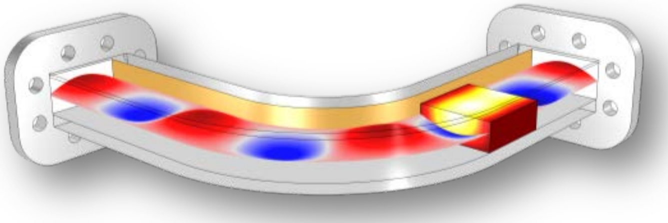
Simulating MRI Heating of Medical Implants

Walter Frei
Applications Engineer
COMSOL

Kyle C. Koppenhoefer
Principal
AltaSim Technologies

Agenda

- Simulating with COMSOL Multiphysics®
- RF heating: Coupling electromagnetics with a heat transfer
- Simulating heating of medical Implants in an MRI scanner
- Live Demo: RF heating of an MRI coil
- Q&A Session
- How To
  - Try COMSOL Multiphysics
  - Contact Us



RF heating of a dielectric block inside a waveguide

Product Suite – COMSOL® 5.1

COMSOL Multiphysics®

COMSOL Server™

ELECTRICAL
- AC/DC Module
- RF Module
- Wave Optics Module
- Ray Optics Module
- MEMS Module
- Plasma Module
- Semiconductor Module

MECHANICAL
- Heat Transfer Module
- Structural Mechanics Module
- Nonlinear Structural Materials Module
- Geomechanics Module
- Fatigue Module
- Multibody Dynamics Module
- Acoustics Module

FLUID
- CFD Module
- Mixer Module
- Microfluidics Module
- Subsurface Flow Module
- Pipe Flow Module
- Molecular Flow Module

CHEMICAL
- Chemical Reaction Engineering Module
- Batteries & Fuel Cells Module
- Electrodeposition Module
- Corrosion Module
- Electrochemistry Module

MULTIPURPOSE
- Optimization Module
- Material Library
- Particle Tracing Module

INTERFACING
- LiveLink™ *for* MATLAB®
- CAD Import Module
- ECAD Import Module
- LiveLink™ *for* Inventor®
- LiveLink™ *for* Revit®
- LiveLink™ *for* PTC® Pro/ENGINEER®
- File Import *for* CATIA® V5
- LiveLink™ *for* Excel®
- Design Module
- LiveLink™ *for* SOLIDWORKS®
- LiveLink™ *for* AutoCAD®
- LiveLink™ *for* PTC® Creo® Parametric™
- LiveLink™ *for* Solid Edge®

A Complete Simulation Environment

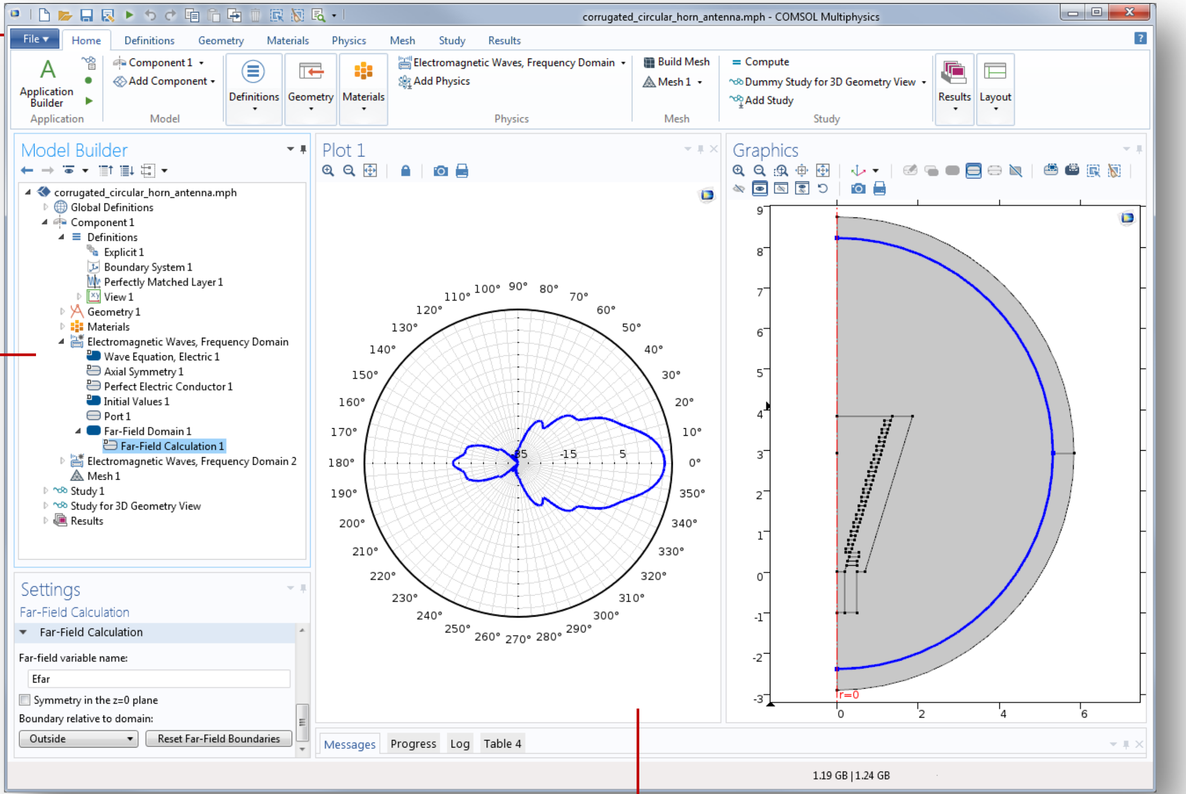
COMSOL Desktop®

Straightforward to use, the Desktop gives insight and full control over the modeling process

