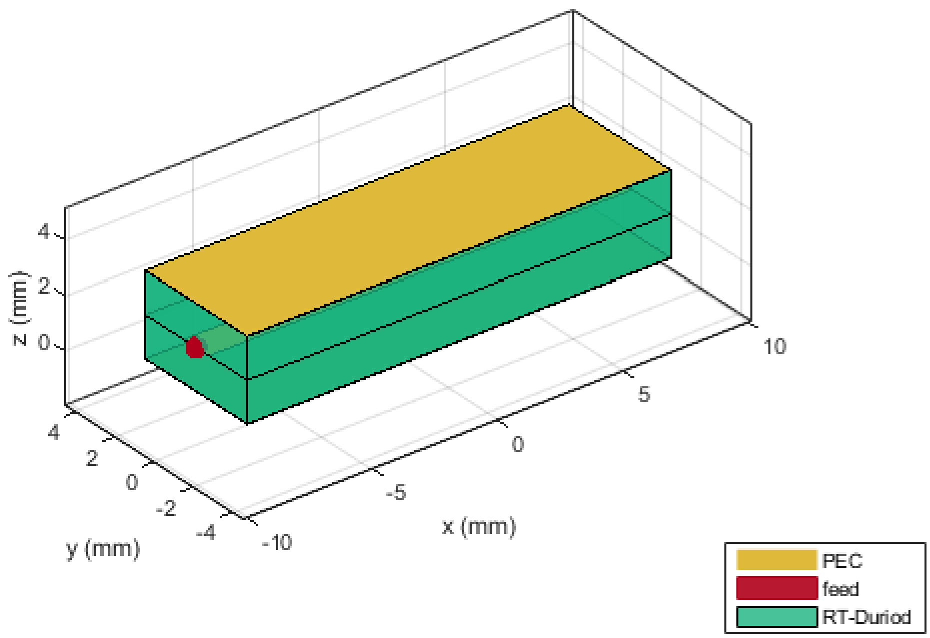
Note PCB components designed using the `design` function resonates around the specified frequency with a 10 to 15% tolerance.

Examples

Design Strip Transmission Line Around 3 GHz

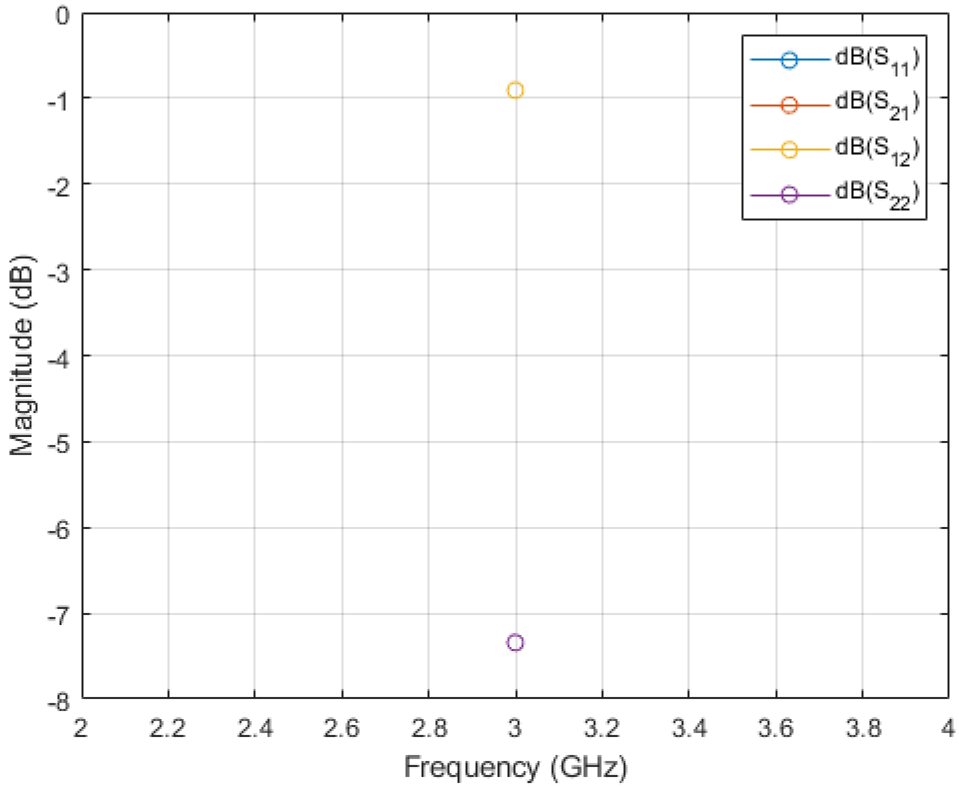
Design a strip transmission line around 3 GHz and with a characteristic impedance of 75 ohms.

```
sline = design(stripLine, 3e9,'Z0',75);
figure;
show(sline);
```



Plot the s-parameters of this transmission line.

```
spar = sparameters(sline,3e9);
rfplot(spar)
```



Input Arguments

`slineobj` — Strip transmission line
`stripLine` object

Strip transmission line, specified as a `stripLine` object.

Example: `sline = stripLine; design(sline,2e9)` designs a microstrip transmission line around a frequency of 2 GHz.

`frequency` — Resonant frequency of transmission line
real positive scalar

Resonant frequency of the transmission line, specified as a real positive scalar.

Example: `55e6`

Data Types: `double`

Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside quotes. You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

Example: `'Z0',2`

`Z0` — Characteristic impedance of line
`50` (default) | positive scalar

Characteristic impedance of the line in ohms, specified as the comma-separated pair consisting of `'Z0'` and a positive scalar.

Data Types: `double`

`LineLength` — Length of line
`0.25` (default) | positive scalar

Length of the in terms of Lambda, specified as the comma-separated pair consisting of `'LineLength'` and a positive scalar.

Data Types: `double`

Output Arguments

`sline` — Strip transmission line operating around specified reference frequency
`stripLine` object

Strip transmission line operating around the specified reference frequency, returned as a `stripLine` object.

See Also

`show`

Introduced in R2016b

design

Design coplanar waveguide transmission line around particular frequency

Syntax

```
waveguide = design(cpw,frequency)
waveguide = design( ___ ,Name=Value)
```

Description

`waveguide = design(cpw,frequency)` designs an coplanar waveguide line around the specified frequency.

`waveguide = design( ___ ,Name=Value)` designs a coplanar waveguide with additional options specified using name-value arguments.

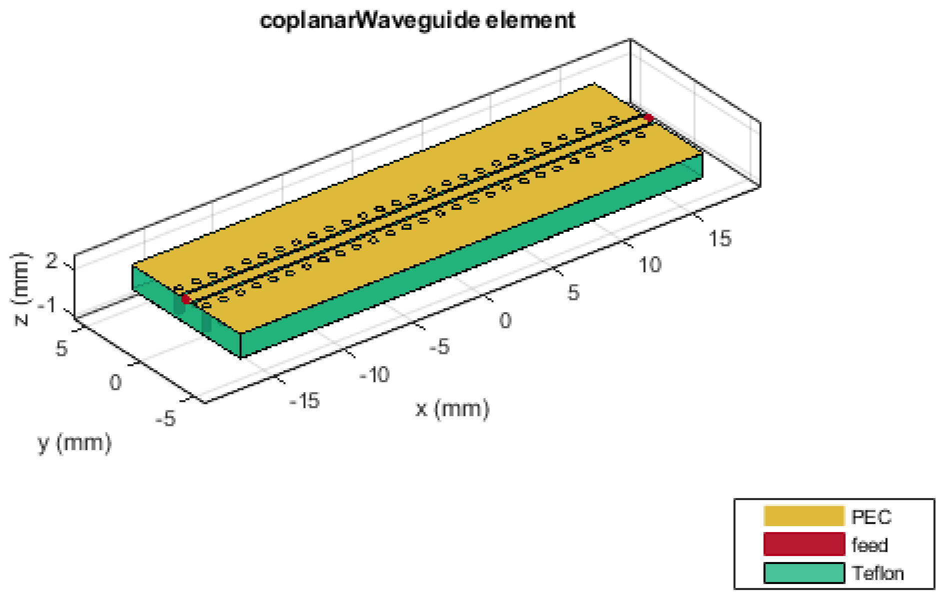
Note PCB components designed using the `design` function operates around the specified frequency with a 10-15% tolerance.

Examples

Design Coplanar Waveguide Around 1.8 GHz

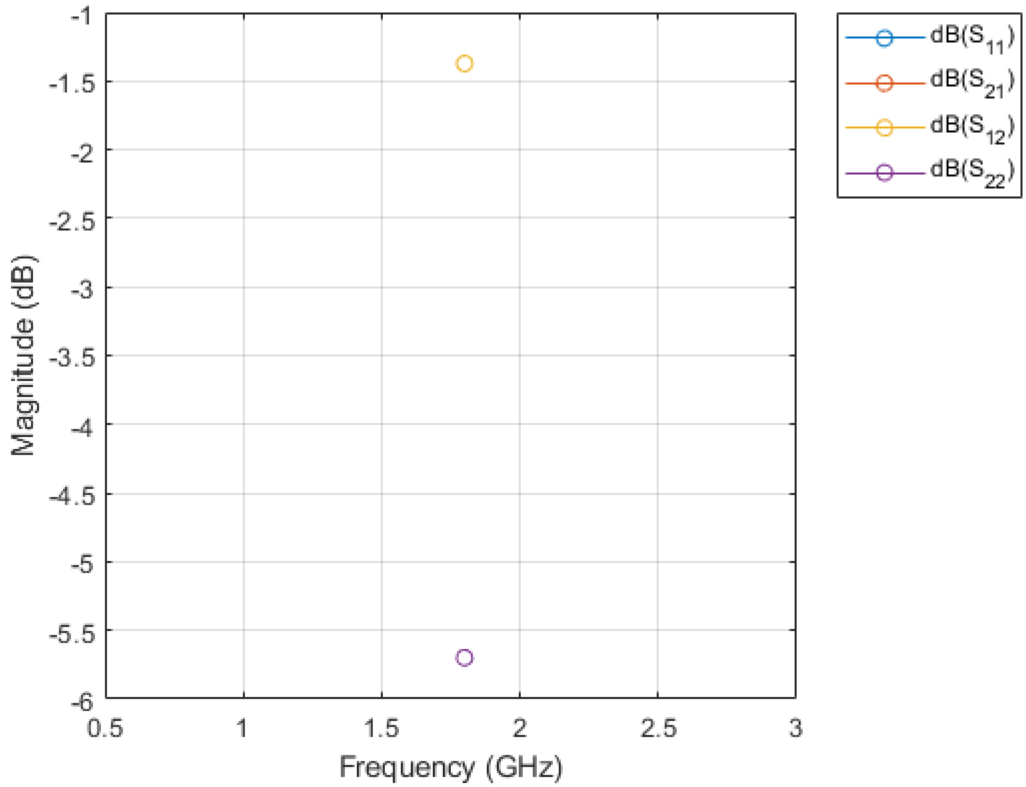
Design a lambda/4 coplanar waveguide at 1.8 GHz with a characteristic impedance of 75 ohms.

```
cpw = design(coplanarWaveguide, 1.8e9,Z0=75,LineLength=0.25);
figure;
show(cpw);
```



Plot the s-parameters of the waveguide at 1.8 GHz.

```
spar = sparameters(cpw,1.8e9);
rfplot(spar)
```



Input Arguments

`cpw` — Coplanar waveguide
`coplanarWaveguide` object

Coplanar waveguide transmission line, specified as a `coplanarWaveguide` object.

