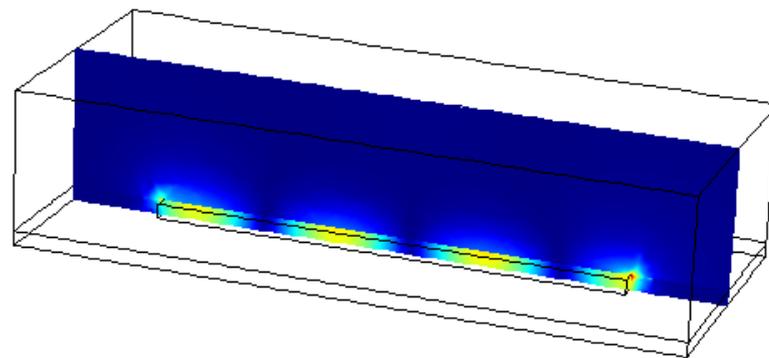
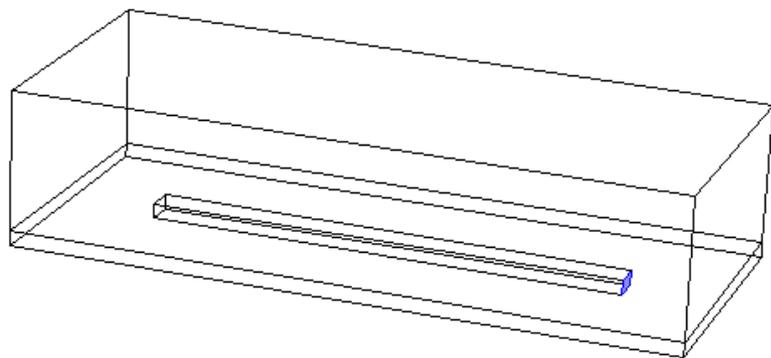
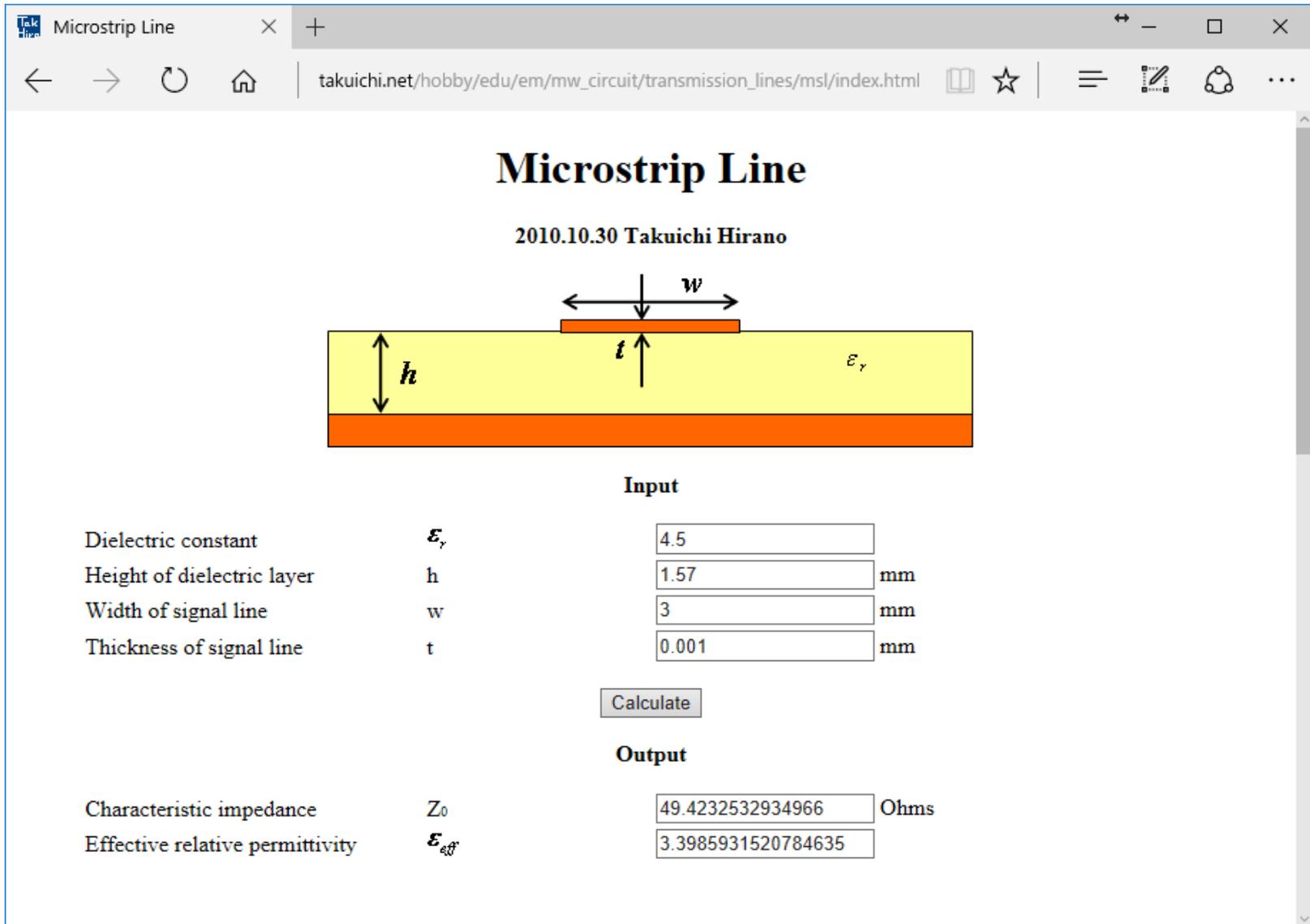