Model Builder

Provides instant access to any of the model settings

- CAD/Geometry
- Materials
- Physics
- Mesh
- Solve
- Results



Far-Field Calculation

Far-field variable name: Efar

☐ Symmetry in the z=0 plane

Boundary relative to domain: Outside

Graphics Window

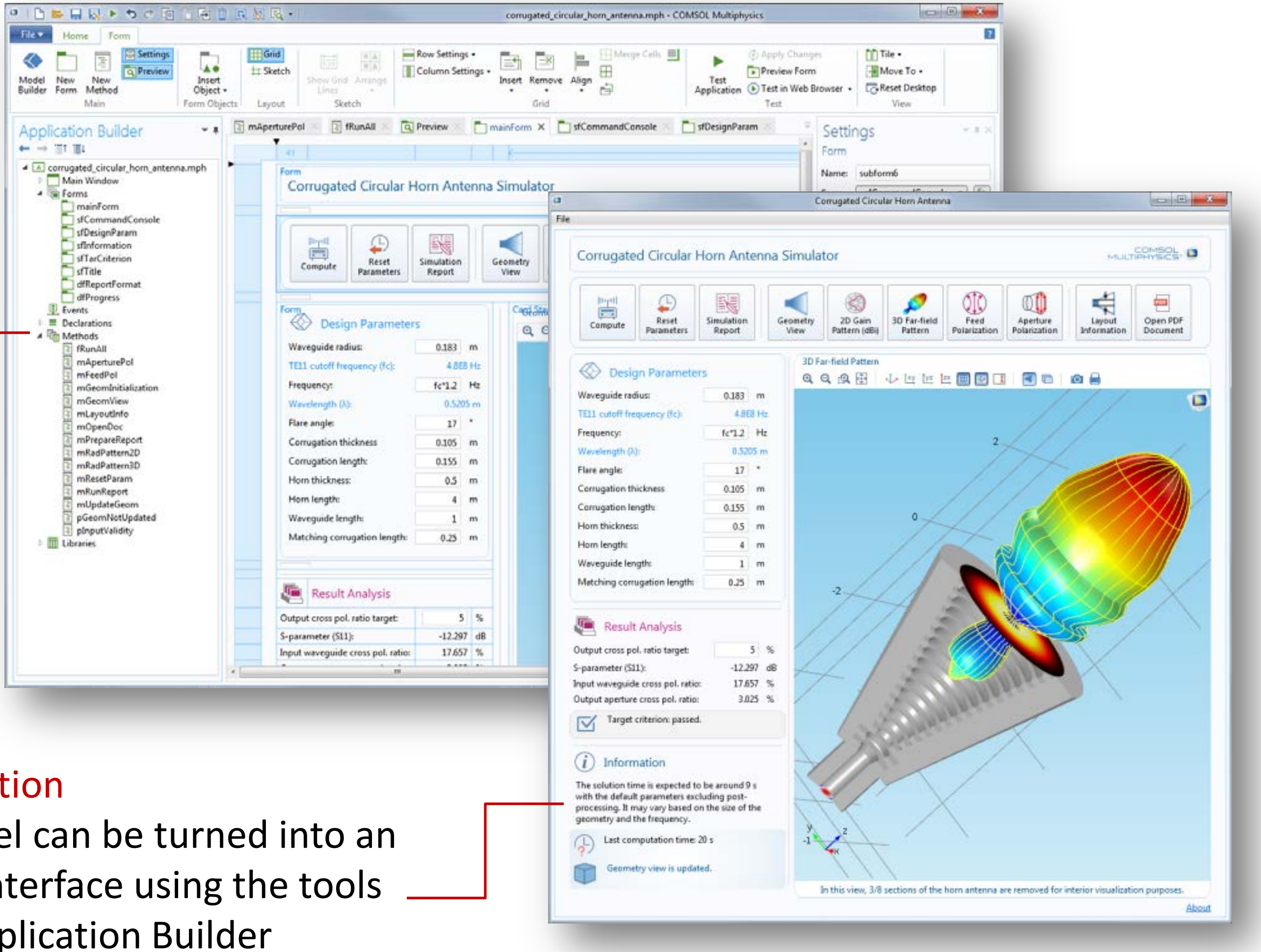
Ultrafast graphic presentation, stunning visualization

Application Design Tools

Application Builder

Provides all the tools needed to build and run simulation apps

- Form Editor
- Method Editor



Design Parameters

Waveguide radius: 0.183 m

TE11 cutoff frequency (fc): 4.8E8 Hz

Frequency: fc*1.2 Hz

Wavelength (λ): 0.5205 m

Flare angle: 17 °

Corrugation thickness: 0.105 m

Corrugation length: 0.155 m

Horn thickness: 0.5 m

Horn length: 4 m

Waveguide length: 1 m

Matching corrugation length: 0.25 m

Result Analysis

Output cross pol. ratio target: 5 %

S-parameter (S11): -12.297 dB

Input waveguide cross pol. ratio: 17.657 %

Output aperture cross pol. ratio: 3.025 %

☑ Target criterion: passed.

Information

The solution time is expected to be around 9 s with the default parameters excluding post-processing. It may vary based on the size of the geometry and the frequency.

Last computation time: 20 s

Geometry view is updated.