Example: `waveguide = coplanarWaveguide; waveguide = design(waveguide,2e9)` designs a coplanar waveguide around a frequency of 2 GHz.

`frequency` — Design frequency of transmission line
real positive scalar

Design frequency of the transmission line, specified as a real positive scalar in hertz.

Example: `3e9`

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `Z0=75`

Z0 — Characteristic impedance of line
50 (default) | positive scalar

Characteristic impedance of the line, specified as a positive scalar in ohms.

Data Types: double

LineLength — Length of coplanar waveguide
0.5 (default) | positive scalar

Length of the coplanar waveguide, specified as a positive scalar in terms of Lambda.

Data Types: double

Output Arguments

waveguide — Coplanar waveguide operating around specified reference frequency
coplanarWaveguide object

Coplanar waveguide operating around the specified reference frequency, returned as a coplanarWaveguide object.

See Also
sparameters

Introduced in R2021b

design

Design coupled microstrip transmission line around particular frequency

Syntax

```
cline = design(clineobj,frequency)
cline = design( ___ ,Name=Value)
```

Description

`cline = design(clineobj,frequency)` designs a coupled microstrip transmission line around the specified frequency.

`cline = design( ___ ,Name=Value)` designs a coupled microstrip transmission line with additional options specified using name-value arguments.

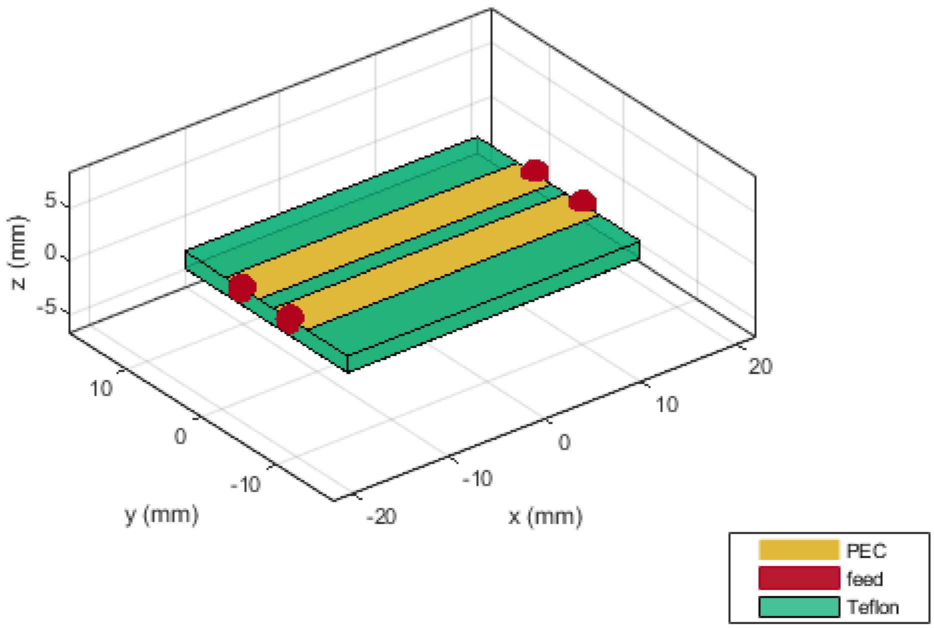
Note PCB components designed using the `design` function operate around the specified frequency with a 10-15% tolerance.

Examples

Design Coupled Microstrip Transmission Line Around 1.8 GHz

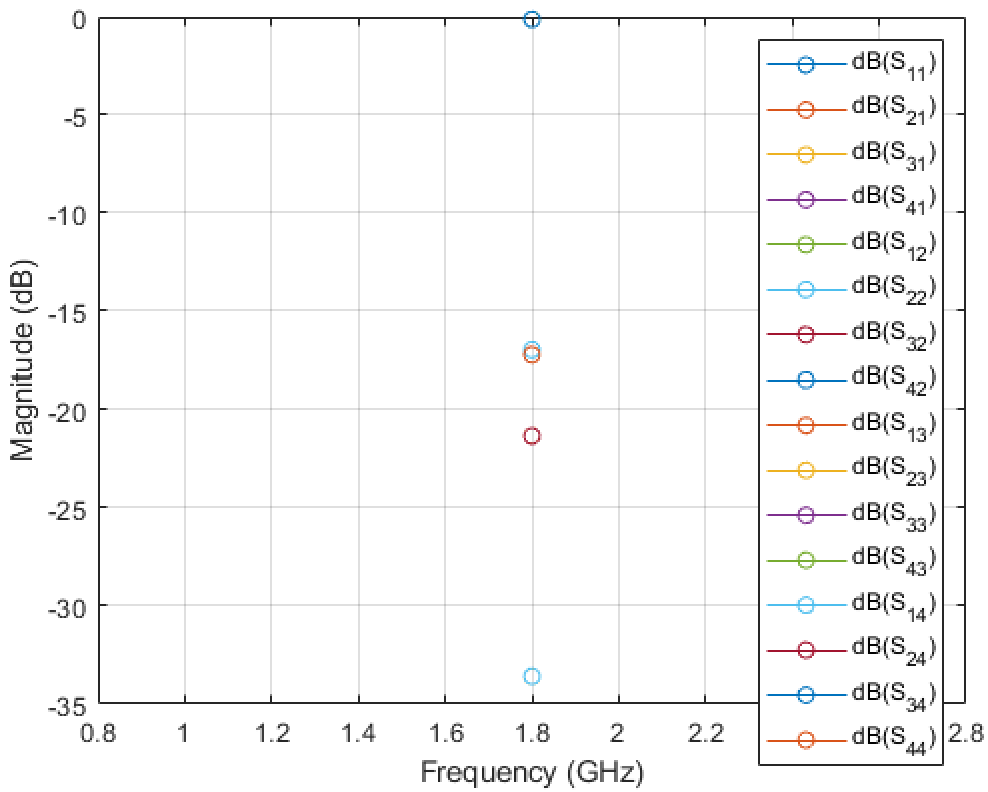
Design a coupled microstrip line at 1.8 GHz with an even mode impedance of 60 ohms.

```
cline = design(coupledMicrostripLine,1.8e9,Z0e=60);
show(cline);
```



Plot the S-parameters of the transmission line.

```
spar = sparameters(cline,1.8e9);
rfplot(spar)
```



Input Arguments

`clineobj` — Coupled microstrip transmission line

`coupledMicrostripLine` object

Coupled microstrip transmission line, specified as a `coupledMicrostripLine` object.

Example: `cline = coupledMicrostripLine; design(cline,2e9)` designs a coupled microstrip transmission line around a frequency of 2 GHz.

`frequency` — Design frequency of transmission line

real positive scalar

Design frequency of the transmission line, specified as a real positive scalar in hertz.

Example: `2e9`

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `Z0e=45`

Z0e — Even mode impedance
52 (default) | positive scalar

Even mode impedance of the transmission line, specified as a positive scalar in ohms.

Data Types: double

Z0o — Odd mode impedance
48 (default) | positive scalar

Odd mode impedance of the transmission line, specified as a positive scalar in ohms.

Data Types: double

Output Arguments

cline — Coupled microstrip transmission line operating around specified frequency
coupledMicrostripLine object

Coupled microstrip transmission line operating around the specified frequency, returned as a coupledMicrostripLine object.

See Also
sparameters

Introduced in R2021b

design

Design symmetric coupled strip transmission line around given frequency

Syntax

```
csline = design(cslineobj,frequency)
csline = design( ___ ,Name,Value)
```

Description

`csline = design(cslineobj,frequency)` designs an coupled strip transmission line around the specified frequency with default even and odd impedances of `55.2` ohms and `45.09` ohms respectively using a λ-by-4 line length.

`csline = design( ___ ,Name,Value)` designs a coupled strip transmission line with additional options specified by name-value pair arguments.

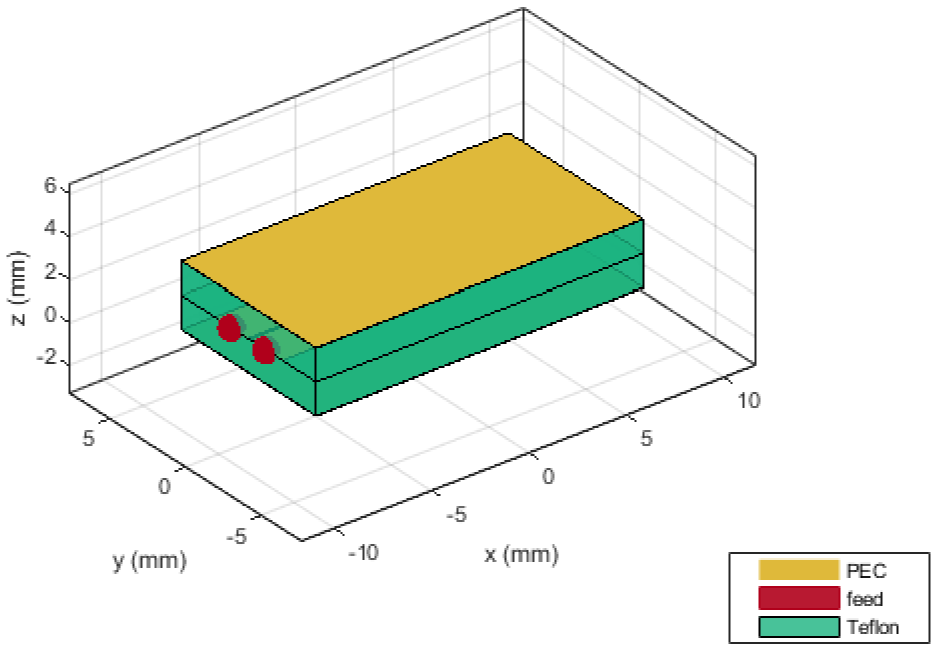
Note PCB components designed using the `design` function resonates around the specified frequency with a 10 to 15% tolerance.

Examples

Design Coupled Strip Transmission Line Around 3 GHz.

