



# P&S COMSOL® Design Tool

## Lecture Week 3: Introduction to COMSOL

Yannik Horst, Manuel Kohli, Xinzhi Zhang

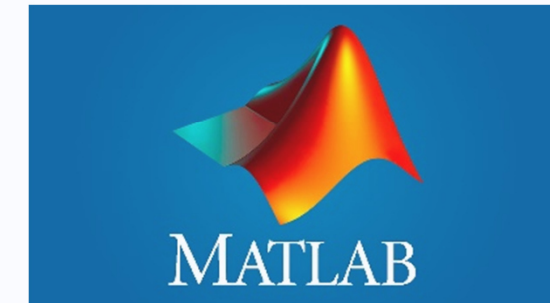
# Content

- Software & Physics
- Meshing
- Boundary Conditions
- Source
- General Workflow
- Tutorial 3: Young's Slit Experiments

# Software & Physics

## Software & Physics

### Simulation Software



### Simulation Method

Finite Element Method (FEM)



Finite Difference Time Domain (FDTD)



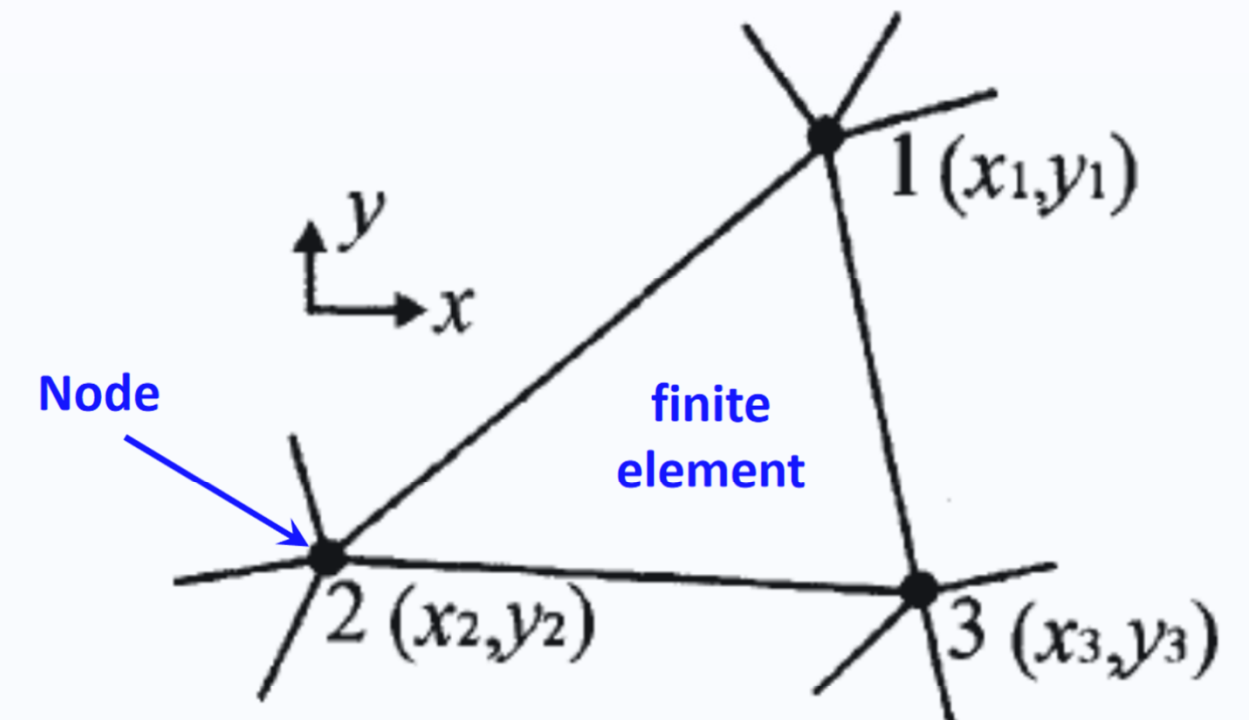
# Software & Physics

## Software & Physics



### FEM

1. *Division of space into finite elements*
  - *Local refinement*
  - *Almost perfectly conformal*
2. *Formulation of a boundary value problem*
  - *Field calculated at the node and interpolated at edges*
3. *Elimination of time derivatives (steady state)*



### Advantages of FEM

- + Modeling
- + Accuracy specification
- + Boundary conditions
- + Visualization
- + Material properties

### Disadvantages of FEM

- Data required as input for mesh
- Sometimes large computational power
- Output can vary
- Frequency domain