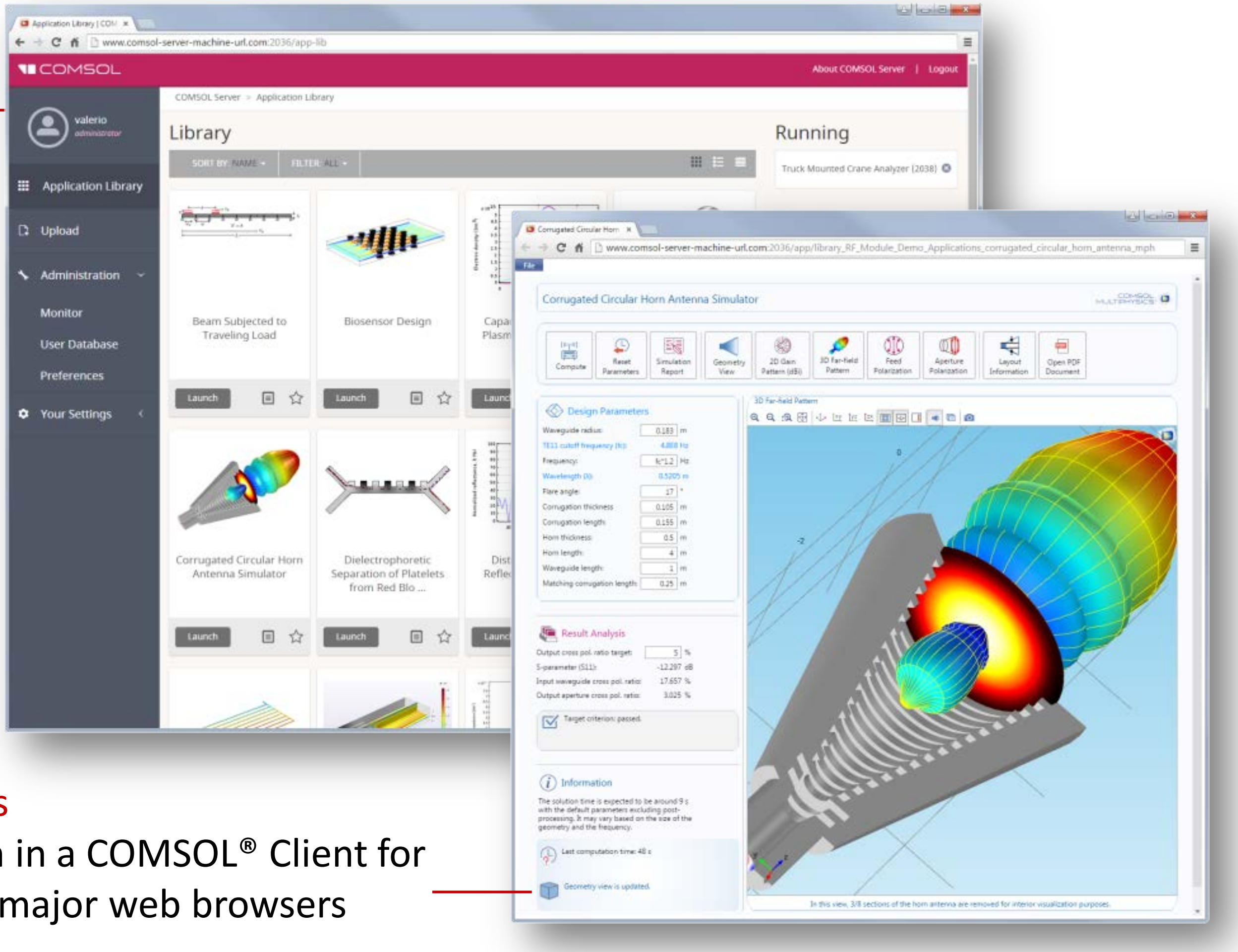
In this view, 3/8 sections of the horn antenna are removed for interior visualization purposes.

Simulation Application

Any COMSOL model can be turned into an app with its own interface using the tools provided in the Application Builder

Run Applications

COMSOL Server™
It’s is the engine for running COMSOL apps and the hub for controlling their deployment, distribution, and use



Waveguide radius: 0.183 m
TE11 cutoff frequency (fc): 4.808 Hz
Frequency: fc*1.2 Hz
Wavelength (λ): 0.5205 m
Flare angle: 17 °
Corrugation thickness: 0.105 m
Corrugation length: 0.155 m
Horn thickness: 0.5 m
Horn length: 4 m
Waveguide length: 1 m
Matching corrugation length: 0.25 m

Output cross pol. ratio target: 5 %
S-parameter (S11): -12.297 dB
Input waveguide cross pol. ratio: 17.657 %
Output aperture cross pol. ratio: 3.025 %

Target criterion: passed.

The solution time is expected to be around 9 s with the default parameters excluding post-processing. It may vary based on the size of the geometry and the frequency.

Last computation time: 48 s

Geometry view is updated.

In this view, 3/8 sections of the horn antenna are removed for interior visualization purposes.

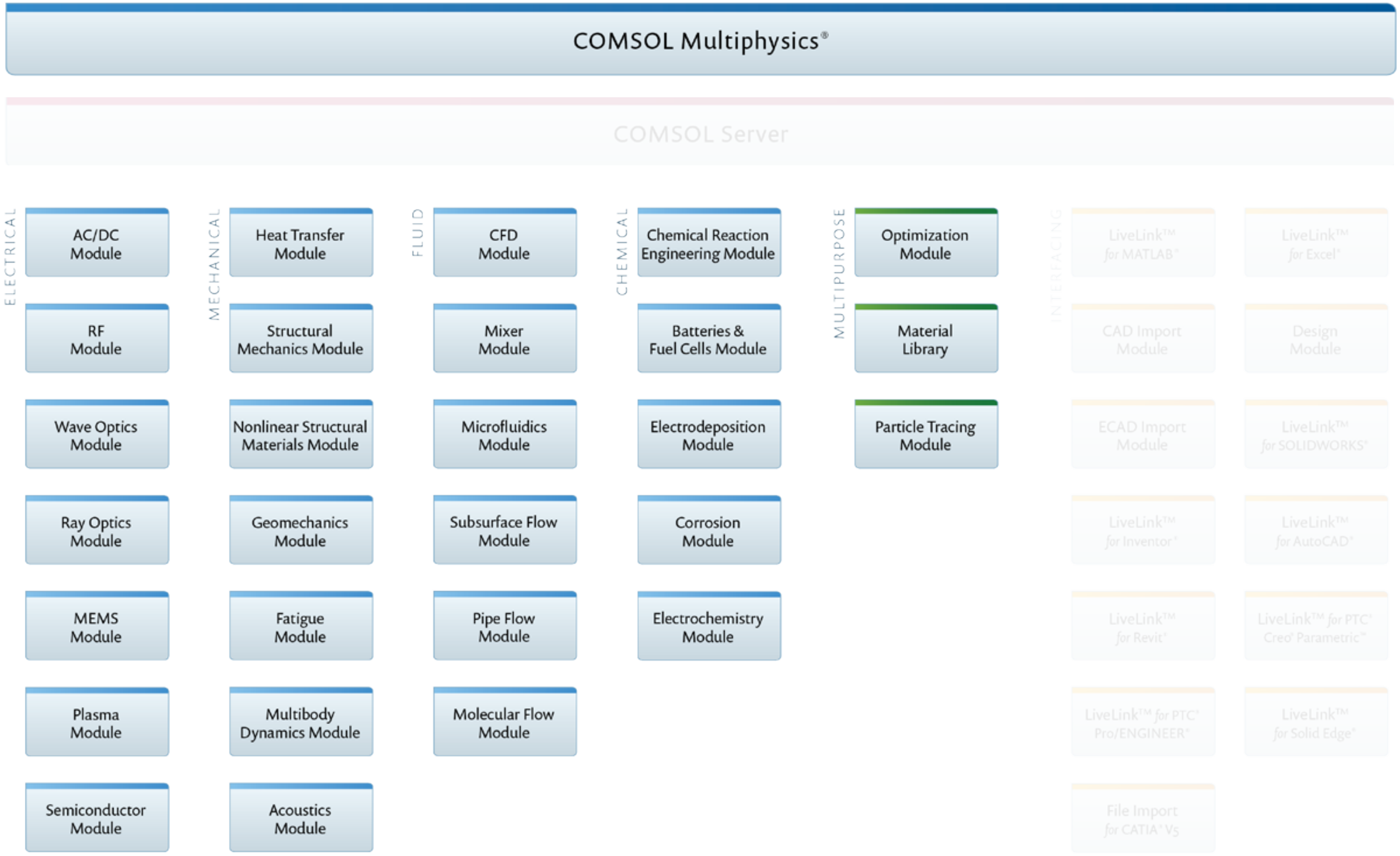
Simulation Apps
They can be run in a COMSOL® Client for Windows® and major web browsers