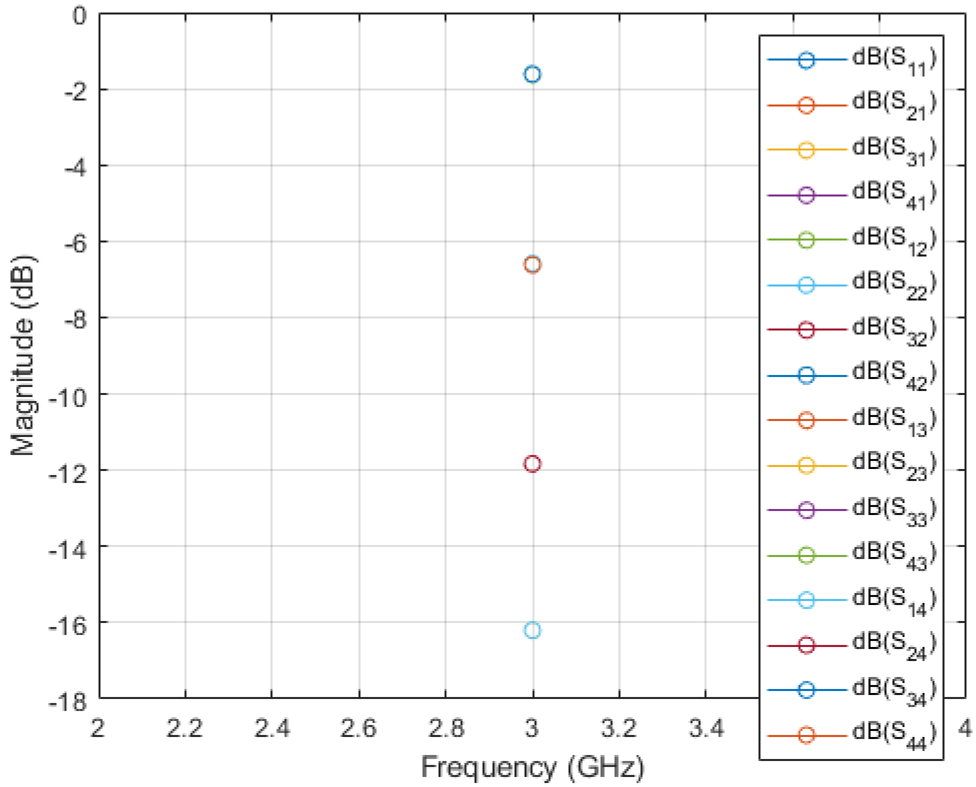
Design a coupled strip transmission line at 3 GHz for an even and odd mode impedance of 70 ohms and 35 ohms respectively and visualize it.

```
csline = design(coupledStripLine,3e9,'Z0e',70,'Z0o',35);
figure; show(csline);
```



Plot the s-parameters of the transmission line at 3 GHz.

```
spar = sparameters(csline,3e9);
rfplot(spar)
```



Input Arguments

`cslineobj` — Coupled strip transmission line
`coupledStripLine` object

Coupled strip transmission line, specified as a `coupledStripLine` object.

Example: `csline = coupledStripLine; design(csline,2e9)` designs a coupled strip transmission line around a frequency of 2 GHz.

`frequency` — Resonant frequency of transmission line
real positive scalar

Resonant frequency of the transmission line, specified as a real positive scalar.

Example: `55e6`

Data Types: `double`

Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside quotes. You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

Example: `'Z0e',53`

Z0e — Even mode impedance
55.2 (default) | positive scalar

Even mode impedance of the transmission line in ohms, specified as the comma-separated pair consisting of 'Z0e' and a positive scalar.

Data Types: double

Z0o — Odd mode impedance
45.09 (default) | positive scalar

Odd mode impedance of the transmission line in ohms, specified as the comma-separated pair consisting of 'Z0e' and a positive scalar.

Data Types: double

Output Arguments

csline — Strip transmission line operating around specified reference frequency
coupledStripLine object

Coupled strip transmission line operating around the specified reference frequency, returned as a coupledStripLine object.

See Also

show

Introduced in R2016b

design

Design branchline coupler around particular frequency

Syntax

```
coupler = design(couplerobj,frequency)
coupler = design( ___ ,Name,Value)
```

Description

`coupler = design(couplerobj,frequency)` designs a branchline coupler around the specified frequency.

`coupler = design( ___ ,Name,Value)` designs a branchline coupler line with additional options specified using name-value arguments.

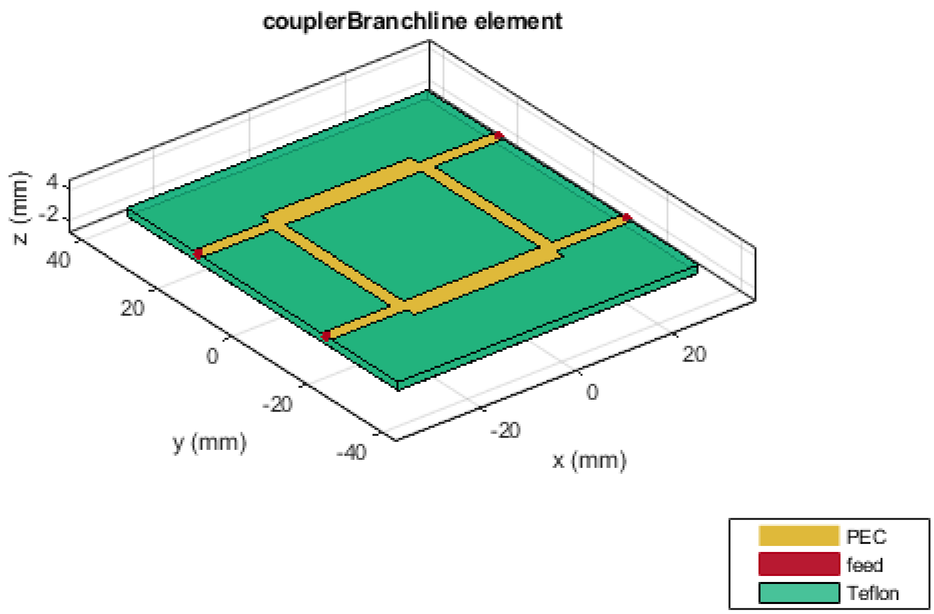
Note PCB components designed using the `design` function operate around the specified frequency with a 10-15% tolerance.

Examples

Design Branchline Coupler Around 1.8 GHz

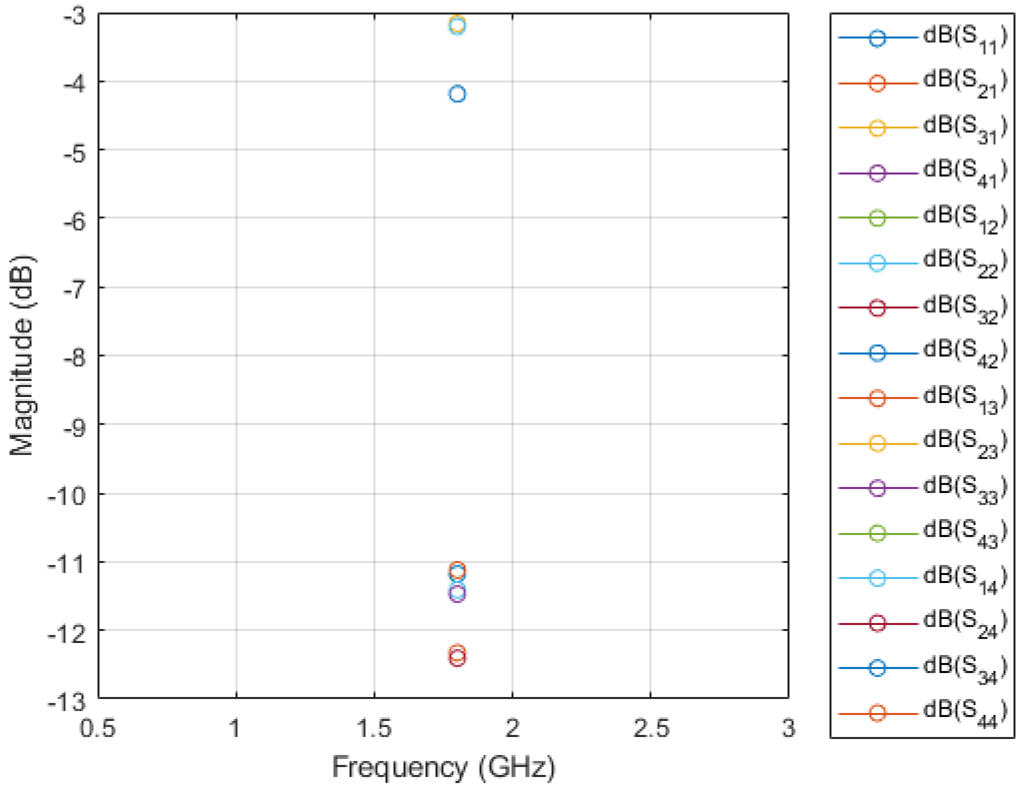
Design a branchline coupler at 1.8 GHz and with a Z0 of 75 ohms.

```
coupler = design(couplerBranchline,1.8e9,Z0=75);
show(coupler);
```



Plot the s-parameters of the coupler at 1.8 GHz.

```
spar = sparameters(coupler,1.8e9);
rfplot(spar)
```



Input Arguments

`couplerobj` — Branchline coupler
`couplerBranchline` object

Branchline coupler, specified as a `couplerBranchline` object.

Example: `coupler = couplerBranchline; design(coupler,2e9)` designs a branchline coupler around a frequency of 2 GHz.

`frequency` — Design frequency of coupler
real positive scalar

Design frequency of the coupler, specified as a real positive scalar in hertz.

Example: `3e9`

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `Z0=70`

Z0 — Characteristic impedance of coupler
50 (default) | positive scalar

Characteristic impedance of the coupler, specified as a positive scalar in ohms.

Data Types: double

Output Arguments

coupler — Branchline coupler operating around specified frequency
couplreBranchline object

Branchline coupler operating around the specified frequency, returned as a couplreBranchline object.

See Also

sparameters

Introduced in R2021b

design

Design rat-race coupler around specified frequency

Syntax

```
coupler = design(couplerobj,frequency)
coupler = design( ___ ,Name=Value)
```

Description

`coupler = design(couplerobj,frequency)` designs a rat-race coupler around the specified frequency.

`coupler = design( ___ ,Name=Value)` designs a rat-race coupler line with additional options specified using name-value arguments.

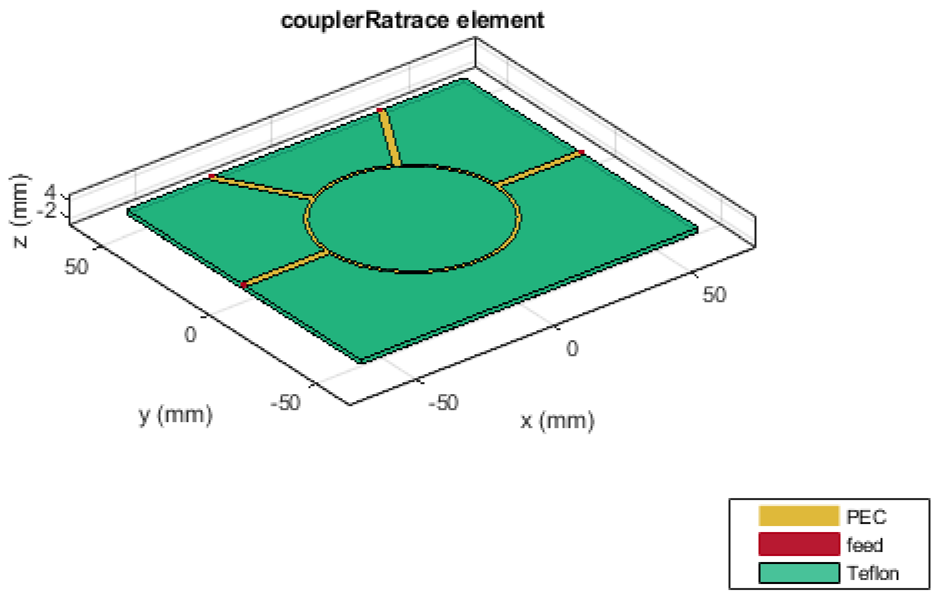
Note PCB components designed using the `design` function operate around the specified frequency with a 10-15% tolerance.