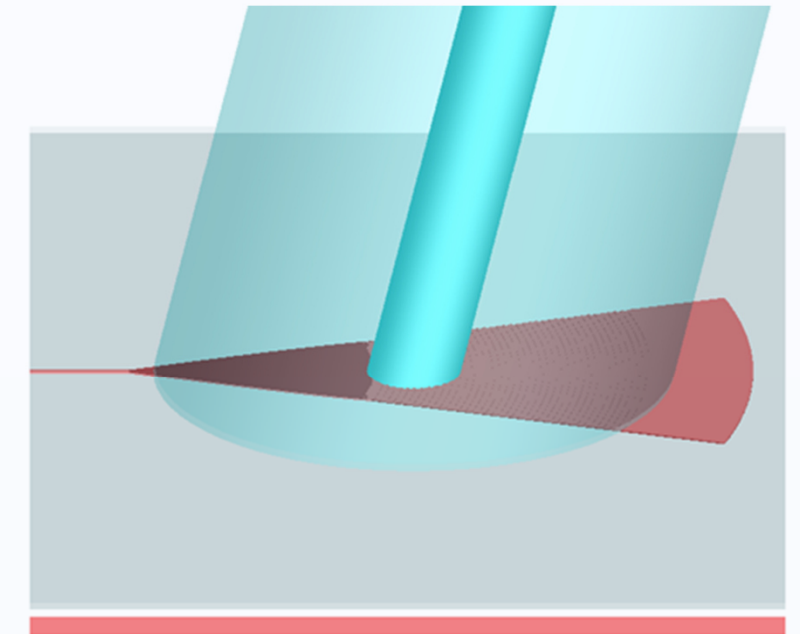
# Software & Physics

## Software & Physics



### FDTD

1. *Replace derivative with finite difference, discretize in space and time*
2. *Solve resulting difference equations*
3. *Evaluate magnetic field one time-step in future*
4. *Evaluate electric field one time-step in future*
5. *Repeat the previous two steps*



### Advantages of FDTD

- + Intuitive
- + Time-domain technique
- + Animated display of evolving fields
- + Direct usage of  $E$ - and  $H$ -Fields