# COMSOLによるマクロストリップ線路の解析 ～ 集中ポート ～



平野 拓一

# マクロストリップ線路(初期設計)



**Microstrip Line**  
2010.10.30 Takuichi Hirano

Diagram illustrating a microstrip line structure. The signal line has width  $w$  and thickness  $t$ . The dielectric layer has height  $h$  and permittivity  $\epsilon_r$ .

**Input**

Dielectric constant	$\epsilon_r$	<input type="text" value="4.5"/>	
Height of dielectric layer	$h$	<input type="text" value="1.57"/>	mm
Width of signal line	$w$	<input type="text" value="3"/>	mm
Thickness of signal line	$t$	<input type="text" value="0.001"/>	mm

**Output**

Characteristic impedance	$Z_0$	<input type="text" value="49.4232532934966"/>	Ohms
Effective relative permittivity	$\epsilon_{eff}$	<input type="text" value="3.3985931520784635"/>	

[http://www.takuichi.net/hobby/edu/em/mw\\_circuit/transmission\\_lines/msl/](http://www.takuichi.net/hobby/edu/em/mw_circuit/transmission_lines/msl/)

# 1: モデル化 (Port1)

The screenshot displays the COMSOL Multiphysics interface for a model named 'msl\_pec\_lump\_port2.mph'. The left-hand 'Model Builder' shows a hierarchical tree with components like 'Global Definitions', 'Materials', 'Definitions', 'ジオメトリ 1' (containing blocks 1, 2, 4), 'Materials', '電磁波 (周波数領域) (em)', 'メッシュ 1', 'スタディ 1', and 'Results'. The central 'Properties' panel is set to 'Lumped Port 1' and shows the following configuration:

- Label: Lumped Port 1
- Boundary Selection: Manual selection of 11 boundaries, which are active.
- Override and Contribution: (Expanded)
- Equation: (Expanded)
- Lumped Port Properties:
  - Lumped port name: 1
  - Type of lumped port: Uniform
  - Terminal type: Cable
  - Wave excitation at this port: On
  - Voltage:  $V_0$  1[V] V
  - Port phase:  $\theta_{in}$  0 rad
- Settings:
  - Characteristic impedance:  $Z_{ref}$  50[ohm]  $\Omega$

The right-hand 'Graphics' window shows a 3D perspective view of a rectangular waveguide. The dimensions are indicated as 0.02 m in height, 0.01 m in width, and 1.5 m in length. A small blue rectangular lumped port is attached to the bottom surface of the waveguide. A coordinate system with x, y, and z axes is visible at the bottom left of the graphics window.

At the bottom of the interface, the 'Messages' window shows the following text:

```
COMSOL Multiphysics 5.3.0.260  
Opened file: G:\Home\hira2\public_html\em_analysis\canonical\msl\msl_pec_lump_port2.mph
```

The status bar at the very bottom indicates a memory usage of 764 MB | 934 MB.