Examples

Design Rat-race Coupler Around 1.8 GHz

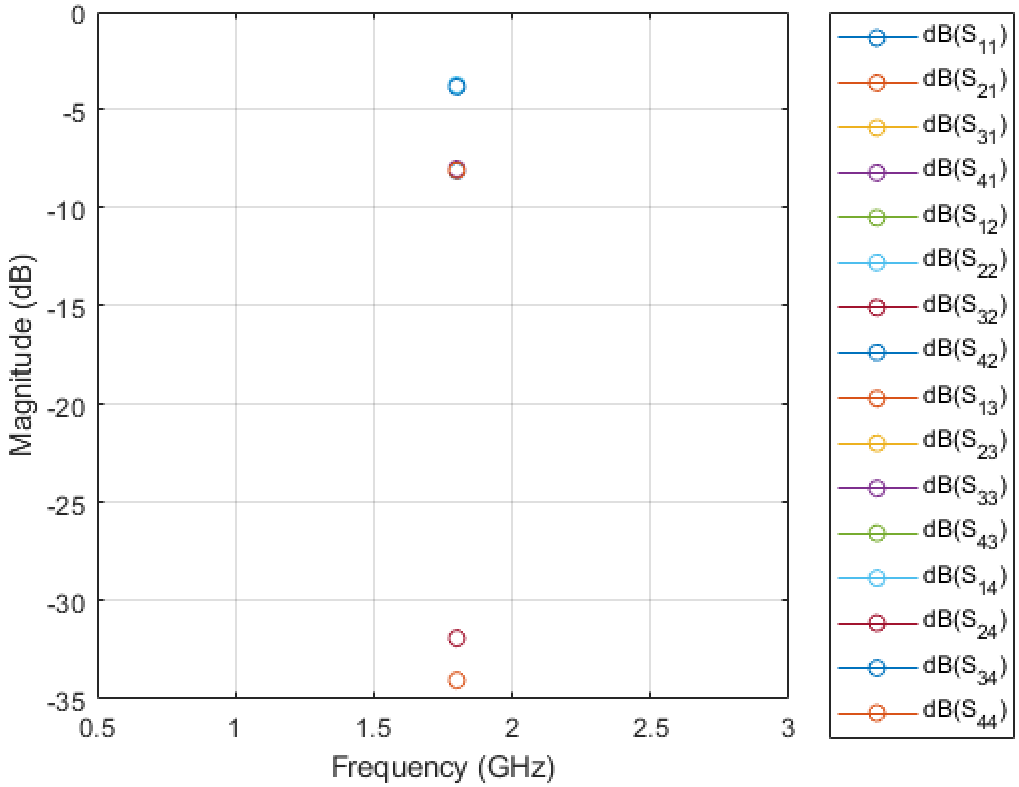
Design a rat-race coupler at 1.8 GHz and with a charecteristic impedance of 75 ohms.

```
coupler = design(couplerRatrace,1.8e9,Z0=75);
show(coupler);
```



Plot the S-parameters of the coupler at 1.8 GHz.

```
spar = sparameters(coupler,1.8e9);
rfplot(spar)
```



Input Arguments

`couplerobj` — Rat-race coupler
`couplerRatrace` object

Rat-race coupler, specified as a `couplerRatrace` object.

Example: `coupler = couplerRatrace; design(coupler,2e9)` designs a rat-race coupler around a frequency of 2 GHz.

`frequency` — Design frequency of rat-race coupler
real positive scalar

Design frequency of the rat-race coupler, specified as a real positive scalar in hertz.

Example: `2.5e9`

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `Z0=70`

`Z0` — Characteristic impedance of coupler
`50` (default) | positive scalar

Characteristic impedance of the coupler in ohms, specified as a positive scalar.

Data Types: `double`

Output Arguments

`coupler` — Rat-race coupler operating around specified reference frequency
`couplerRatrace` object

Rat-race coupler operating around the specified reference frequency, returned as a `couplerRatrace` object.

See Also

`sparameters`

Introduced in R2021b

sparameters

Calculate S-parameters for RF PCB objects

Syntax

```
sobj = sparameters(rfpcbobj,freq)
sobj = sparameters( ___ ,Z0)
sobj = sparameters( ___ ,Name=Value)

sobj = sparameters(data,freq)
sobj = sparameters(data,freq,Z0)

sobj = sparameters(filename)
```

Description

`sobj = sparameters(rfpcbobj,freq)` calculates the S-parameters for the RF PCB object `rfpcbobj` over the specified frequency values.

`sobj = sparameters( ___ ,Z0)` calculates the S-parameters for the reference impedance `Z0`.

`sobj = sparameters( ___ ,Name=Value)` calculates S-parameters using one or more name-value arguments in addition to any of the input argument combinations in previous syntaxes

`sobj = sparameters(data,freq)` creates an S-parameter object from the S-parameter data provided in `data` over the specified frequencies values.

`sobj = sparameters(data,freq,Z0)` creates an S-parameter object for the reference impedance `Z0`.

`sobj = sparameters(filename)` creates an S-parameter object from the data provided in the Touchstone file specified in `filename`.

Examples

Calculate S-parameters for Wilkinson Power Splitter

Create a Wilkinson power splitter object.

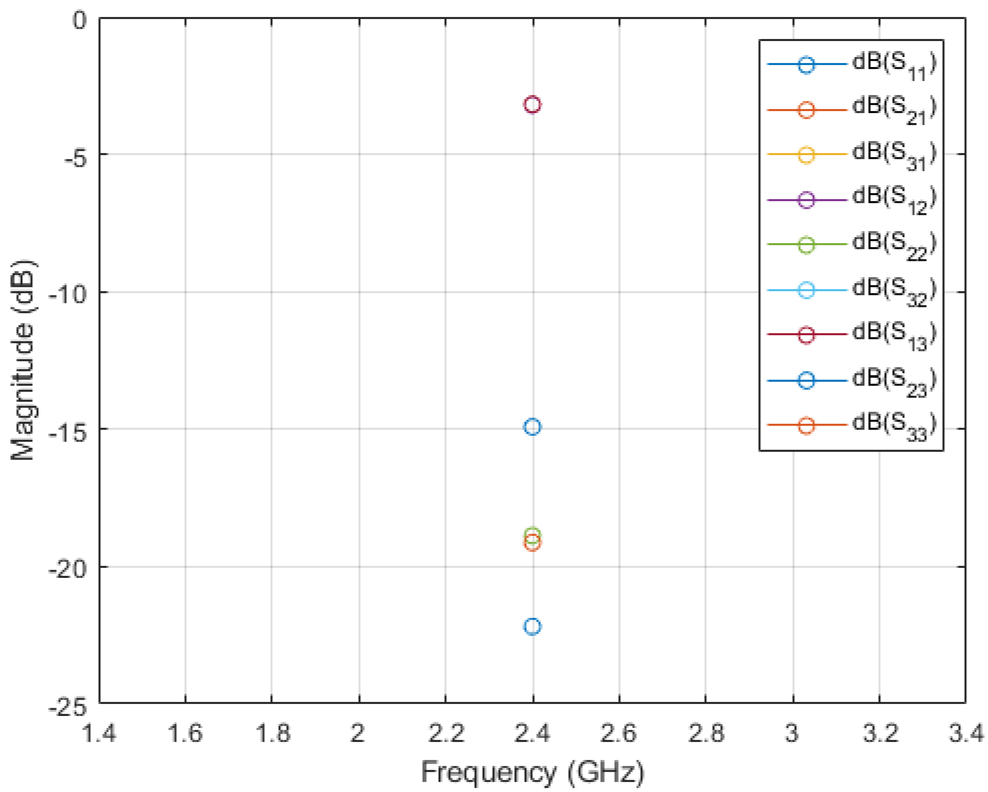
```
rfpcbobj = wilkinsonSplitter;
```

Calculate the S-parameters for the Wilkinson power splitter at 2.4 GHz with the reference impedance of 50 ohms.

```
Sobj = sparameters(wilkinsonSplitter,2.4e9,50);
```

Plot the S-parameters using the `rfplot` function.

```
rfplot(Sobj)
```



Calculate S-Parameters of Right Angle Bend

Design a microstrip transmission line at 3 GHz with 75 ohms impedance.

```
m = microstripLine(Length=0.0379,Width=0.0027,Height=0.0016,GroundPlaneWidth=0.0133);
```

Create a right angle bend with length equal to half the length of the transmission line and width equal to the width of the transmission line.

```
layer2d = bendRightAngle( Length=[m.Length/2 m.Length/2],...
Width=[m.Width m.Width]);
```

Convert the right angle bend to a 3-D component.

```
robj = pcbComponent(layer2d);
```

Add thickness and substrate layers to the board.

```
robj.BoardThickness = m.Substrate.Thickness;
robj.Layers{2} = m.Substrate;
```

Define frequency points to calculate the S-parameters.

```
freq = (1:2:40)*100e6;
```

Calculate the S-parameters of the right angle bend using the behavioral model.

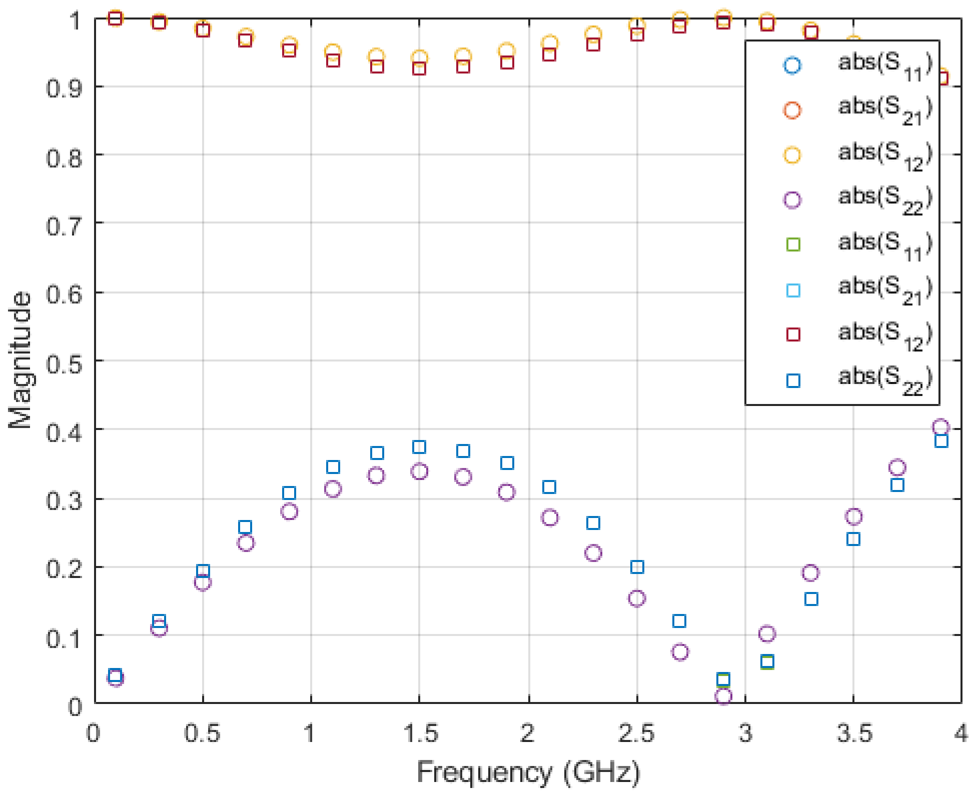
```
Sckt1 = sparameters(robj,freq,Behavioral=true);
```

Calculate the S-parameters of the right angle bend using the electromagnetic solver.

```
Sem1 = sparameters(robj,freq);
```

Plot the S-parameter data using the `rfplot` function.

```
rfplot(Sckt1,'abs','o')
hold on
rfplot(Sem1,'abs','s')
```



Input Arguments

`rfpcbobj` — Input object
RF PCB object

Input object, specified as a RF PCB object. You can specify either a PCB component, a microstrip bend, or a trace. For complete list of PCB components, microstrip bends, and traces, see "PCB Components Catalog" and "Custom Geometry and PCB Fabrication".