### Disadvantages of FDTD

- Rectangular mesh
  - Conformality and refinement
- Material boundaries

# Meshing

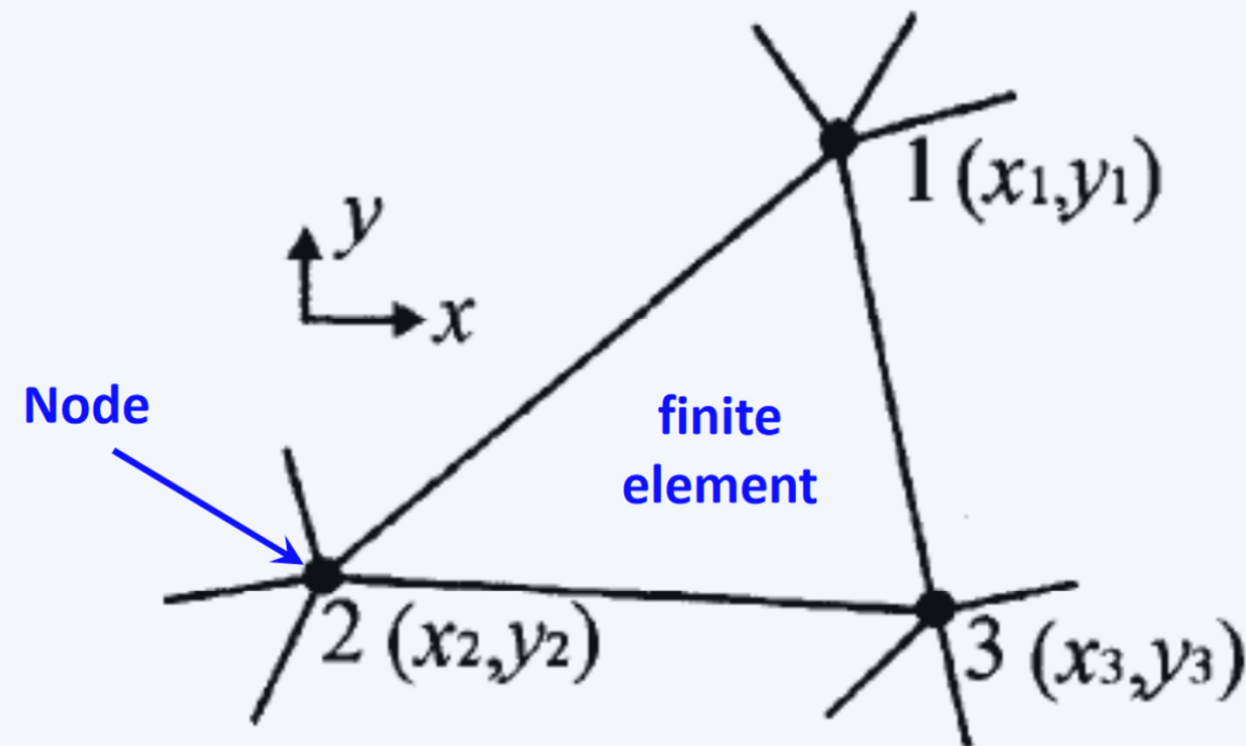
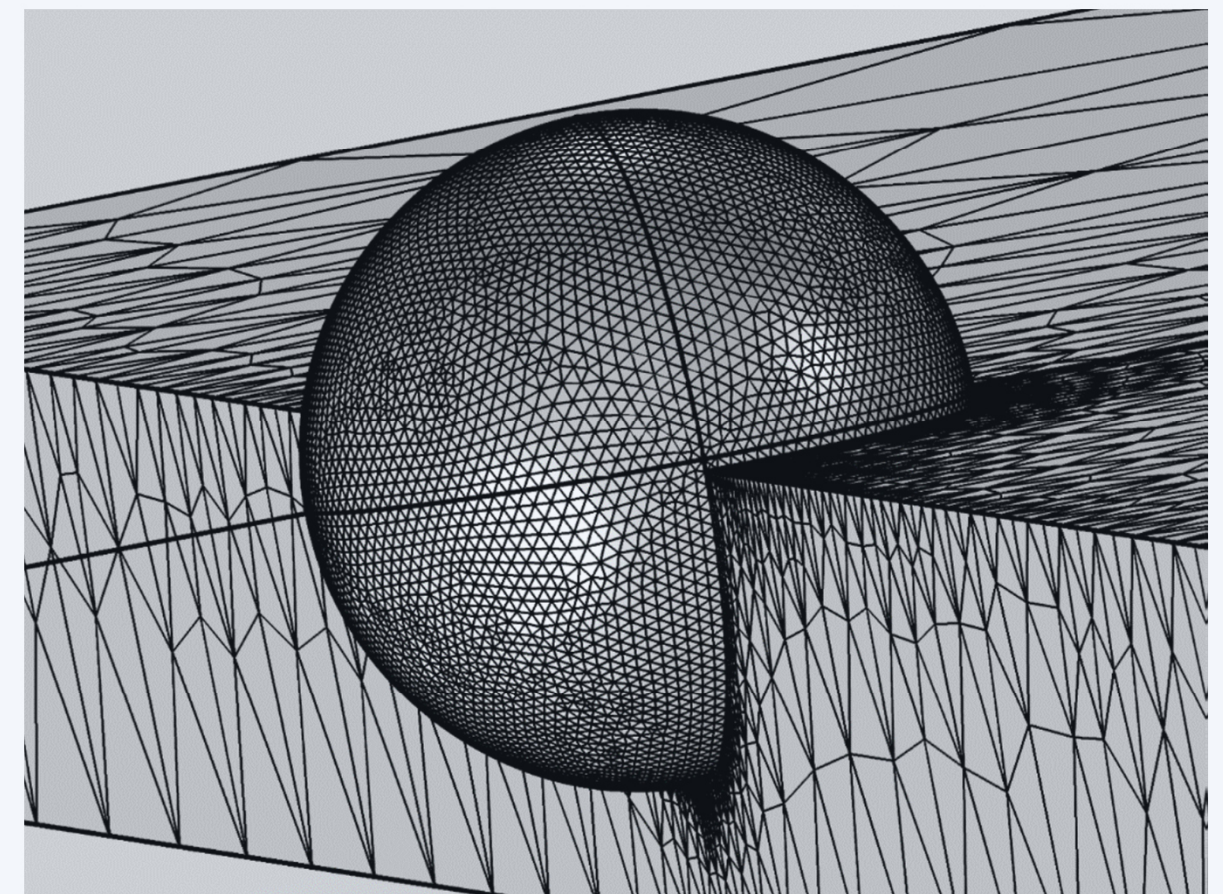
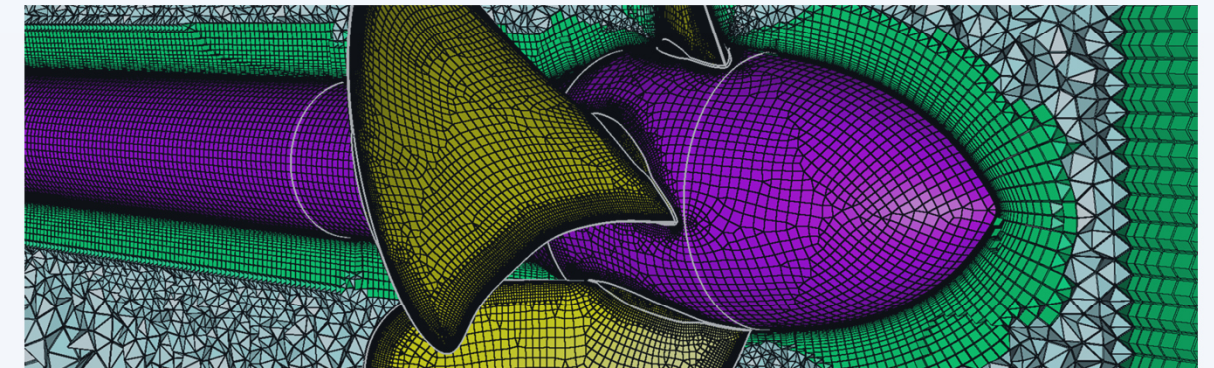
## Software & Physics