Poll Question

Are you currently simulating both RF fields and temperature distributions in the same model?

- Yes, I'm simulating them in the same software.
- Yes, I'm simulating them in different software.
- No, I'm simulating them individually.

Physics Modeling Products



COMSOL Multiphysics®

COMSOL Server

ELECTRICAL
- AC/DC Module
- RF Module
- Wave Optics Module
- Ray Optics Module
- MEMS Module
- Plasma Module
- Semiconductor Module

MECHANICAL
- Heat Transfer Module
- Structural Mechanics Module
- Nonlinear Structural Materials Module
- Geomechanics Module
- Fatigue Module
- Multibody Dynamics Module
- Acoustics Module

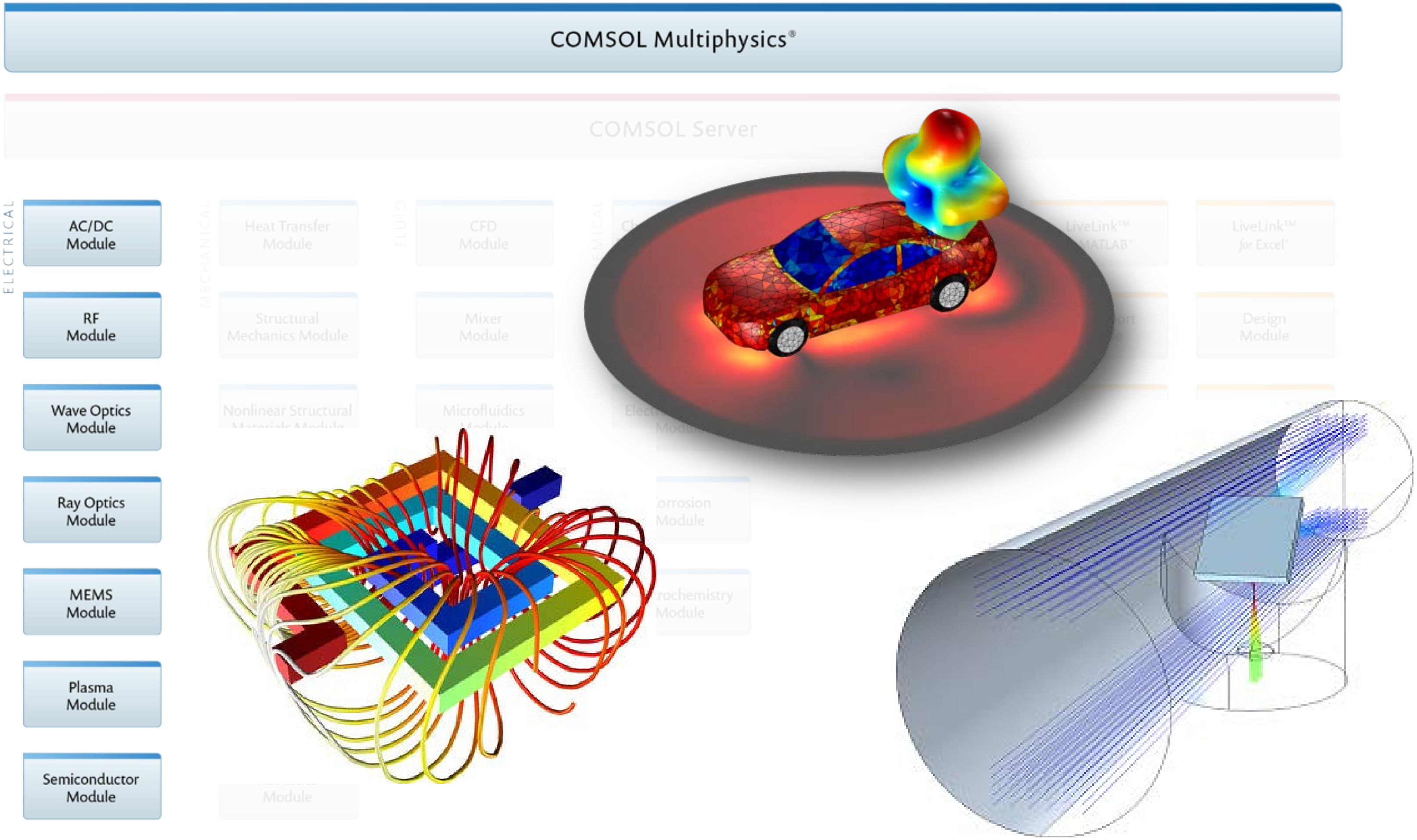
FLUID
- CFD Module
- Mixer Module
- Microfluidics Module
- Subsurface Flow Module
- Pipe Flow Module
- Molecular Flow Module

