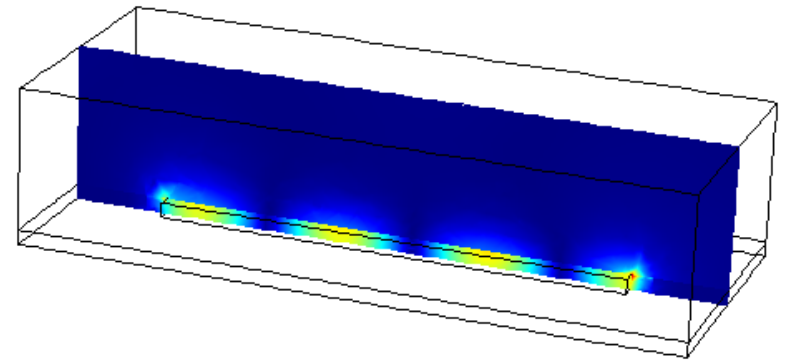
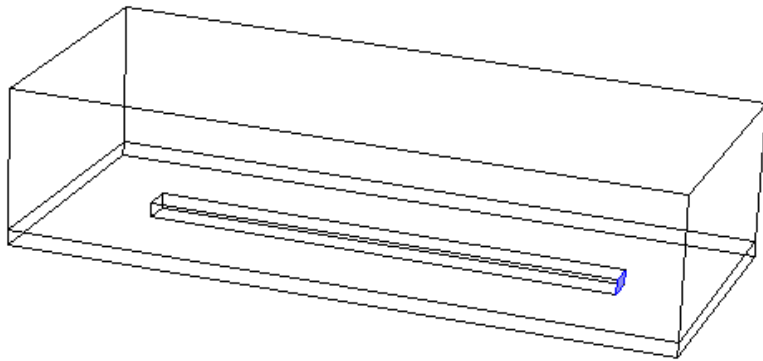
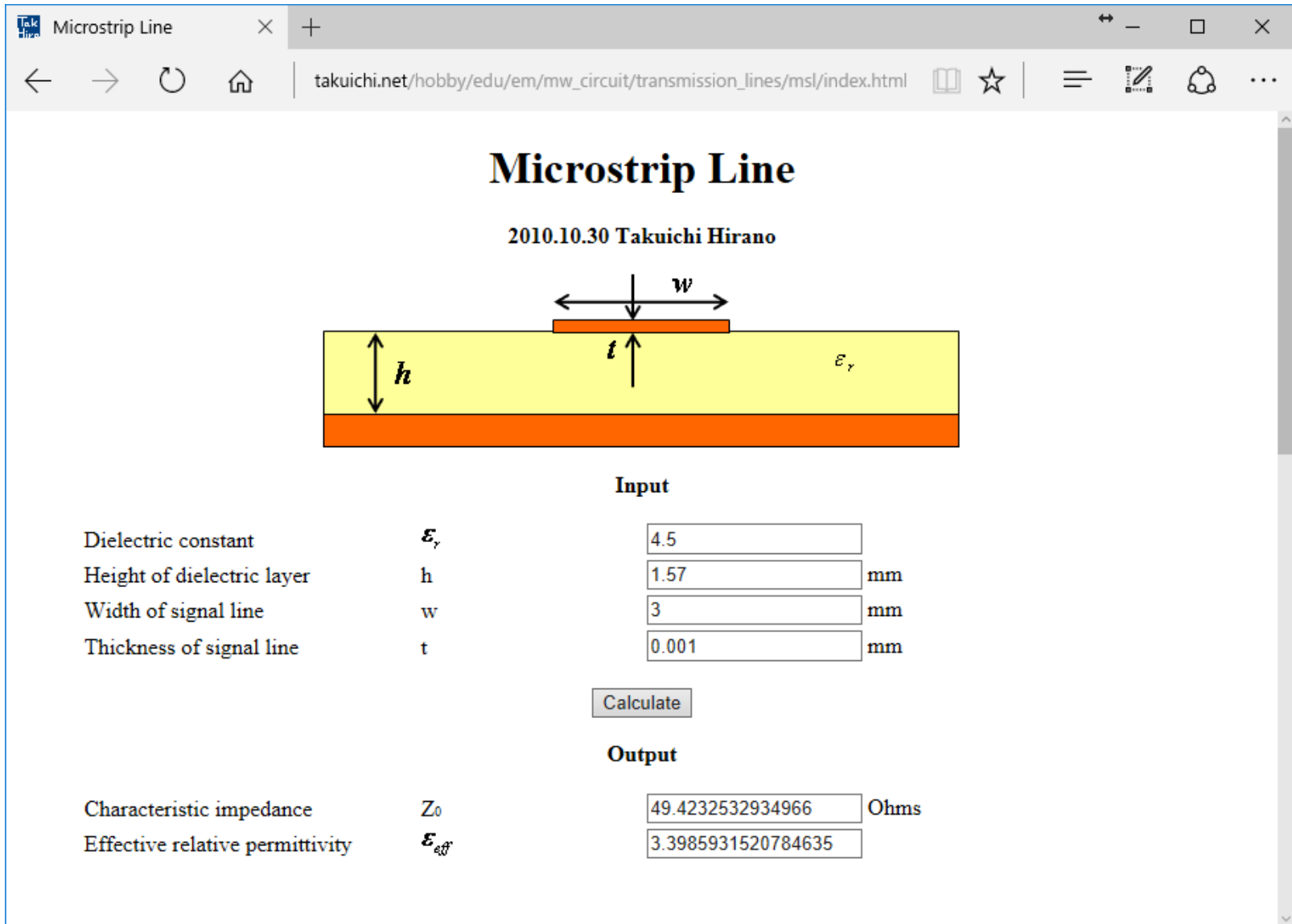