FEM, FDTD, FDFD,  
FEM, BEM

Frequency Domain,  
Acoustics, Fluids,  
Plasma, etc

## Domain & Mesh



# Meshing

## Software & Physics

COMSOL



cādence

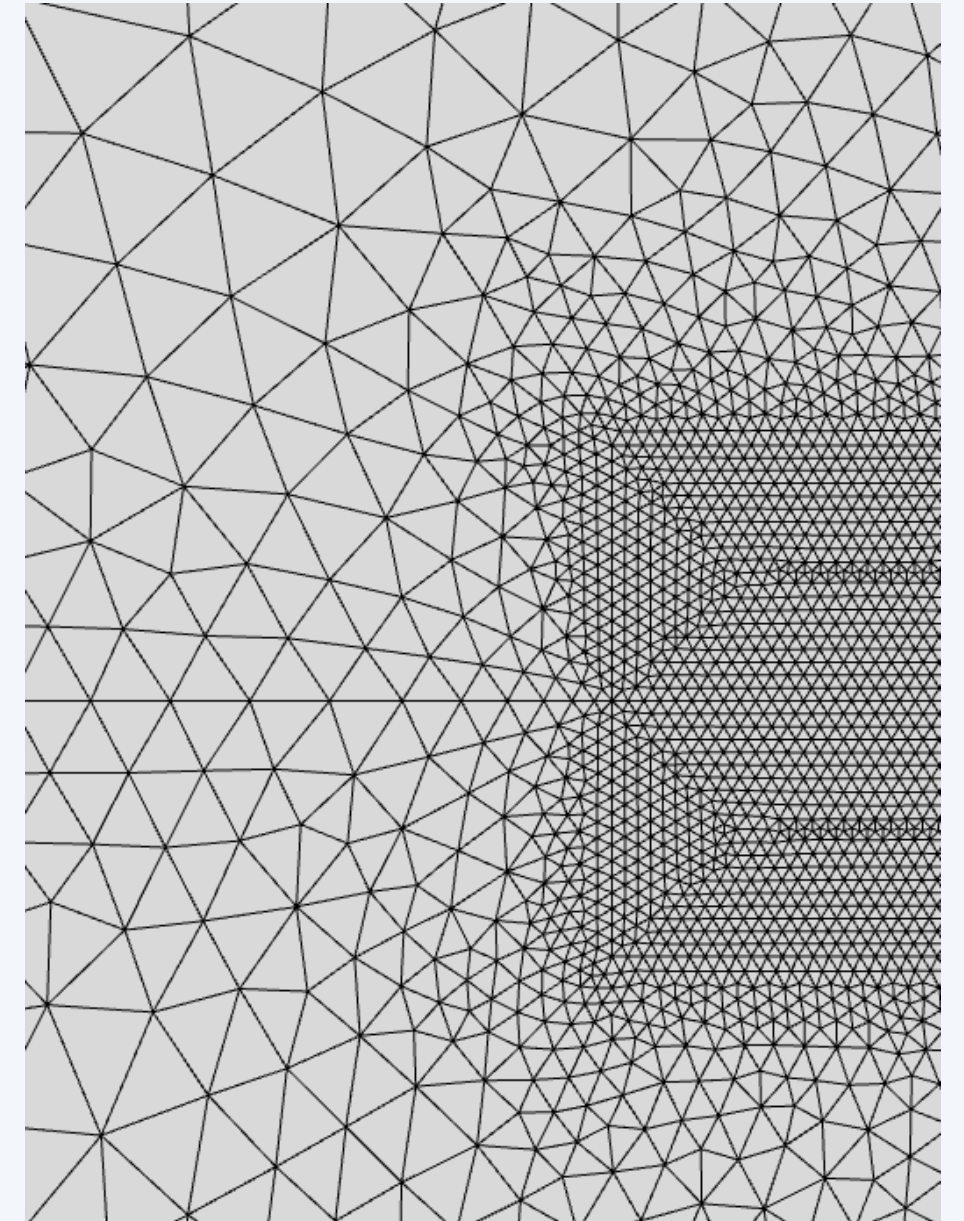


FEM, FDTD, FDFD,  
FEM, BEM

Frequency Domain,  
Acoustics, Fluids,  
Plasma, etc

## Domain & Mesh

- Discretization of simulation domain
- Mesh size determines accuracy of solution
  - Too large mesh → wrong results
- Accuracy vs. simulation time
  - Too small mesh → very large simulation time
  - If RAM is too low, data is written onto hard drive → ultra large simulation time