`data` — S-parameter data
array of complex numbers

S-parameter data, specified as an array of complex numbers of the size *N*-by-*N*-by-*K*, where *K* represents number of frequency points.

`freq` — S-parameter frequency
scalar | vector

S-parameter frequencies, specified as a scalar or vector of positive real numbers in the ascending order.

`Z0` — Reference impedance
`50` (default) | positive real scalar

Reference impedance in ohms, specified as a positive real scalar.

`filename` — Name of Touchstone file
character vector | string scalar

Name of the Touchstone file containing network parameter data, specified as a character vector or string scalar. If the file is in the current folder or in a folder on the MATLAB® path, specify the file name. If the file is not in the current folder or in a folder on the MATLAB path, then specify the full or relative path name.

Example: `sobj = sparameters('defaultbandpass.s2p');`

Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside quotes. You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

Example: `Sobj = sparameters(robj,freq,Behavioral=true)`

`Behavioral` — Behavioral model of RF PCB component and bend
`false or 0` (default) | `true or 1`

Behavioral model of an RF PCB component and bend, specified as `true` (`1`) or `false` (`0`). You can compute the behavioral model for these `rfpcb` objects:

Bends	• `bendRightAngle` • `bendCurved` • `bendMitered`
Traces	• `traceTee` • `traceCross`
Transmission line objects	• `microstripLine` • `coplanarWaveguide`
Inductor	`spiralInductor`
Capacitor	`interdigitalCapacitor`

Note

- Before using the `sparameters` function to calculate S-parameters for bends and traces, convert bends and traces to PCB components using the `pcbComponent` function.
- The `sparameters` function does not support using the behavioral model argument for:

- Objects with unequal widths like `bendRightAngle`, `bendCurved`, and `bendMitered`
- Asymmetric tee and cross-junction traces

Example: `Sobj = sparameters(microstripline,freq,Behavioral = true)`

Output Arguments

`sobj` — S-parameters
S-parameter object

S-parameters, returned as an object with the following properties:

- `NumPorts` — Number of ports, *N*, returned as an integer. The function calculates this value automatically when you create the object.
- `Frequencies` — S-parameter frequency, returned as a scalar or row vector of length, *K*, in the ascending order. The function sets this property from the `filename` or `freq` input arguments.
- `Parameters` — S-parameters, returned as an *N*-by-*N*-by-*K* array of complex numbers. The function sets this property from the `filename` or `data` input arguments.
- `Impedance` — Reference impedance in ohms, returned as a positive real scalar. The function sets this property from the `filename` or `Z0` input arguments. If you do not provide reference impedance,, the function uses a default value of `50`.

See Also

`current` | `getZ0`

Introduced in R2021b

layout

Plot all metal layers and board shape

Syntax

`layout(rfpcbobject)`

Description

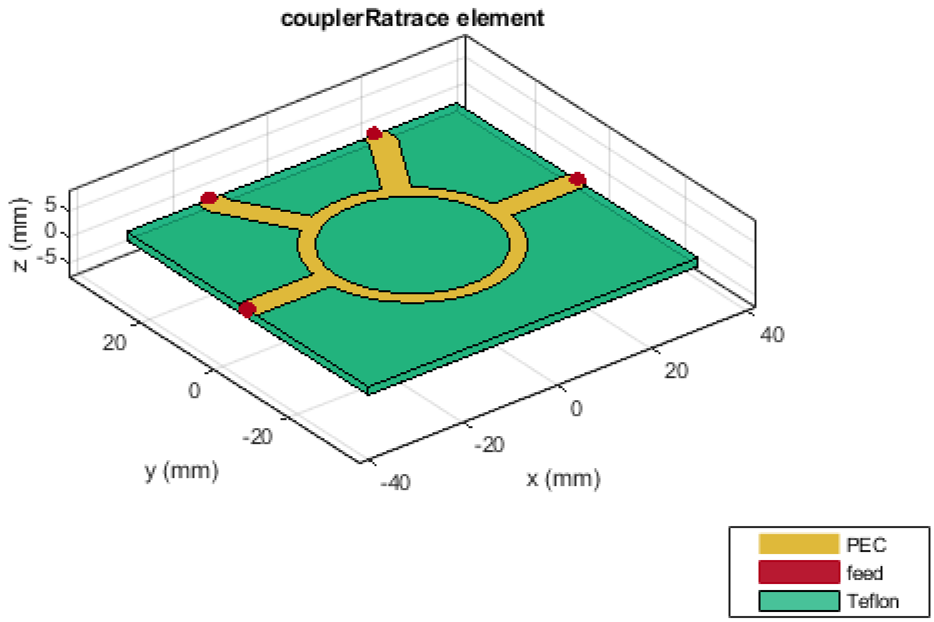
`layout(rfpcbobject)` displays all the metal layers and the PCB shape in the figure window. The red filled circle correspond to PCB feed points and the blue filled circles correspond to vias.

Examples

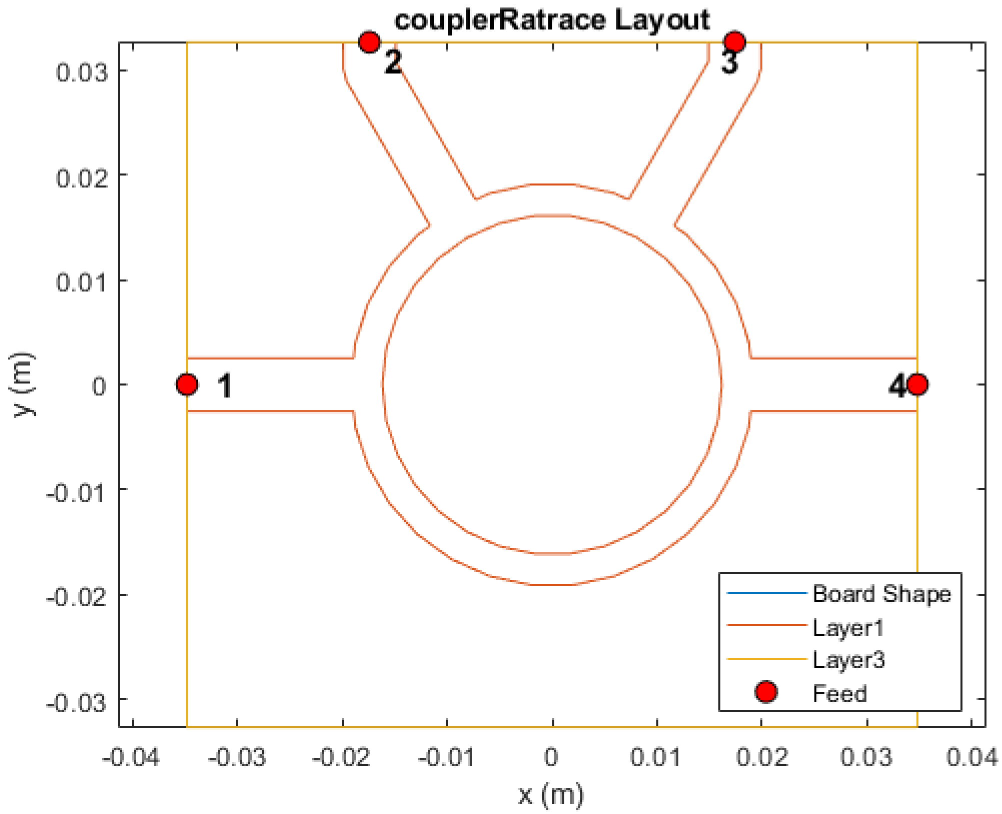
Display Ratrace Coupler Layout

Display the layout of a ratrace coupler.

```
coupler = couplerRatrace;
show(coupler)
```



```
layout(coupler)
```



Input Arguments

`rfpcbobject` — PCB component
object handle

PCB component, specified as an object handle. For complete list of PCB components and shapes, see "PCB Components Catalog"

See Also

`show` | `info`

Introduced in R2021b

shapes

Extract all metal layer shapes of PCB component

Syntax

```
shapes(rfpcbobject)
```

Description

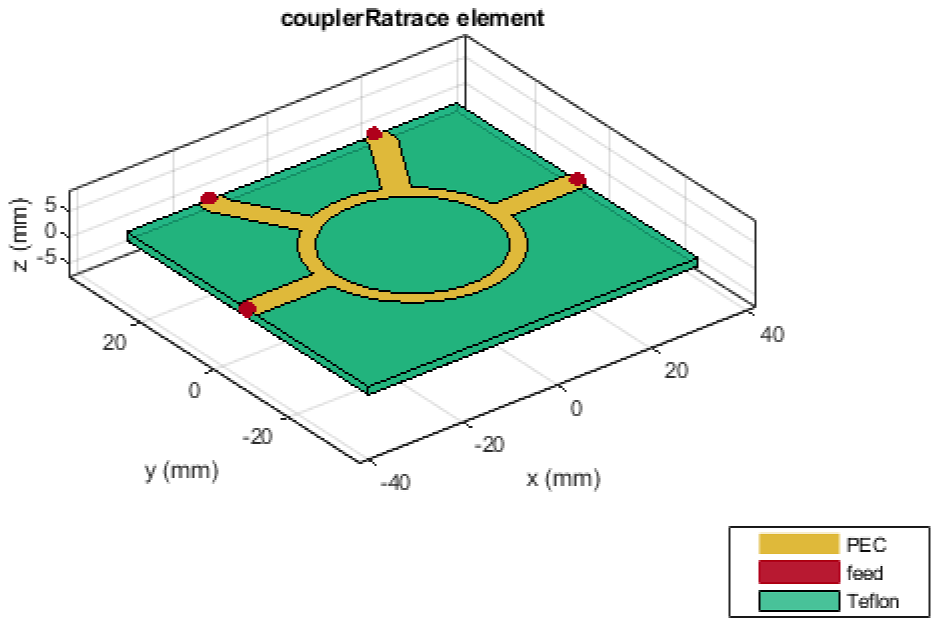
`shapes(rfpcbobject)` extracts all metal layer shapes of a PCB component and organizes them into an output structure.

Examples

Extract Ratrace Coupler Shapes

Extract the shapes of a ratrace coupler.

```
coupler = couplerRatrace;
show(coupler)
```



```
shapes(coupler)
```

```
ans = struct with fields:
    Layer1: [1×1 antenna.Polygon]
    Layer2: [1×1 antenna.Rectangle]
```

Input Arguments

`rfpcbobject` — PCB component object

RF PCB object

PCB component object, specified as an RF PCB object. For a complete list of the PCB components, see "PCB Components Catalog".