COMSOLによるマクロストリップ線路の解析 ～ 集中ポート ～



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 ϵ_r .

Input

Dielectric constant	ϵ_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	ϵ_{eff}	<input type="text" value="3.3985931520784635"/>	

http://www.takuichi.net/hobby/edu/em/mw_circuit/transmission_lines/msl/

1: モデル化 (Port1)

The screenshot displays the COMSOL Multiphysics software interface for a model named 'msl_pec_lump_port2.mph'. The interface is divided into several main sections:

- Model Builder (Left):** A tree view showing the model's structure. It includes 'Global Definitions' (Parameters, Materials), 'Component 1 (comp1)' (Definitions, Geometry 1 with blocks 1, 2, 4, and a completed integrated model), 'Materials' (Electromagnetic (frequency domain), Wave equation (electric field), PEC walls, Initial values, Lumped ports), 'Mesh 1', 'Study 1', and 'Results' (Data Sets, Views, Derived Values, Tables, Electric field (emw), Export, Reports).
- Properties Panel (Center):** Shows the configuration for 'Lumped Port 1'.
 - Boundary Selection:** Selection: Manual. A list shows boundary 11 is selected and active.
 - 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: θ_{in} 0 rad
 - Characteristic impedance: Z_{ref} 50[ohm] Ω
- Graphics (Right):** A 3D wireframe view of a rectangular waveguide. The dimensions are indicated: length 0.02 m, width 0.01 m, and height 0.01 m. A coordinate system (x, y, z) is shown at the bottom left. A small blue rectangular port is visible on the bottom surface of the waveguide.
- Messages (Bottom Right):** A log window showing the message: 'COMSOL Multiphysics 5.3.0.260 Opened file: G:\Home\hira2\public_html\em_analysis\canonical\msl\msl_pec_lump_port2.mph'.

At the bottom of the window, the memory usage is shown as 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** on the left shows a hierarchical tree with components like `コンポーネント 1 (comp1)`, `ジオメトリ 1`, and `電磁波 (周波数領域) (em)`. The **Properties** panel in the center is set to `Lumped Port 2` and shows the following settings:

- Label:** Lumped Port 2
- Boundary Selection:** Manual selection, with 14 active boundaries.
- Lumped Port Properties:**
 - Lumped port name: 2
 - Type of lumped port: Uniform
 - Terminal type: Cable
 - Wave excitation at this port: Off
- Settings:** Characteristic impedance: $Z_{ref} = 50[\text{ohm}]$