# Boundary Conditions

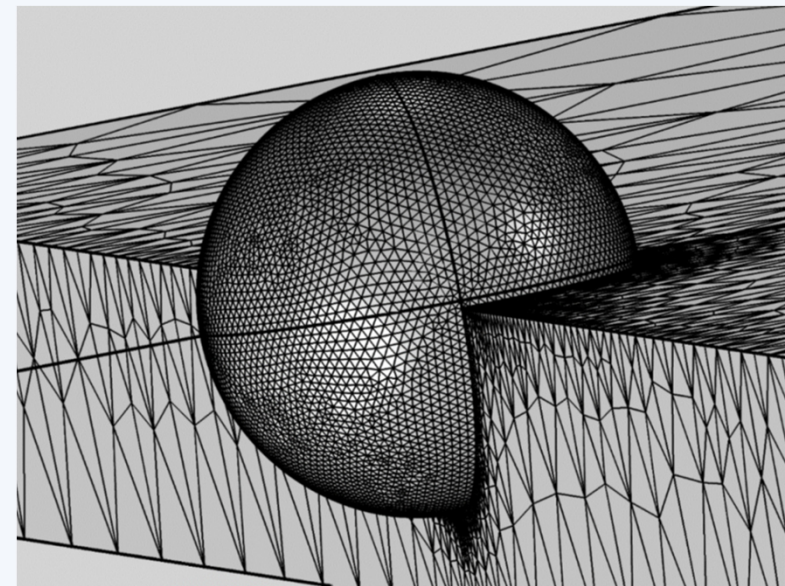
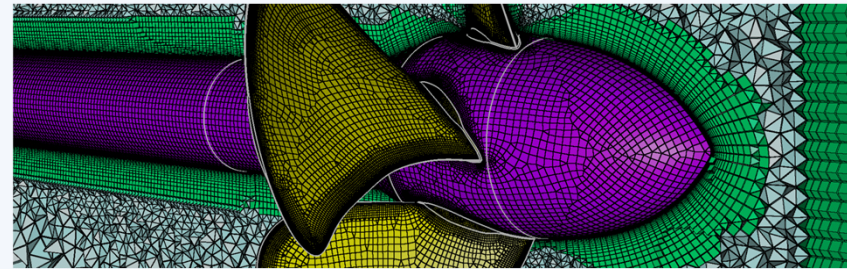
## Software & Physics



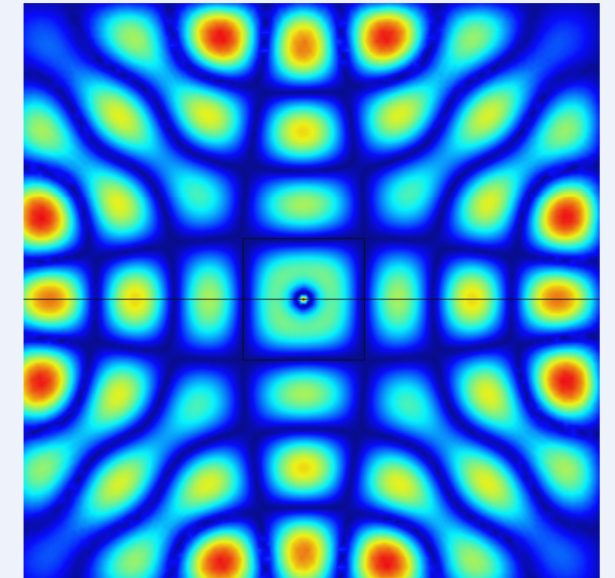
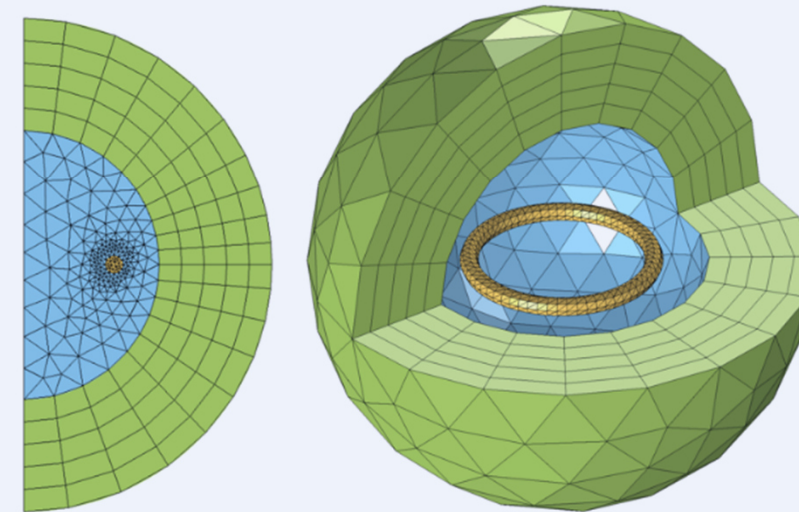
FEM, FDTD, FDFD,  
FEM, BEM