CHEMICAL
- Chemical Reaction Engineering Module
- Batteries & Fuel Cells Module
- Electrodeposition Module
- Corrosion Module
- Electrochemistry Module

MULTIPURPOSE
- Optimization Module
- Material Library
- Particle Tracing Module

INTERFACING
- LiveLink™ *for* MATLAB®
- LiveLink™ *for* Excel®
- CAD Import Module
- Design Module
- ECAD Import Module
- LiveLink™ *for* SOLIDWORKS®
- LiveLink™ *for* Inventor®
- LiveLink™ *for* AutoCAD®
- LiveLink™ *for* Revit®
- LiveLink™ *for* PTC® Creo® Parametric™
- LiveLink™ *for* PTC® Pro/ENGINEER®
- LiveLink™ *for* Solid Edge®
- File Import *for* CATIA® V5

Electrical Branch



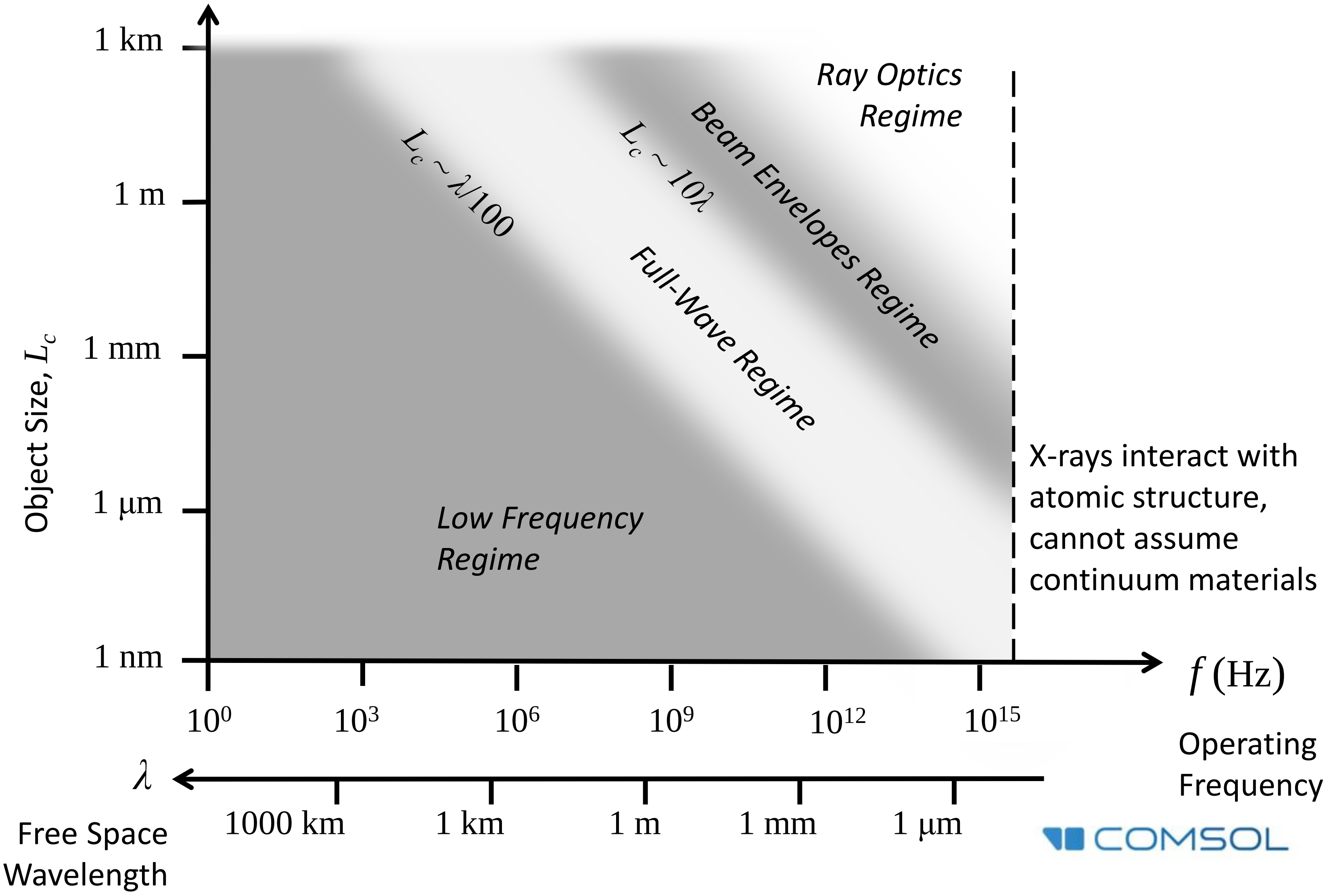
COMSOL Multiphysics®

COMSOL Server

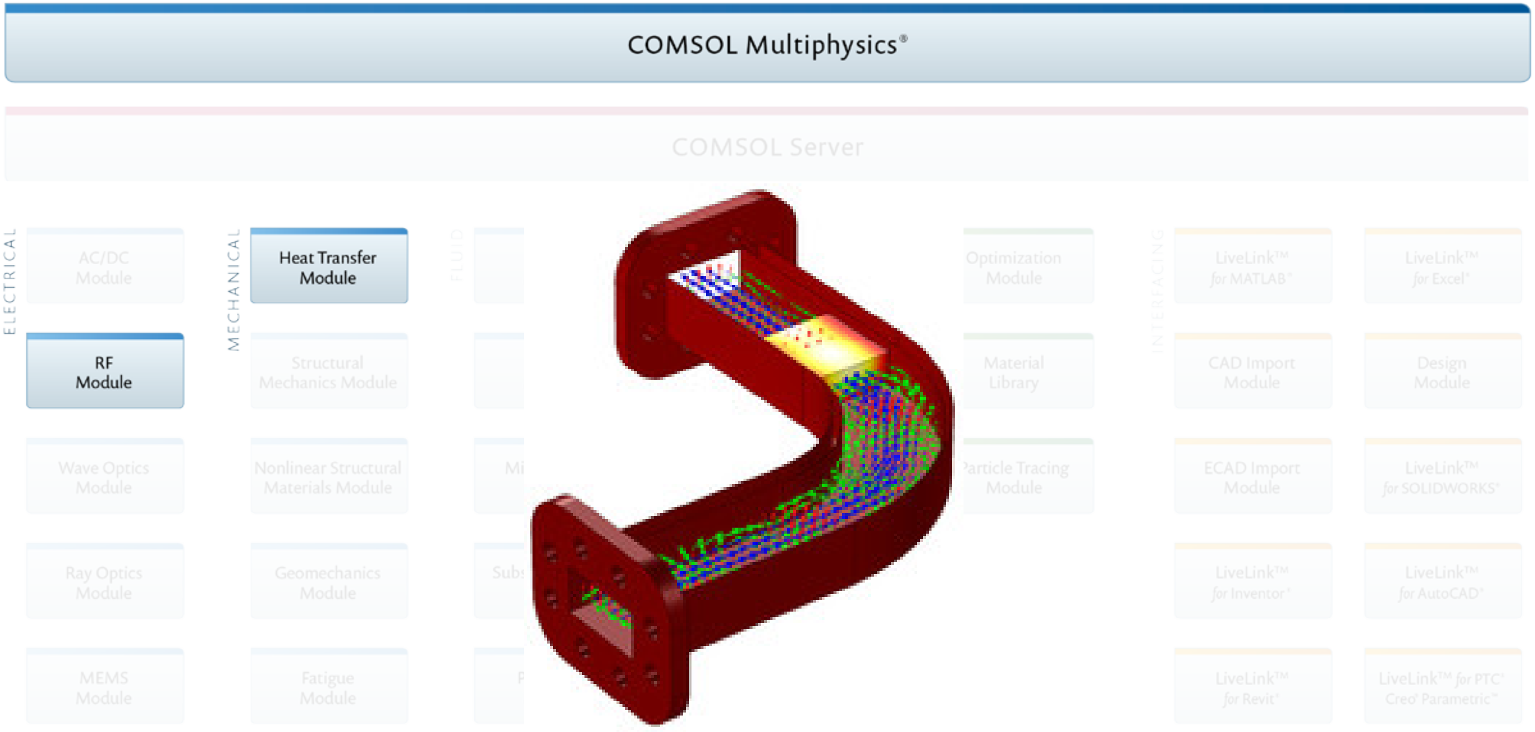
ELECTRICAL

- AC/DC Module
- RF Module
- Wave Optics Module
- Ray Optics Module
- MEMS Module
- Plasma Module