See Also

`layout` | `show`

Introduced in R2021b

design

Design Wilkinson splitter around specified frequency

Syntax

```
wsplitter = design(wsplitterobj,frequency)
wsplitter = design( ___ ,Name=Value)
```

Description

`wsplitter = design(wsplitterobj,frequency)` designs a Wilkinson splitter around the specified frequency.

`wsplitter = design( ___ ,Name=Value)` designs a Wilkinson splitter with additional options specified using name-value arguments.

Note PCB components designed using the `design` function operate around the specified frequency with a 10-15% tolerance.

Examples

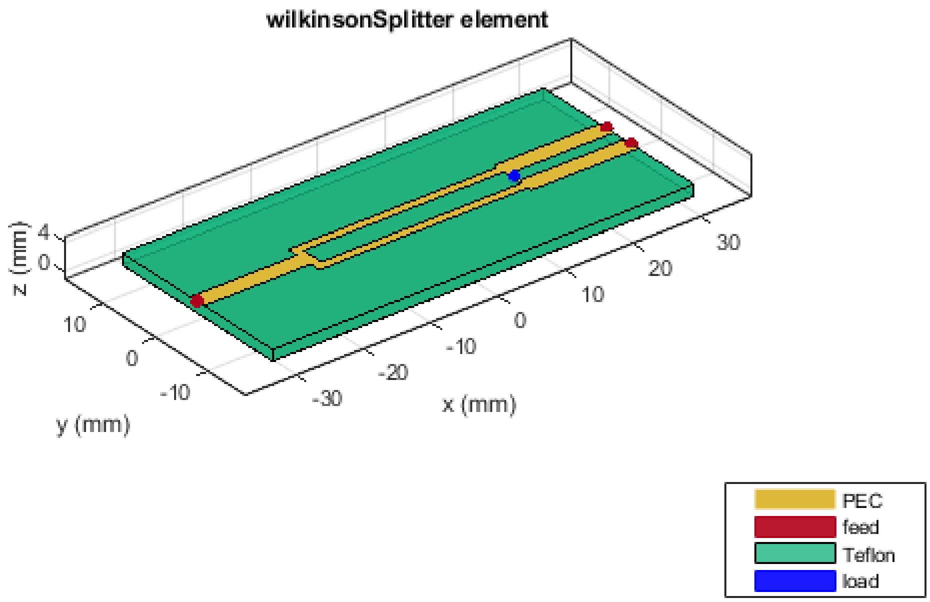
Design Wilkinson Splitter at 1.8 GHz

Design a Wilkinson splitter at 1.8 GHz and with a Z0 of 75 ohm.

```
wsplitter = design(wilkinsonSplitter,1.8e9,Z0=75);
```

View the splitter.

```
figure;
show(wsplitter);
```



Input Arguments

wsplitterobj — Wilkinson splitter
wilkinsonSplitter object

Wilkinson splitter, specified as a wilkinsonSplitter object.

Example: wsplitterobj = wilkinsonSplitter; design(wsplitterobj,2e9) designs a Wilkinson splitter around a frequency of 2 GHz.

frequency — Design frequency of Wilkinson splitter
real positive scalar

Design frequency of the Wilkinson splitter, specified as a real positive scalar in hertz.

Example: 55e6

Data Types: double

Name-Value Pair Arguments

Specify optional pairs of arguments as Name1=Value1,...,NameN=ValueN, where Name is the argument name and Value is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: Z0=2

Z0 — Characteristic impedance of splitter

50 (default) | positive scalar

Characteristic impedance of the splitter, specified as a positive scalar in ohms.

Data Types: double

Output Arguments

wsplitter — Wilkinson splitter operating around specified frequency

wilkinsonSplitter object

Wilkinson splitter operating around the specified frequency, returned as a wilkinsonSplitter object.

See Also

sparameters

Introduced in R2021b

design

Design unequal Wilkinson splitter around specified frequency

Syntax

```
uwsplitter = design(uwsplitterobj,frequency)
uwsplitter = design( ___ ,Name=Value)
```

Description

`uwsplitter = design(uwsplitterobj,frequency)` designs a unequal Wilkinson splitter around the specified frequency.

`uwsplitter = design( ___ ,Name=Value)` designs a unequal Wilkinson splitter with additional options specified using name-value arguments.

Note PCB components designed using the `design` function operate around the specified frequency with a 10-15% tolerance.

Examples

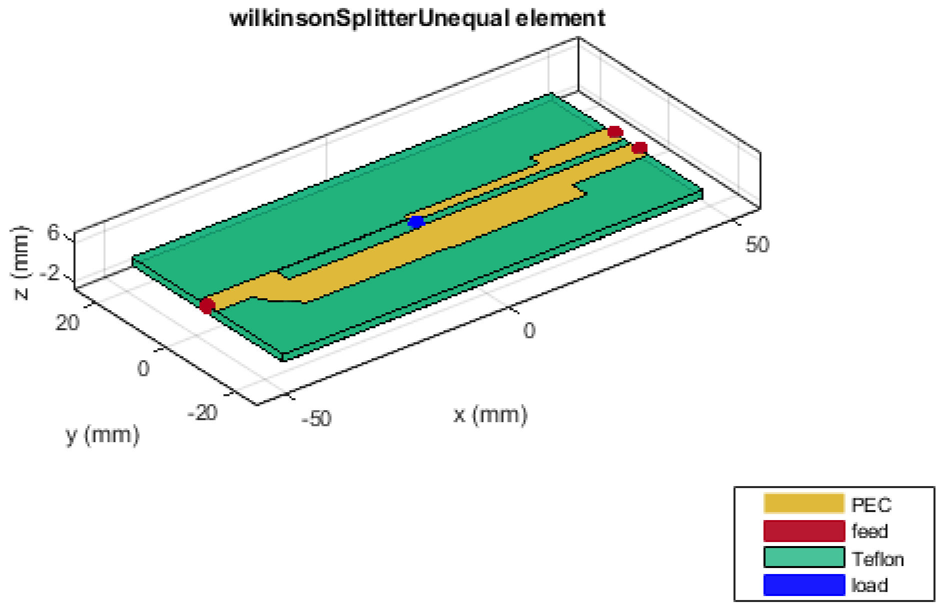
Design Unequal Wilkinson Splitter at 1.8 GHZ

Design an unequal wilkinson at 1.8 GHz with a characteristic impedance of 50 ohms and a PowerRatio of 6.

```
uwilk = design(wilkinsonSplitterUnequal,1.8e9,Z0=50,PowerRatio=6);
```

View the splitter

```
show(uwilk);
```



Input Arguments

`uwsplitterobj` — Unequal Wilkinson splitter
`wilkinsonSplitterUnequal` object

Unequal Wilkinson splitter, specified as a `wilkinsonSplitterUnequal` object.

Example: `uwsplitterobj = wilkinsonSplitterUnequal; design(wsplitterobj,2e9)` designs a unequal Wilkinson splitter around a frequency of 2 GHz.

`frequency` — Design frequency of unequal Wilkinson splitter
real positive scalar

Design frequency of the unequal Wilkinson splitter, specified as a real positive scalar in hertz.

Example: `2.5e9`

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `Z0=75`

`Z0` — Characteristic impedance of splitter
`50` (default) | positive scalar

Characteristic impedance of the splitter, specified as a positive scalar in ohms.

Data Types: `double`

`PowerRatio` — Power division ratio between two output ports of splitter
`2` (default) | positive scalar

Power division ratio between the two output ports of the splitter, specified as a positive scalar.

Data Types: `double`

Output Arguments

`uwsplitter` — Unequal Wilkinson splitter operating around specified frequency
`wilkinsonSplitterUnequal` object

Unequal Wilkinson splitter operating around the specified frequency, returned as a `wilkinsonSplitterUnequal` object.

See Also

Introduced in R2021b

design

Design coupled line filter around specified frequency

Syntax

```
clfilter = design(clfilterobj,frequency)
clfilter = design( ___ ,Name,Value)
```

Description

`clfilter = design(clfilterobj,frequency)` designs a coupled line filter around the specified frequency.

`clfilter = design( ___ ,Name,Value)` designs a unequal Wilkinson splitter with additional options specified using name-value arguments.

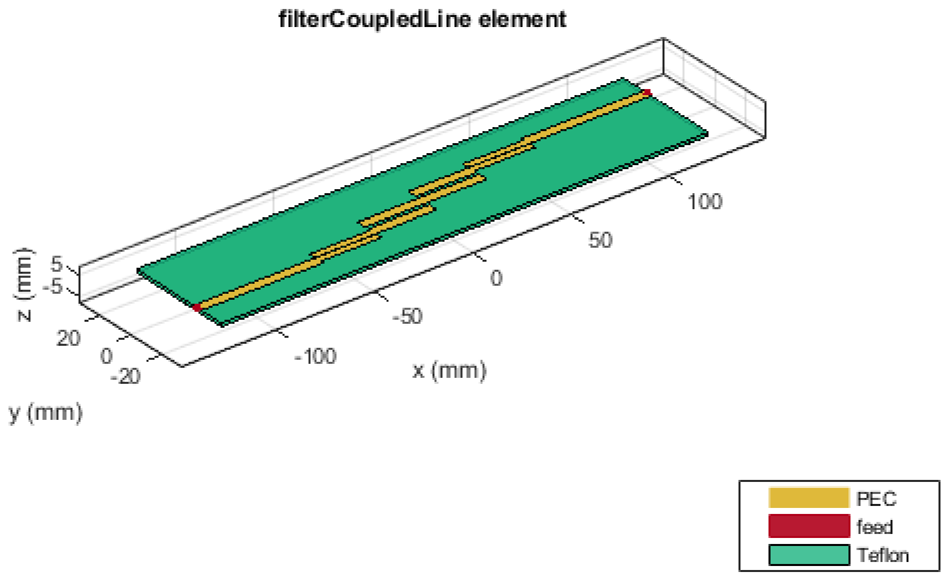
Note PCB components designed using the `design` function operate around the specified frequency with a 10-15% tolerance.

Examples

Design Coupled Line Filter Around 1.8 GHz.

Design a coupled line filter around 1.8 GHz and a fractional bandwidth of 10 percent.

```
clfilter = design(filterCoupledLine,1.8e9,FBW=10);
show(clfilter)
```



Input Arguments

`clfilterobj` — Coupled line filter
`filterCoupledLine` object

Coupled line filter, specified as a `filterCoupledLine` object.

Example: `clfilterobj = filterCoupledLine; design(clfilterobj,2e9)` designs a coupled line filter around a frequency of 2 GHz.

`frequency` — Design frequency of coupled line filter
real positive scalar

Design frequency of coupled line filter, specified as a real positive scalar in hertz.

Example: `5e9`

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `FilterType='ButterWorth'`

`FilterType` — Type of filter
`'Butterworth'` (default) | positive scalar

Type of filter, specified as `'Butterworth'`, `'Chebyshev'`, or `'InverseChebyshev'`.

Data Types: `double`

`FBW` — Fractional bandwidth of filter response
`10` (default) | positive scalar

Fractional bandwidth of the filter response, specified as positive scalar in percents.

Data Types: `double`

`RippleFactor` — Passband factor of Chebyshev filter
`0.01` (default) | positive scalar

Passband factor of the Chebyshev filter, specified as positive scalar in decibels. For Butterworth filter, the passband factor is not required.