Frequency Domain,  
Acoustics, Fluids,  
Plasma, etc

## Domain & Mesh



## Boundary Conditions



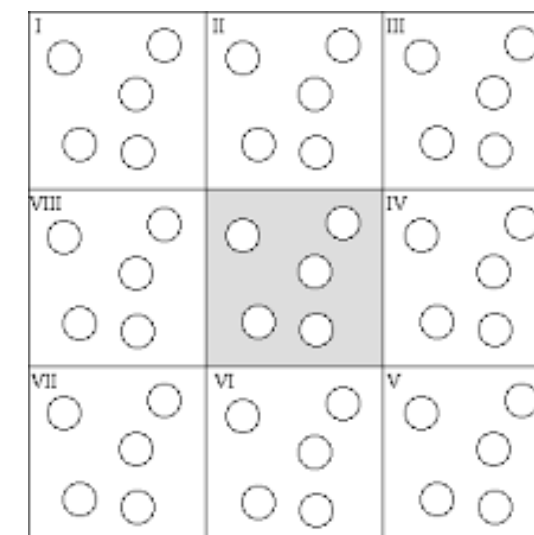
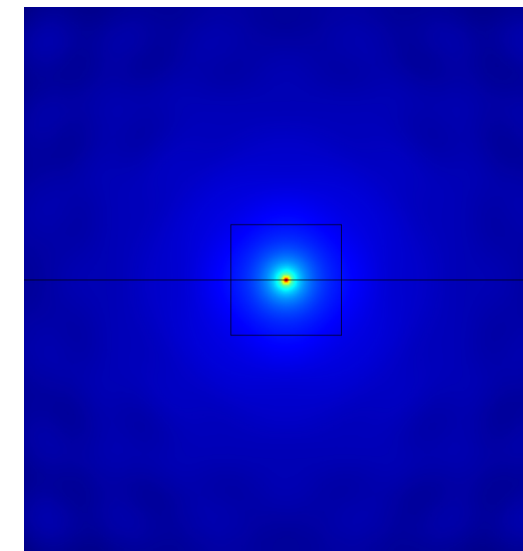
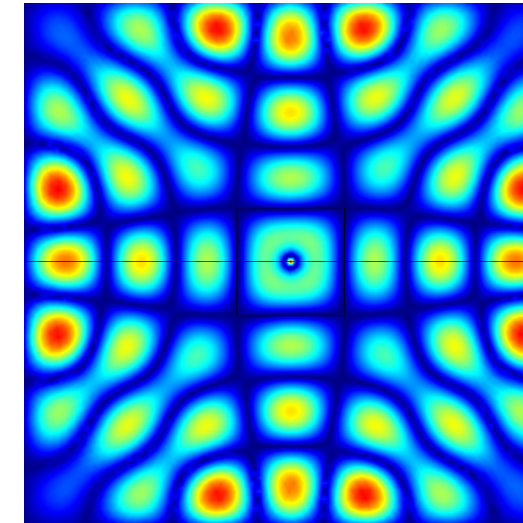
S

- Types of boundary conditions in COMSOL
- Perfect Electric Conductor (PEC)
  - Perfect Magnetic Conductor (PMC)
  - Scattering Boundary Condition
  - Periodic Boundaries Condition (PBC)
  - Perfectly Matched Layer (PML)



# Boundary Conditions

- **Perfect Electric Conductor (PEC)**
  - Properties
    - Electric field cannot penetrate → reflection of electric field
    - Equivalent to infinite electric conductivity
- **Perfect Magnetic Conductor (PMC)**
  - Properties
    - Magnetic field cannot penetrate → reflection of magnetic field
    - Equivalent to infinite magnetic conductivity
- **Scattering Boundary Condition**
  - Properties
    - Electric field is absorbed → no reflection
- **Periodic Boundary Condition**
  - For repeating structures
  - Use a unit cell for the analysis
  - Simulates systems expanding infinitely in 1D/2D