# 1: モデル化 (Port2)

The screenshot displays the COMSOL Multiphysics interface for a model named `mssl_pec_lump_port2.mph`. The **Model Builder** tree on the left shows the hierarchy: `Global Definitions` (Parameters, Materials), `コンポーネント 1 (comp1)` (Definitions, ジオメトリ 1 with blocks `blk1`, `blk5`, `blk4`, and a completed integrated model), `Materials` (Electromagnetic (Frequency Domain) `em` with `波動方程式 (電場) 1`, `電気壁 (PEC) 1`, `初期値 1`, `電気壁 (PEC) 2`, `散乱境界条件 1`, `Lumped Port 1`, and `Lumped Port 2`), `メッシュ 1`, `スタディ 1`, and `Results` (Data Sets, Views, Derived Values, Tables, `電場 (emw)`, Export, Reports).

The **Settings** pane for `Lumped Port 2` is active, showing the following configuration:

- Label:** Lumped Port 2
- Boundary Selection:** Selection: Manual, Active: 14
- Override and Contribution:** (Expanded)
- Equation:** (Expanded)
- Lumped Port Properties:**
  - Lumped port name: 2
  - Type of lumped port: Uniform
  - Terminal type: Cable
  - Wave excitation at this port: Off
- Settings:** Characteristic impedance:  $Z_{ref}$  50[ohm]  $\Omega$

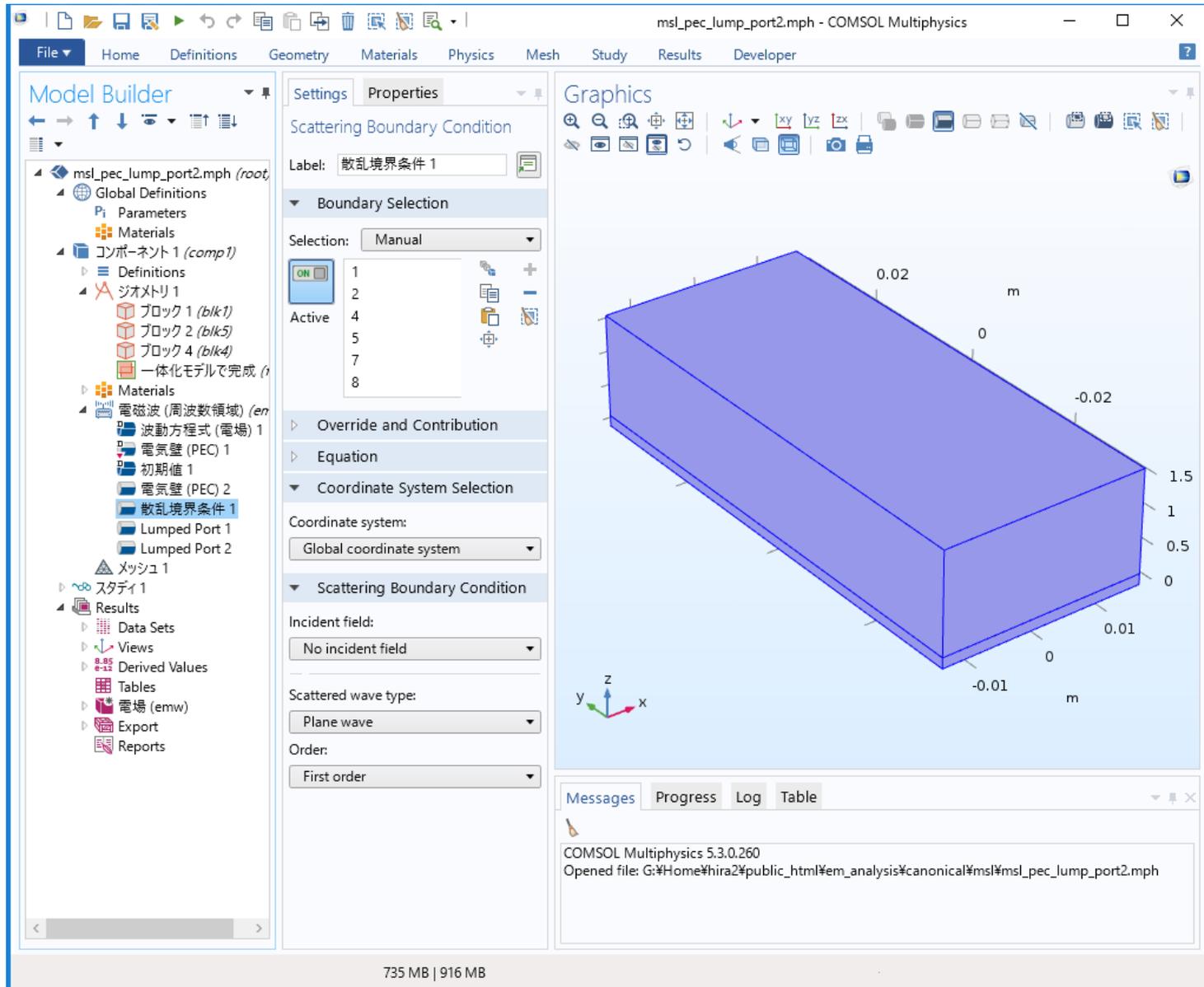
The **Graphics** window shows a 3D wireframe model of a rectangular waveguide. The dimensions are indicated as 0.02 m in the x-direction, 0.01 m in the y-direction, and 1.5 m in the z-direction. A small blue rectangular port is located on the bottom surface of the waveguide. A coordinate system with x, y, and z axes is shown at the bottom left of the graphics window.