- Semiconductor Module

AC/DC, RF, Wave Optics, or Ray Optics?



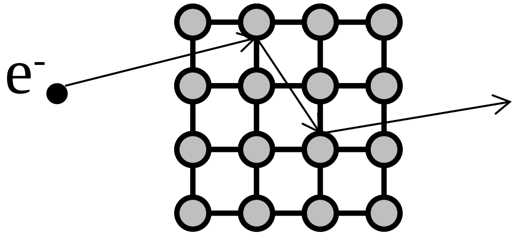
RF Heating is the bidirectional combination of a Full-Wave Electromagnetics Model with a Heat Transfer



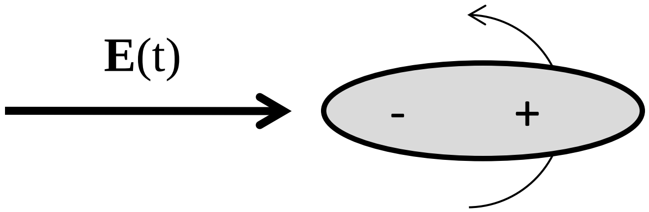
http://www.comsol.com/video/simulating-rf-heating-comsol-multiphysics

http://www.comsol.com/model/rf-heating-6078

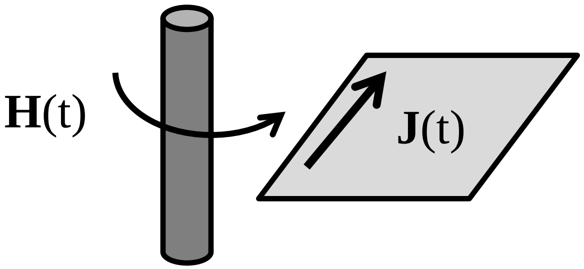
RF heating occurs when energy is transferred (lost) from the electromagnetic fields into heat energy



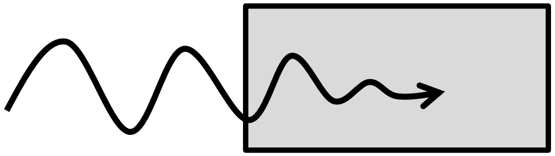
Conduction Current Losses
Electrons moving through a conductor lose energy



Displacement Current Losses
Dipolar molecules rotate in time varying electric field

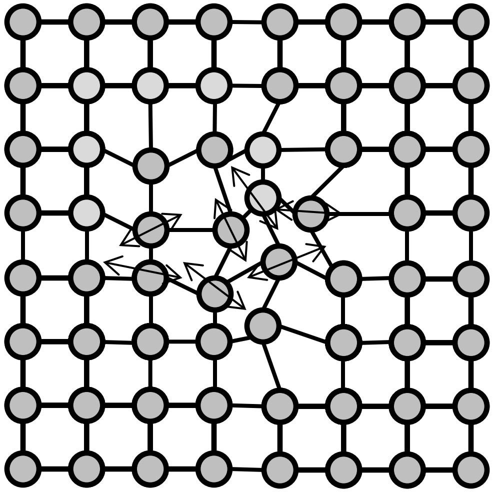


Induction Current Losses
Time varying magnetic fields induce currents in a conductor