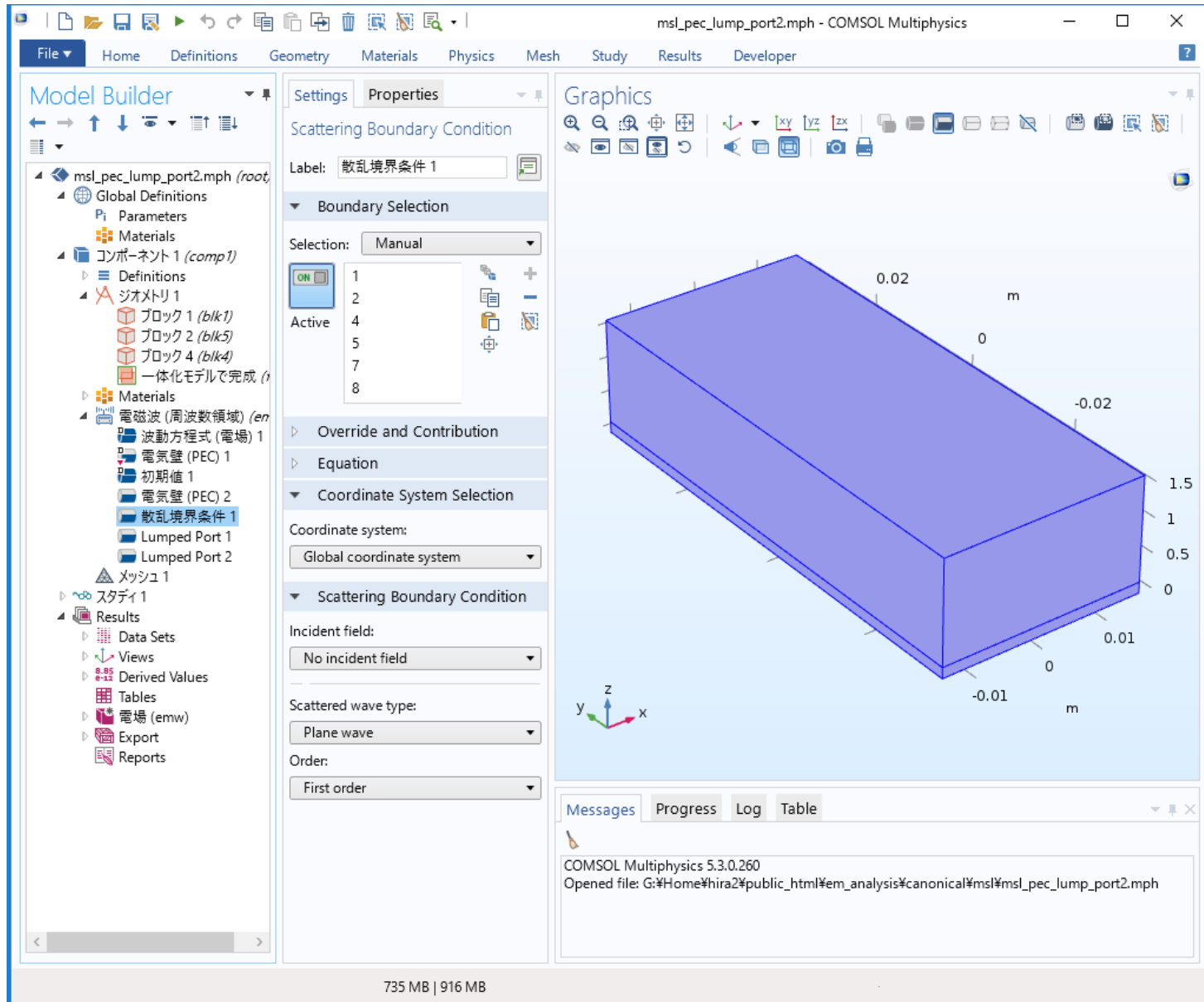
The **Graphics** window on the right shows a 3D wireframe model of a rectangular waveguide. The dimensions are indicated as `0.02 m` for the width, `0.01 m` for the depth, and `1.5 m` for the length. A coordinate system with `x`, `y`, and `z` axes is shown at the bottom left of the graphics area.

The **Messages** window 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)



735 MB | 916 MB

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 **Model Builder** tree on the left shows the following structure:

- 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 (i)
 - Materials
 - 電磁波 (周波数領域) (em)
 - 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