The **Messages** pane at the bottom right displays the following text:

```
COMSOL Multiphysics 5.3.0.260
Opened file: G:\Home\hira2\public_html\em_analysis\canonical\mssl\mssl_pec_lump_port2.mph
```

At the bottom of the interface, the file size is shown as 758 MB | 927 MB.

# 1: モデル化 (散乱境界条件)



# 1: モデル化 (電気壁1)

The screenshot displays the COMSOL Multiphysics interface for a model named "msl\_pec\_lump\_port2.mph". The software is running on version 5.3.0.260. The main window is divided into several panes:

- Model Builder:** Shows the hierarchical structure of the model. The "Perfect Electric Conductor" boundary condition is applied to "電気壁 (PEC) 1".
- Settings/Properties:** Displays the configuration for the selected boundary condition. The label is "電気壁 (PEC) 1". The boundary selection is set to "All boundaries". The "Active" checkbox is checked. The "Override and Contribution" and "Equation" sections are currently collapsed.
- Graphics:** Shows a 3D visualization of the model. The geometry is a rectangular box with a central slot. The dimensions are indicated as 0.02 m in the x-direction, 0.01 m in the y-direction, and 1.5 m in the z-direction. The PEC boundary is highlighted in blue.
- Messages:** Shows the log of the software startup, indicating the file path: "G:\Home\hira2\public\_html\em\_analysis\canonical\msl\msl\_pec\_lump\_port2.mph".

At the bottom of the window, the system memory usage is shown as 745 MB | 929 MB.

# 1: モデル化 (電気壁2)

msl\_pec\_lump\_port2.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results Developer

Model Builder

- msl\_pec\_lump\_port2.mph (root)
  - Global Definitions
    - Parameters
    - Materials
  - コンポーネント 1 (comp1)
    - Definitions
    - ジオメトリ 1
      - ブロック 1 (blk1)
      - ブロック 2 (blk5)
      - ブロック 4 (blk4)
      - 一体化モデルで完成 (f)
    - Materials
      - 電磁波 (周波数領域) (emw)
        - 波動方程式 (電場) 1
        - 電気壁 (PEC) 1
        - 初期値 1
        - 電気壁 (PEC) 2
        - 散乱境界条件 1
        - Lumped Port 1
        - Lumped Port 2
    - メッシュ 1
  - スタディ 1
  - Results
    - Data Sets
    - Views
    - Derived Values
    - Tables
    - 電場 (emw)
    - Export
    - Reports

744 MB | 931 MB

# 2: 解析条件 (周波数領域)

The screenshot displays the COMSOL Multiphysics software interface for a frequency domain analysis. The main window is titled "msl\_pec\_lump\_port2.mph - COMSOL Multiphysics".

**Model Builder:** The left sidebar shows the model tree. Under "コンポーネント 1 (comp1)", the "電磁波 (周波数領域) (emw)" component is selected. The "Step 1: 周波数領域" is also highlighted.