An electromagnetic wave induces all of the above

The electrons and molecules move randomly in response to electric and magnetic fields



Heat is a measure of the volume averaged energy of these random vibrations

Temperature is a measure of the average magnitude of these vibrations

An ***Electromagnetic Heating*** model computes the rise in ***Temperature*** due to the transfer of energy from the electromagnetic fields into ***Heat***

COMSOL solves Maxwell’s equations to compute the electromagnetic energy, as well as the heat losses into the material

$$\nabla \times \left(\mu_r^{-1} \nabla \times \mathbf{E}\right) - k_0^2 \left(\varepsilon_r - j\sigma / \omega\varepsilon_0\right)\mathbf{E} = \mathbf{0}$$

Frequency domain form of Maxwell’s equations describing the electric fields inside of the domain, at a known excitation frequency

COMSOL solves Maxwell's equations to compute the electromagnetic energy, as well as the heat losses into the material

Electric Field

$$\nabla \times \left(\mu_r^{-1} \nabla \times \mathbf{E}\right) - k_0^2 \left(\varepsilon_r - j\sigma / \omega\varepsilon_0\right)\mathbf{E} = \mathbf{0}$$

COMSOL solves Maxwell’s equations to compute the electromagnetic energy, as well as the heat losses into the material

Electric Field

$$\nabla \times \left(\mu_r^{-1} \nabla \times \mathbf{E}\right) - k_0^2 \left(\varepsilon_r - j\sigma / \omega\varepsilon_0\right)\mathbf{E} = \mathbf{0}$$

Vacuum Permittivity

Wavevector in Free Space

Excitation Frequency

COMSOL solves Maxwell’s equations to compute the electromagnetic energy, as well as the heat losses into the material

Electric Field

$$\nabla \times \left(\mu_r^{-1} \nabla \times \mathbf{E}\right) - k_0^2 \left(\varepsilon_r - j\sigma / \omega\varepsilon_0\right)\mathbf{E} = \mathbf{0}$$

Relative Permeability

Vacuum Permittivity

Wavevector in Free Space

Excitation Frequency

Relative Permittivity

Electric Conductivity

Material properties needed for analysis, and what they mean

Conductivity, σ, relates the current flow to the applied electric field: $\mathbf{J} = \sigma\mathbf{E}$

Relative Permittivity, ε_r, relates the displacement field to the electric field: $\mathbf{D} = \varepsilon_r \varepsilon_0 \mathbf{E}$

Relative Permeability, μ_r, relates the magnetic flux to the magnetic field: $\mathbf{B} = \mu_r \mu_0 \mathbf{H}$

To model electromagnetic energy being converted into heat, introduce a complex-valued term:

$$\nabla \times \left(\mu_r^{-1} \nabla \times \mathbf{E}\right) - k_0^2 \left(\varepsilon_r - j\sigma / \omega\varepsilon_0\right)\mathbf{E} = \mathbf{0}$$

$$\varepsilon_r = \varepsilon_r' - j\varepsilon_r''$$

Common way of modeling losses in dielectric materials

$$\mu_r = \mu_r' - j\mu_r''$$

Mostly applicable for ferrites, with low conductivity

COMSOL offers other material loss models

$$\varepsilon_r = \varepsilon_r'\left(1 - j\tan\delta\right)$$

Same idea, but different way of describing material loss, assumes zero conductivity

$$\varepsilon_r = \left(n - jk\right)^2$$

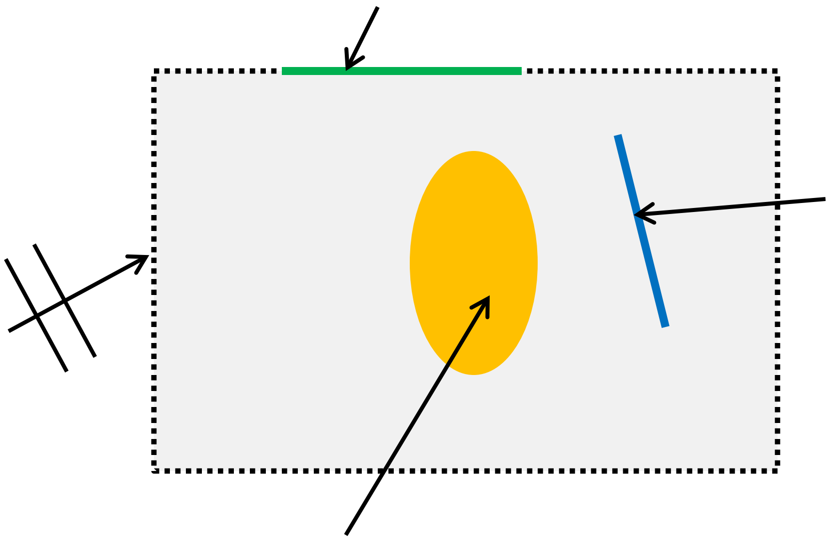
Refractive index, with real, n, and imaginary, k, components. Assumes $\sigma = 0$ and $\mu_r = 1$.