# Source

## Software & Physics

COMSOL



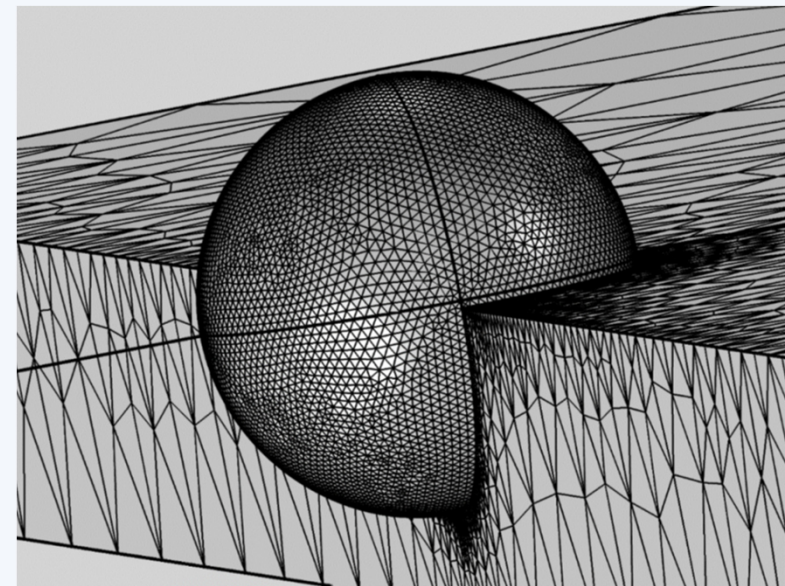
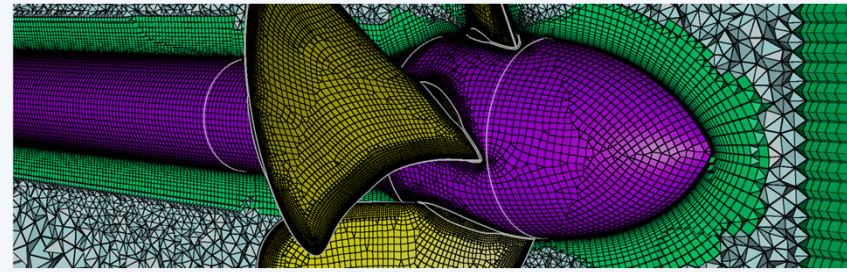
cadence



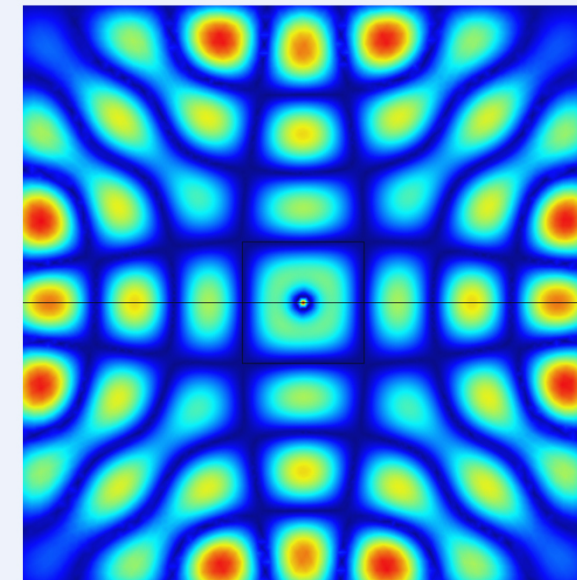
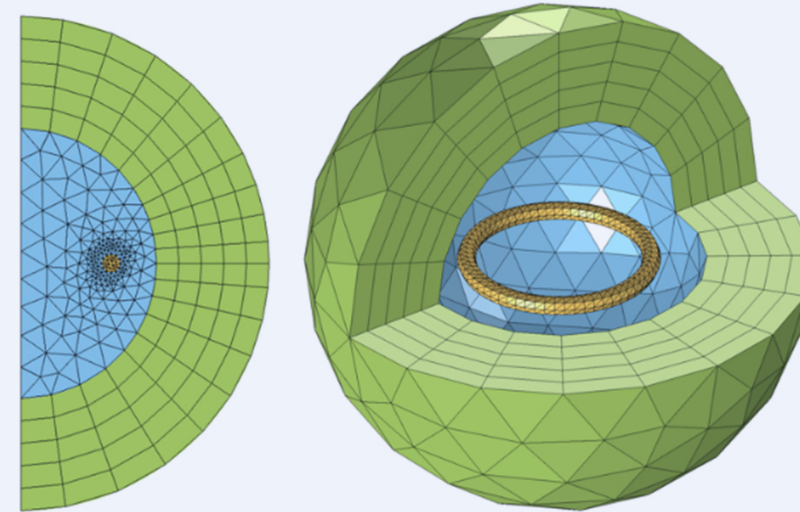
FEM, FDTD, FDFD,  
FEM, BEM