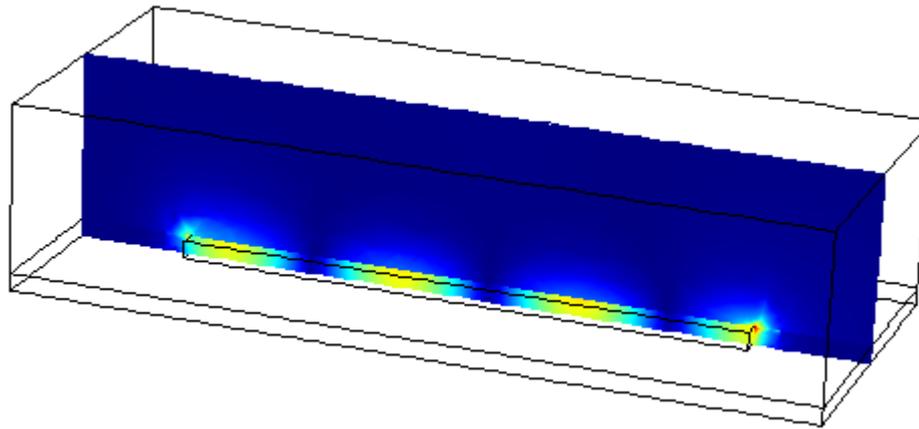
**Settings - Properties:** The central panel shows the configuration for the "周波数領域" (Frequency Domain) study. The "Frequency unit" is set to "GHz". The "Frequencies" field is set to "5 GHz". The "Load parameter values" and "Reuse solution from previous step" options are visible.

**Graphics:** The right panel shows a 3D wireframe model of a rectangular waveguide structure. The dimensions are indicated as 0.02 m in the x-direction, 0.01 m in the y-direction, and 1.0 m in the z-direction. A coordinate system (x, y, z) is shown at the bottom left of the graphics area.

**Messages:** The bottom panel shows the message log, indicating the file path: "G:\Home\hira2\public\_html\em\_analysis\canonical\msl\msl\_pec\_lump\_port2.m".

766 MB | 941 MB

# 3: 結果



freq (GHz)  
5.0000

S パラメーター (dB), 11 成分 (dB)  
-18.779

S パラメーター (dB), 21 成分 (dB)  
-0.35669

# 4: 周波数スイープと全ポート解析

The screenshot displays the COMSOL Multiphysics software interface for a project named 'msl\_pec\_lump\_port2.mph'. The top ribbon includes tabs for File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, Results, and Developer. The Physics tab is active, showing 'Electromagnetic Wave (Frequency Domain)' and 'Add Physics' options. The Mesh tab shows 'Build Mesh' and 'メッシュ 1'. The Study tab shows 'Compute', 'Study 2', and 'Add Study'. The Results tab shows 'S-parameter (emw) 1' and 'Add Plot Group'. The Layout tab shows 'Windows' and 'Reset Desktop'.

The Model Builder on the left shows a tree view of the model structure. Under 'Component 1 (comp1)', the 'Electromagnetic Wave (Frequency Domain)' interface is expanded, showing 'Wave Equation, Electric 1', 'Perfect Electric Conductor (PEC)', 'Initial Values 1', 'Perfect Electric Conductor (PEC)', 'Scattering Boundary Condition', 'Lumped Port 1', 'Lumped Port 2', and 'Far-Field Domain 1'. A pink box highlights 'Study 2' and its sub-items: 'Step 1: Frequency Domain', 'Solver Configurations', 'Job Configurations', and 'Results'.

The Properties window for the 'Frequency Domain' study is open, showing the following settings:

- Label: Frequency Domain
- Study Settings:
  - Frequency unit: GHz
  - Frequencies: range(3,1,10) GHz
  - Load parameter values: Browse... Read File
  - Reuse solution from previous step: Auto
- Results While Solving
- Physics and Variables Selection:
  - Modify model configuration for study step:
  - Physics interface: Electromagnetic Wave... Solve for:  Discretization: Physics settin...
- Values of Dependent Variables
- Mesh Selection
- Adaptation and Error Estimates
- Study Extensions

The Graphics window on the right shows a 3D model of a rectangular structure with a central slot, rendered in a light green color. The coordinate system (x, y, z) is visible at the bottom left of the graphics area.

The Progress window at the bottom right shows 'Progress' and 'Log' tabs, with a progress bar and a log area.