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

Settings Properties

Perfect Electric Conductor

Label: 電気壁 (PEC) 2

Boundary Selection

Selection: Manual

ON 13

Active

Override and Contribution

Equation

Graphics

0.02 m

0

-0.02

1.5

1

0.5

0

0.01

-0.01 m

z

y

x

Messages Progress Log Table

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

744 MB | 931 MB

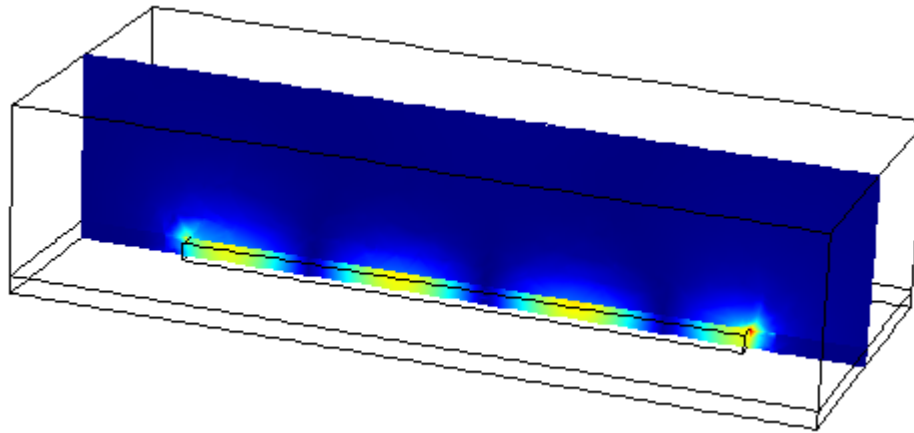
2: 解析条件 (周波数領域)

The screenshot displays the COMSOL Multiphysics software interface for a model named 'msl_pec_lump_port2.mph'. The interface is divided into several main sections:

- Model Builder:** Shows a hierarchical tree of the model. The 'Study 1' (スタディ 1) is expanded to 'Step 1: 周波数領域' (Step 1: Frequency Domain). The physics tree includes 'Electromagnetic Waves, Frequency Domain' (電磁波 (周波数領域) (emw)) with various boundary conditions and ports.
- Settings / Properties:** The 'Frequency Domain' settings are visible. The 'Label' is '周波数領域'. The 'Frequency unit' is set to 'GHz'. The 'Frequencies' are set to '5 GHz'. The 'Load parameter values' and 'Reuse solution from previous step' options are also shown.
- Graphics:** A 3D wireframe model of a rectangular waveguide structure is displayed. The dimensions are indicated as 0.02 m in length, 0.01 m in width, and 0.005 m in height. A coordinate system (x, y, z) is shown at the bottom left.
- Messages:** The bottom panel shows the message log with the text: 'COMSOL Multiphysics 5.3.0.260' and 'Opened file: G:\Home\hira2\public_html\em_analysis\canonical\msl\msl_pec_lump_port2.m'.

At the bottom center of the window, the memory usage is displayed as '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 'メッシュ 1' (Mesh 1) is also listed. 'Study 2' is selected, and its 'Step 1: Frequency Domain' is highlighted with a pink box. Below it are 'Solver Configurations', 'Job Configurations', and 'Results'.

The Properties panel for the 'Frequency Domain' study is visible. It includes a 'Settings' tab and a 'Properties' tab. The 'Frequency Domain' section shows 'Compute' as the study type. The 'Study Settings' section includes 'Frequency unit' set to 'GHz', 'Frequencies' set to 'range(3,1,10) GHz', and 'Reuse solution from previous step' set to 'Auto'. The 'Physics and Variables Selection' section shows 'Electromagnetic Wave...' selected under 'Solve for'.

The Graphics window on the right shows a 3D view of the model, which is 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 window.

The Progress and Log window at the bottom right shows the simulation progress and log output.

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'. The 'Graphics' window on the right shows a 3D model of a rectangular waveguide structure on a substrate. The 'Progress' and 'Log' windows are visible at the bottom.

チェック

クリック

自動で生成される

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 model named 'mSl_pec_lump_port2.mph'. The software is in the 'Study' tab, showing a 'Parametric Sweep' configuration for 'Study 2'. The 'Model Builder' on the left lists the model's components, including 'Geometry 1' (Block 1, 2, 3, Form Union), 'Materials' (Vacuum, Dielectric), 'Electromagnetic Wave (Frequency Domain)', and 'Mesh 1'. The 'Parametric Sweep' settings are shown in the center, with 'Sweep type' set to 'Specified combinations' and a table defining the parameter 'PortName' with a value of '1,2'. The 'Output While Solving' section is also visible, with 'Plot group' set to 'E-field'. The 'Graphics' window on the right shows a 3D view of the model, a green rectangular structure with a central slot, on a grid. The status bar at the bottom indicates '1.37 GB | 1.85 GB'.

Parameter na	Parameter value list	Parameter unit
PortName	1,2	

自動で生成されている

→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 view 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). S11, S12, and S21 are constant at 0 dB, while S22 starts at approximately -28 dB and decreases to -50 dB. A pink annotation 'Sijが計算される' is placed above the plot.