Frequency Domain,  
Acoustics, Fluids,  
Plasma, etc

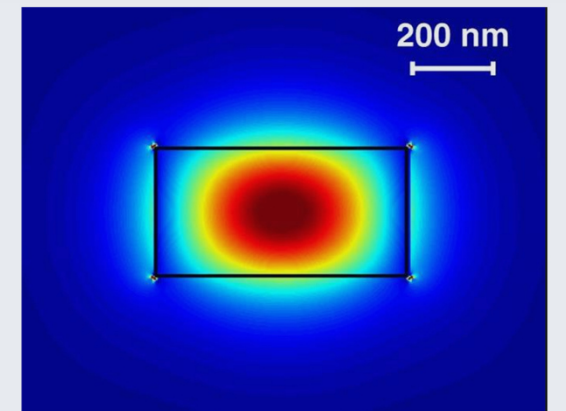
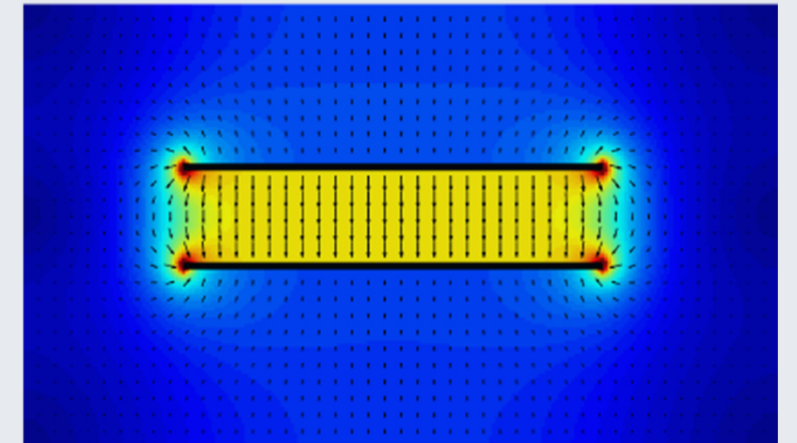
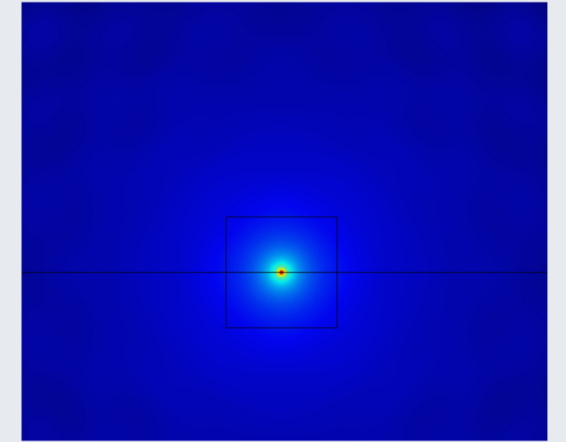
## Domain & Mesh



## Boundary Condition



## Source



# General Workflow

- Model Wizard!
- Select Geometry type
  - 2D
- Select Physics
  - Either emw or ewfd → they have the same functionality

# General Workflow

The screenshot displays the COMSOL Multiphysics software interface, illustrating the general workflow for setting up an electromagnetic simulation. The interface is divided into several key areas:

- Home Tab (Top):** Contains various toolbars for model building, including 'Application Builder', 'Component 1', 'Add Component', 'Build All', 'Add Material', 'Electromagnetic Waves, Frequency Domain', 'Add Physics', 'Build Mesh', 'Mesh 1', 'Compute', 'Study 1', 'Add Study', and 'Select P Group'.
- Model Builder (Left):** Shows a hierarchical tree structure of the model. Key elements are highlighted with colored circles:
  - Geometry 1:** Circled in red, indicating the geometry definition step.
  - Materials:** Circled in green, indicating the material assignment step.
  - Electromagnetic Waves, Frequency Domain (emw):** Circled in blue, indicating the physics selection step.
  - Mesh 1:** Circled in pink, indicating the meshing step.
- Settings (Right):** Shows the configuration for the selected 'Geometry 1' object. Key settings include:
  - Label:** Geometry 1
  - Units:** Length unit is set to 'm' and Angular unit is set to 'Degrees'.
  - Advanced:** Default relative repair tolerance is set to '1E-6' and 'Automatic rebuild' is checked.

# Starting COMSOL

- Every Student has access to Linux Server

## Connection to Linux Server

1) Download and install Cisco Anyconnect: <https://ethz.ch/content/dam/ethz/special-interest/hest/isg-hest-dam/documents/pdf/vpn-de.pdf>

Available for MS,MAC & Linux

2) Connect to ETH network using Ciso Anyconnect

3) Connect to Linux Server

- Linux & Mac: using terminal type «ssh [\\$username@itet-ief-l0.ethz.ch](mailto:$username@itet-ief-l0.ethz.ch) -X»
- Windows, Linux, Max: using any remote desktop with xserver capability (e.g. Remmina)

4) Enter Username and Password

5) Start COMSOL

type: «comsol &» into the terminal