762 MB | 963 MB

# 4: 周波数スイープと全ポート解析

The screenshot displays the COMSOL Multiphysics interface for a model named 'msl\_pec\_jump\_port2.mph'. The 'Model Builder' tree on the left shows the hierarchy: Global Definitions, Component 1 (comp1), Electromagnetic Wave (Frequency Domain), and Parametric Sweep. The 'Settings' window for the 'Electromagnetic Waves, Frequency Domain' study is open, showing 'Domain Selection' set to 'All domains' and 'Formulation' set to 'Full field'. The 'Port Sweep Settings' section is expanded, with 'Use port sweep' checked. A 'Configure Sweep Settings' button is highlighted with a pink box and the label 'クリック'. Below it, 'Export Touchstone file' is checked, and the 'Touchstone file export' path is set to 'D:\tmp\%s\_para.s2p'. A pink box around the 'Touchstone file export' section is labeled '自動で生成される'. A pink arrow points from the 'Parametric Sweep' node in the Model Builder to the 'Use port sweep' checkbox, with the label 'チェック'. The 'Graphics' window on the right shows a 3D visualization of a rectangular waveguide structure on a substrate.

Touchstoneファイルをエクスポートすると便利

# 4: 周波数スイープと全ポート解析

The screenshot displays the COMSOL Multiphysics interface for a study named "mSl\_pec\_lump\_port2.mph". The software is in the "Study" tab, showing a "Parametric Sweep" configuration. The "Model Builder" on the left lists the model components: Geometry 1 (Block 1, 2, 3, Form Union), Materials (Vacuum, Dielectric), Electromagnetic Wave (Frequency Domain) (Wave Equation, Electric 1, Perfect Electric Conductor, Initial Values 1, Perfect Electric Conductor, Scattering Boundary Condition, Lumped Port 1, Lumped Port 2, Far-Field Domain 1), and Mesh 1. The "Study" tree shows "Study 2" with a "Parametric Sweep" step highlighted. The "Settings" panel for the Parametric Sweep shows "Sweep type" set to "Specified combinations" and a table with one parameter: "PortName" with a value list of "1,2". The "Output While Solving" section has "Plot" checked and "Plot group" set to "E-field". The "Probes" section has "All" selected. The "Default solver sequence generation" is set to "Using global parameters". The "Graphics" window shows a 3D model of a rectangular structure with a central slot, rendered in green. The "Progress" window at the bottom shows the simulation progress.

COMSOL Multiphysics Interface: mSl\_pec\_lump\_port2.mph

Study: Electromagnetic Wave (Frequency Domain)

Parametric Sweep Settings:

- Label: Parametric Sweep
- Compute: Update Solution
- Sweep type: Specified combinations
- Parameter list:

Parameter name	Parameter value list	Parameter unit
PortName	1,2	
- Output While Solving: Plot (checked), Plot group: E-field
- Probes: All
- Default solver sequence generation: Using global parameters
- Keep solutions in memory: All

Model Builder:

- Definitions
- Geometry 1
  - Block 1 (blk1)
  - Block 2 (blk4)
  - Block 3 (blk5)
  - Form Union (fin)
- Materials
  - Vacuum (mat1)
  - Dielectric (mat2)
- Electromagnetic Wave (Frequency Domain)
  - Wave Equation, Electric 1
  - Perfect Electric Conductor (PEC) 1
  - Initial Values 1
  - Perfect Electric Conductor (PEC) 2
  - Scattering Boundary Condition 1
  - Lumped Port 1
  - Lumped Port 2
  - Far-Field Domain 1
- メッシュ 1

Study 2:

- Step 1: Frequency Domain
- Solver Configurations
- Parametric Sweep (highlighted)
- Step 1: Frequency Domain
- Solver Configurations
- Job Configurations
- Results

Graphics: Convergence Plot 1

Progress: Log

1.37 GB | 1.85 GB

→Study 2をCompute

# 4: 周波数スイープと全ポート解析

The screenshot displays the COMSOL Multiphysics interface for a parametric sweep analysis. The Model Builder on the left shows a tree structure with 'Study 2' containing a 'Parametric Sweep' and 'Step 1: Frequency Domain'. The Properties panel in the center shows the 'Global' settings for the 'Plot' of 'Global 1'. The 'y-Axis Data' table is as follows:

Expression	Unit	Description
emw.S11dB	1	S11
emw.S12dB	1	S12
emw.S21dB	1	S21
emw.S22dB	1	S22

The 'Convergence Plot 1' on the right shows the S-parameter (dB) versus Frequency (GHz). The plot includes four data series: S11 (blue), S12 (green), S21 (red), and S22 (cyan). A pink annotation 'Sijが計算される' is placed above the plot. The S22 series shows a significant decrease in dB as frequency increases, while S11, S12, and S21 remain relatively flat near 0 dB.