Data Types: `double`

Output Arguments

`clfilter` — Coupled line filter around specified reference frequency
`filterCoupledLine` object

Coupled line filter around the specified frequency, returned as a `filterCoupledLine` object.

See Also

`sparameters`

Introduced in R2021b

design

Design stepped impedance low pass filter around desired cut-off frequency

Syntax

```
sifilter = design(sifilterobj,frequency)
sifilter = design( ___ ,Name=Value)
```

Description

`sifilter = design(sifilterobj,frequency)` designs a stepped impedance low pass filter around the cut-off frequency.

`sifilter = design( ___ ,Name=Value)` designs a stepped impedance low pass filter with additional options specified by name-value arguments.

Note

- PCB components designed using the `design` function operate around the specified frequency with a 10-15% tolerance.
- The design for stepped impedance low pass filter is based on analytical equations. Analyzing the parameters using EM-simulation model causes a shift in the cut-off frequency towards the lower end of the frequency range. This is an expected behavior due to the coupling effect.

Examples

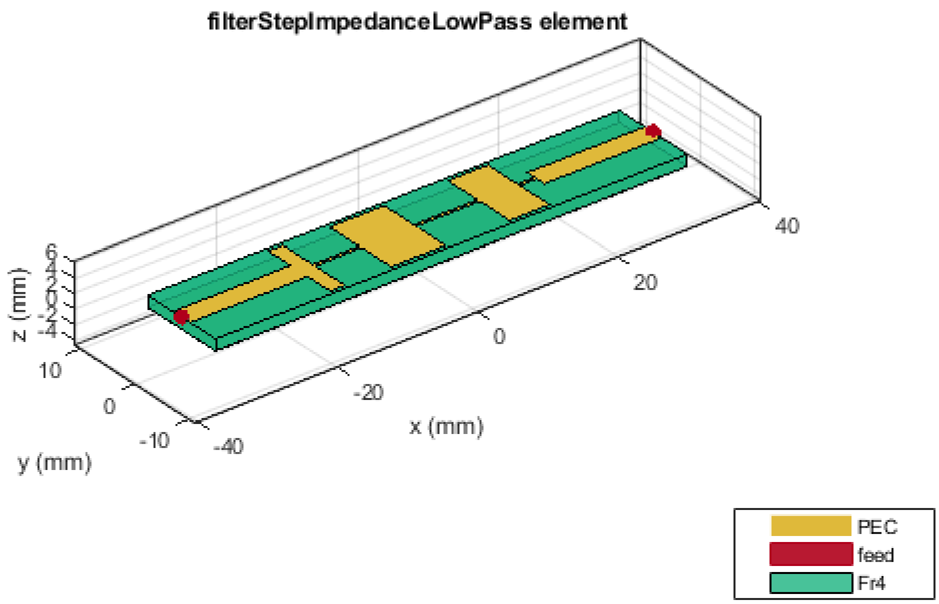
Design Stepped Impedance Low Pass Filter at 2.5 GHz

Design a sixth order stepped impedance low pass filter at 2.5 GHz with 20 ohm low impedance line, 120 ohm high impedance line on FR4 substrate of thickness 1.58 mm.

```
sifilter = filterStepImpedanceLowPass(FilterOrder=6,Height=1.58e-3) ;
Sub  = dielectric(Name='Fr4',EpsilonR=4.2,LossTangent=0.02,Thickness=1.58e-3);
sifilter.Substrate = Sub;
sifilterobj = design(sifilter,2.5e9,Z0=50,LowZ=20,HighZ=120);
```

View the filter.

```
show(sifilterobj);
```



Input Arguments

`sifilterobj` — Stepped impedance low pass line filter
`filterStepImpedanceLowPass` object

Stepped impedance low pass filter, specified as a `filterStepImpedanceLowPass` object.

Example: `sifilterobj = filterStepImpedanceLowPass; design(sifilterobj,2e9)` designs a coupled line filter around a frequency of 2 GHz.

`frequency` — Design frequency of stepped impedance low pass filter
real positive scalar

Design frequency of the stepped impedance low pass filter, specified as a real positive scalar in hertz.

Example: `5e9`

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `Z0=75`

Z0 — Reference impedance
50 (default) | positive scalar

Reference impedance, specified as a positive scalar in ohms.

Data Types: double

LowZ — Low impedance line
20 (default) | positive scalar

Low impedance line, specified as a positive scalar in ohms.

Data Types: double

HighZ — High impedance line
120 (default) | positive scalar

High impedance line, specified as a positive scalar in ohms.

Data Types: double

FilterType — Type of filter
'Butterworth' (default) | positive scalar

Type of filter, specified as 'Butterworth', or 'Chebyshev'.

Data Types: char | string

RippleFactor — Passband factor of Chebyshev filter
0.01 (default) | positive scalar

Passband factor of the Chebyshev filter, specified as positive scalar in decibels. For Butterworth filter, the passband factor is not required.

Data Types: double

Output Arguments

sifilter — Stepped impedance low pass filter around specified frequency
filterStepImpedanceLowPass object

Stepped impedance low pass filter around specified frequency, returned as a filterStepImpedanceLowPass object.

See Also

sparameters

Introduced in R2021b

design

Design hairpin filter around specified frequency

Syntax

```
hpfilter = design(hpfilterobj,frequency)
hpfilter = design( ___ ,Name=Value)
```

Description

`hpfilter = design(hpfilterobj,frequency)` designs a hairpin filter around the specified frequency.

`hpfilter = design( ___ ,Name=Value)` designs a hairpin filter with additional options specified by name-value arguments.

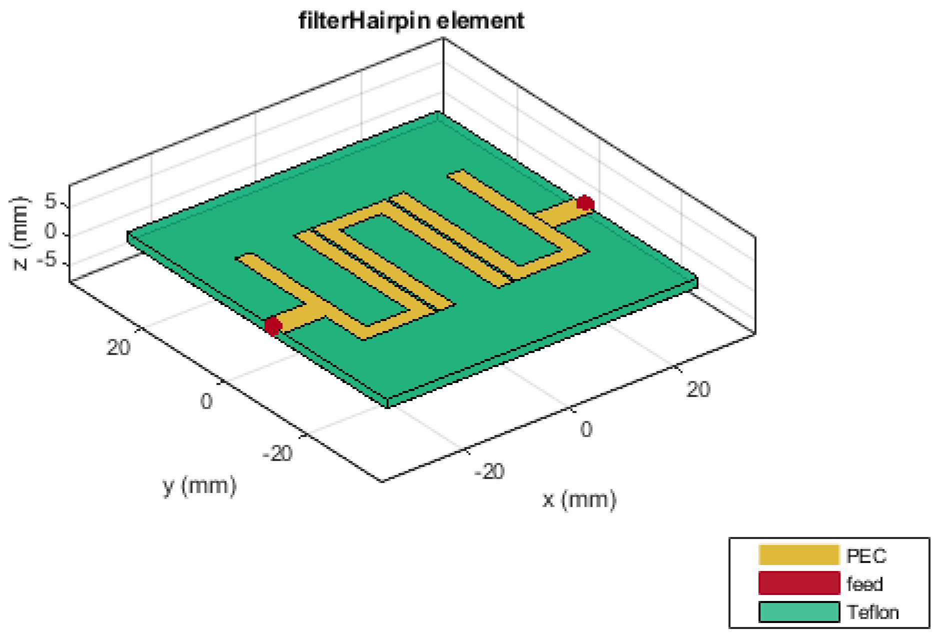
Note PCB components designed using the `design` function operate around the specified frequency with a 10-15% tolerance.

Examples

Design Chebyshev Hairpin Filter at 1.8 GHz

Design a Hairpin filter with a Chebyshev response at 1.8 GHz and a fractional bandwidth of 10 percent.

```
hpfilt = design(filterHairpin,1.8e9,FBW=10,FilterType='Chebyshev');
show(hpfilt);
```



Design Fifth Order Hairpin Filter

Design a 5th order tapped input hairpin filter with a Chebyshev response at 1.8 GHz and a fractional bandwidth of 10 percent.

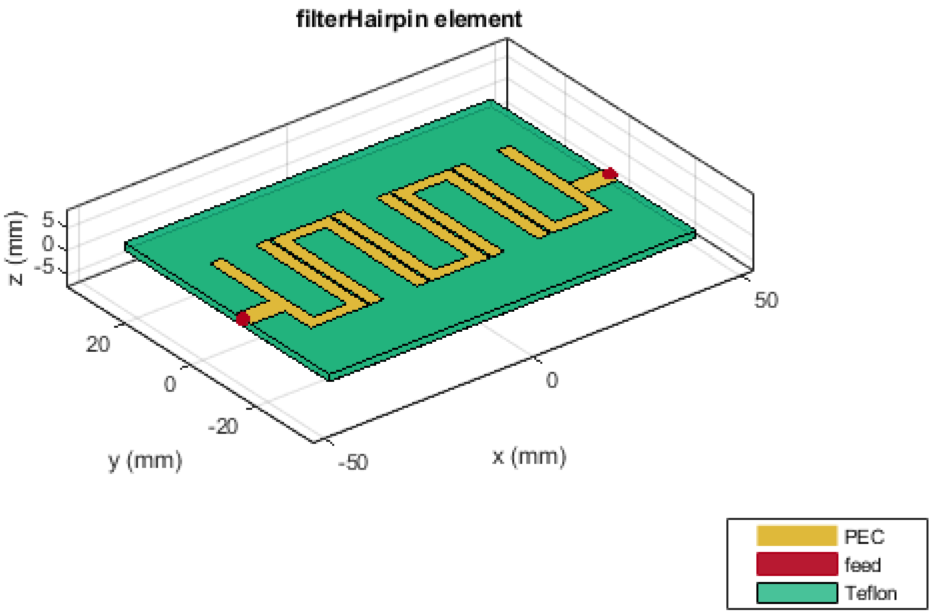
```
hpfilt = filterHairpin(FeedType='Tapped')
```

```
hpfilt =
  filterHairpin with properties:

           Resonator: [1x1 ubendRightAngle]
         FilterOrder: 3
     ResonatorOffset: [0 0 0]
             Spacing: [4.0000e-04 4.0000e-04]
      PortLineLength: 0.0080
       PortLineWidth: 0.0050
          FeedOffset: [-0.0055 -0.0055]
            FeedType: 'Tapped'
              Height: 0.0016
    GroundPlaneWidth: 0.0567
           Substrate: [1x1 dielectric]
           Conductor: [1x1 metal]
```

```
hpfilt.FilterOrder = 5;
hpfilt = design(hpfilt, 1.8e9,FBW=10,FilterType='Chebyshev');
```

View the filter.

```
show(hpfilt);
```



Input Arguments

`hpfilterobj` — Hairpin filter
`filterHairpin` object

Hairpin filter, specified as a `filterHairpin` object.

Example: `hpfilterobj = filterHairpin; design(hpfilterobj,2e9)` designs a hairpin filter around a frequency of 2 GHz.

`frequency` — Design frequency of hairpin filter
real positive scalar

Design frequency of the hairpin filter, specified as a real positive scalar in hertz.

Example: `5e9`

Data Types: `double`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1,...,NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Example: `RippleFactor=0.02`

`FilterType` — Type of filter
`'Butterworth'` (default) | positive scalar

Type of filter, specified as `'Butterworth'`, or `'Chebyshev'`.

Data Types: `char` | `string`

`RippleFactor` — Passband factor of Chebyshev filter
`0.01` (default) | positive scalar

Passband factor of the Chebyshev filter, specified as positive scalar in decibels. For Butterworth filter, the passband factor is not required.

Data Types: `double`

Output Arguments

`hpfilter` — Hairpin filter operating around specified frequency
`filterHairpin` object

Hairpin filter operating around the specified frequency, returned as a `filterHairpin` object.

See Also
`sparameters`

Introduced in R2021b

pcbcascade

Create new component using cascade operation

Syntax

```
combinedcomponent = pcbcascade(component1,component2)
combinedcomponent = pcbcascade(component1,component2,m,n)
```

Description

`combinedcomponent = pcbcascade(component1,component2)` creates a new component by using a cascade operation along port 2 of the first component and port 1 of the second component.

`combinedcomponent = pcbcascade(component1,component2,m,n)` creates a new component by using a cascade operation along port *m* of the first component and port *n* of the second component.

Note

- `pcdcascade` only supports: 2-metal layer PCB components, feeds specified at the edge of components, and identical substrate properties in both components.
- If either of the components is an antenna, the new component that the object creates is a `pcbStack` object.
- To use `pcbStack` object you require Antenna Toolbox.

Examples

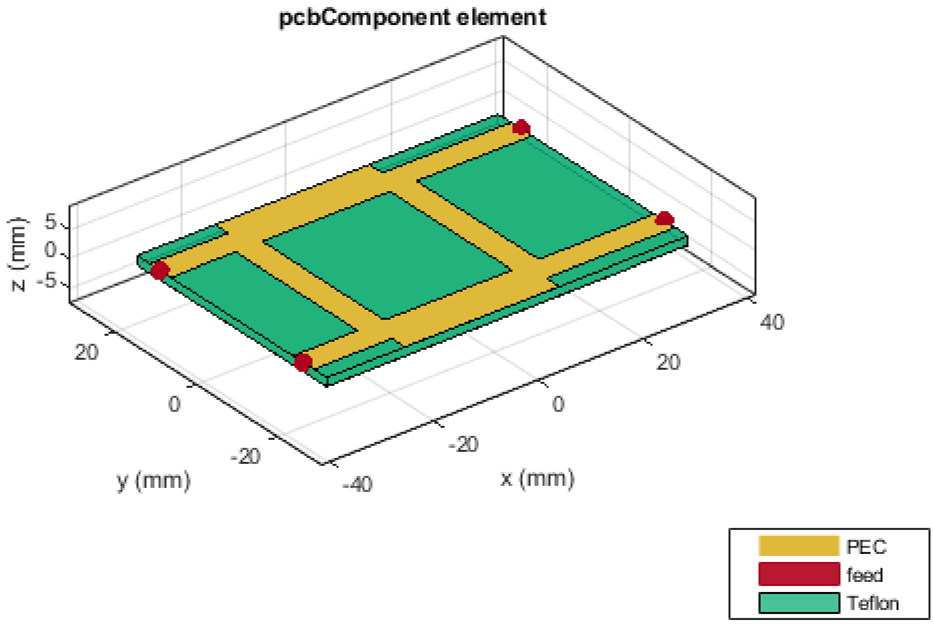
Create Component From Branchline Coupler and Coupled Microstrip Line

Create a new component by cascading a branchline coupler with a coupled microstrip line.

```
c = design(couplerBranchline,5.6e9);
mc = design(coupledMicrostripLine,5.6e9);
mc.Spacing = c.ShuntArmLength;
r = pcbcascade(c,mc);
```

View the new component.

```
figure
show(r)
```



Input Arguments

component1 — PCB component or antenna
PCB component object | antenna object

PCB component or antenna, specified a PCB component object or antenna object. For a complete list of the PCB components, see “PCB Components Catalog”.

Example: `mline1 = microstripLine; mline2 = design(microstripLine,3e9); component = pcbcascade(mline1,mline2)` creates a new component by cascading `mline1` and `mline2`.

Data Types: `char` | `string`

component2 — PCB component or antenna
PCB component object | antenna object

PCB component or antenna, specified a PCB component object or antenna object. For a complete list of the PCB components, see “PCB Components Catalog”.

Example: `mline1 = microstripLine; mline2 = design(microstripLine,3e9); component = pcbcascade(mline1,mline2)` creates a new component by cascading `mline1` and `mline2`.

Data Types: `char` | `string`

`m` — Port number of first component
`1` (default) | positive scalar

Port number of the first component, specified as a positive scalar.

Example: `coupler = couplerRatrace;mline = microstripLine;component = pcbcascade(coupler,mline,3,1)` creates a new component by cascading Port 3 of the coupler to Port 1 of the microstrip transmission line.

Data Types: `double`

`n` — Port number of second component
`1` (default) | positive scalar

Port number of second component, specified as a positive scalar.

Example: `coupler = couplerRatrace;mline = microstripLine;component = pcbcascade(coupler,mline,3,1)` creates a new component by cascading Port 3 of the coupler to Port 1 of the microstrip transmission line.

Data Types: `double`

See Also
`pcbComponent`

Introduced in R2021b

design

Design PCB component around particular frequency

Syntax

```
rfpcbcomponent = design(rfpcbobject,frequency)
```

Description

`rfpcbcomponent = design(rfpcbobject,frequency)` designs a PCB component around the specified frequency.

Note PCB components designed using the `design` function operate around the specified frequency with a 10-15% tolerance.

Examples

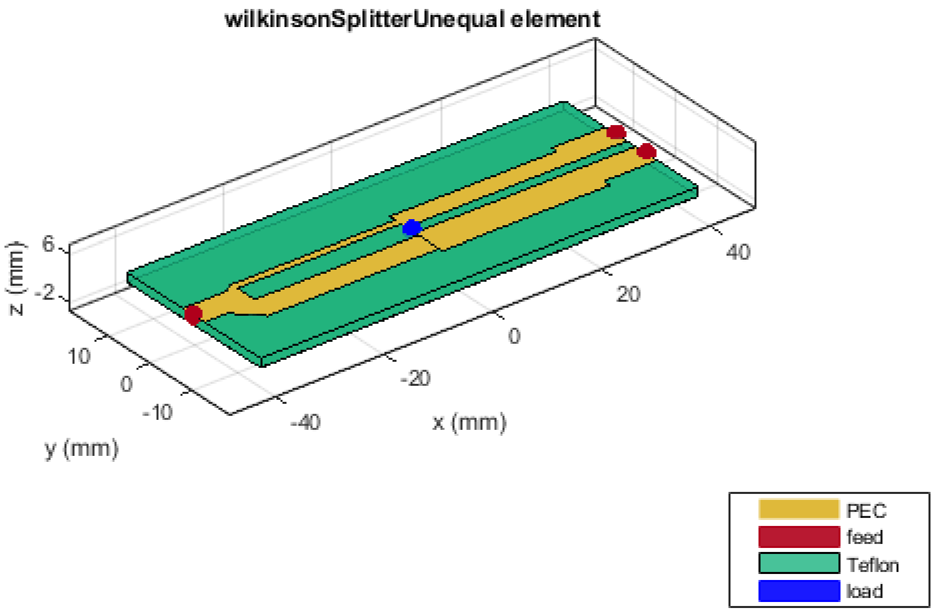
Design Unequal Wilkinson Splitter at 1.8 GHZ

Design an unequal Wilkinson around 1.8 GHz.

```
uwilk = design(wilkinsonSplitterUnequal,1.8e9);
```

View the splitter

```
show(uwilk);
```

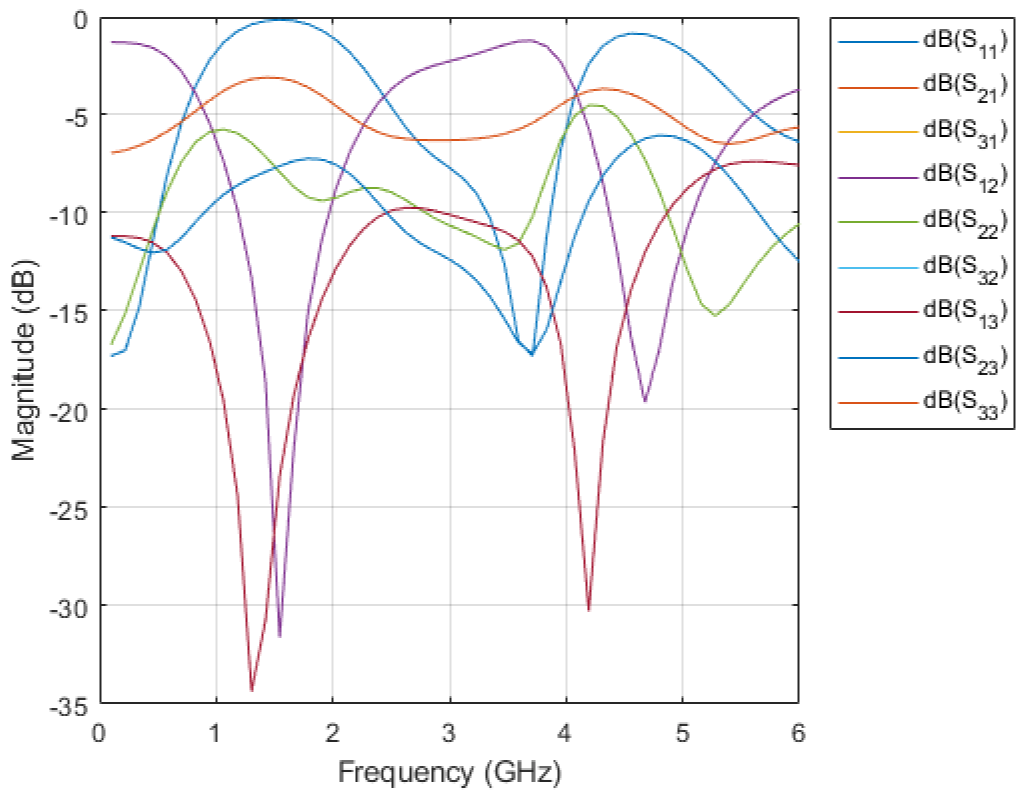


Plot S-parameters between the frequency range of 0.1 GHz to 6 GHZ.

```
spar = sparameters(uwilk,linspace(0.1e9,6e9,50))
```

```
spar = 
  sparameters: S-parameters object

       NumPorts: 3
    Frequencies: [50x1 double]
     Parameters: [3x3x50 double]
      Impedance: 50

  rfparam(obj,i,j) returns S-parameter Sij
```

```
rfplot(spar)
```



Input Arguments

`rfpcbobject` — PCB component object
object handle

PCB component object, specified as a RF PCB object. For complete list of PCB components, microstrip bends, and traces, see “PCB Components Catalog”.

Note The following PCB catalog components are not supported by this function: `couplerLange`, `spiralInductor`, `interdigitalCapacitor`, and `stubRadialShunt`.

`frequency` — Design frequency of PCB component
real positive scalar

Design frequency of the PCB component, specified as a real positive scalar in hertz.

Example: `3e9`

Data Types: `double`

Output Arguments

`rfpcbcomponent` — PCB catalog component operating around specified frequency
object handle

PCB catalog component operating around specified frequency, returned as a object handle.

See Also

`sparameters`

Introduced in R2021b