Equations of electromagnetic losses

$$Q_{electric} = \frac{1}{2}\mathrm{Re}\left(\sigma\mathbf{E}\cdot\mathbf{E}^{*} - j\omega\varepsilon\mathbf{E}\cdot\mathbf{E}^{*}\right)$$

$$Q_{magnetic} = \frac{1}{2}\mathrm{Re}\left(-\frac{j}{\omega}\mu^{-1}\left(\nabla\times\mathbf{E}\right)\cdot\left(\nabla\times\mathbf{E}\right)^{*}\right)$$

It is also possible to include losses on the external and internal boundaries of a model

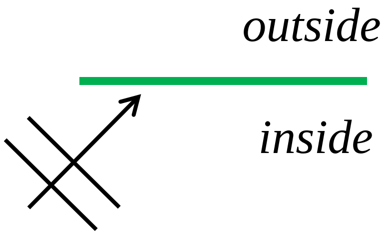


Lossy Exterior Boundary

Lossy
Interior
Boundary

Lossy Domain

Losses on exterior boundaries can be modeled with an Impedance Boundary Condition (IBC)

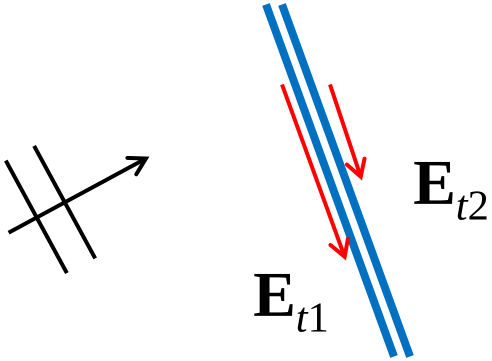


$$\sqrt{\frac{\mu}{\varepsilon - j\frac{\sigma}{\omega}}}\,\mathbf{n}\times\mathbf{H}+\mathbf{E}-(\mathbf{n}\cdot\mathbf{E})\mathbf{n}=\mathbf{0}$$

The IBC is appropriate for the exterior boundaries of the modeling space. It is typically used to describe the boundaries of an object that has a small skin depth relative to the characteristic size of the model.

The IBC is appropriate for modeling objects of high conductivity (metals) or high relative impedance (sea surface) as compared to the modeling domain.

Losses on interior boundaries can be modeled with a Transition Boundary Condition (TBC)



$$\mathbf{J}_{s1} = \frac{(Z_S\mathbf{E}_{t1} - Z_T\mathbf{E}_{t2})}{Z_S^2 - Z_T^2}$$

$$\mathbf{J}_{s2} = \frac{(Z_S\mathbf{E}_{t2} - Z_T\mathbf{E}_{t1})}{Z_S^2 - Z_T^2}$$

$$Z_S = \frac{-j\omega\mu}{k}\frac{1}{\tan(kd)}$$

$$Z_T = \frac{-j\omega\mu}{k}\frac{1}{\sin(kd)}$$

$$k = \omega\sqrt{(\varepsilon + (\sigma/(j\omega)))\mu}$$

The TBC is modeled as having zero thickness, but different electric fields are computed on either side of the boundary.

For the problem to remain numerically well-posed, the boundary should not completely block, or transmit, the fields.

The TBC is appropriate for modeling thin, lossy, films such as anti-reflective coatings.

Putting it together with Heat Transfer

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \left(-k \nabla T \right) = Q$$

Governing equation within the domain

$$\left.\begin{aligned} -n \cdot \left(-k \nabla T \right) &= q \\ T &= T_0 \end{aligned}\right\}$$

Boundary conditions

Poll Question

Are you currently involved in MRI simulation?

- Yes, I'm simulating only the magnetic aspect.
- Yes, I'm simulating both the magnetic and thermal aspects.
- No, I would like to start simulating such an application.

Simulating Heating of Medical Implants in an MRI Scanner

Kyle Koppenhoefer, Ph.D.

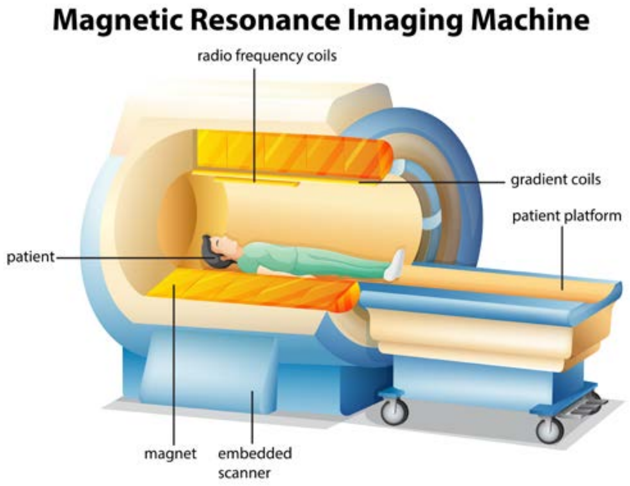
Principal

AltaSim Technologies

Overview

- Motivation for simulating MRI scanning of medical implants
- Previous methods determining MR compatibility of medical implants
  - Experimentation
  - Finite Different Time Domain Solutions
- Multiphysics modeling of MRI heating
  - Full-body modeling
  - Vascular flow effects
- Overview of modeling methodology
  - Modeling of bird cage coil
  - Addition of electromagnetic losses
  - Example of ASTM F2182-11a calibration rod
  - Comparison with experimental data

Motivation



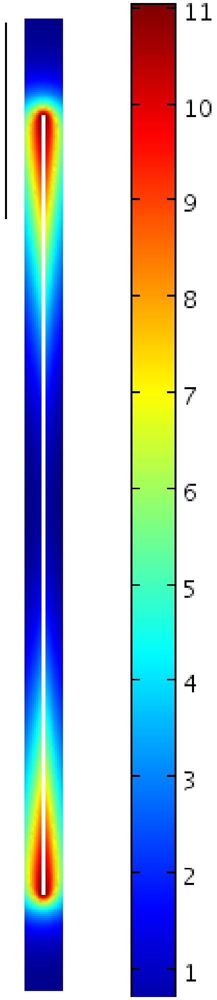
- MRI imaging
  - Static magnetic (1.5 or 3 T)
  - Gradient coil
  - RF coil
- MRI and medical product interactions
  - Interaction with magnetic field
  - Image artifacts
  - Tissue heating

Tissue Heating during MRI

- Time varying RF field induces current in metallic implants
- Induced currents generate local time varying magnetic field
- Induction current losses generated in tissue

$$Q_{electric} = \frac{1}{2}\mathrm{Re}\left(\sigma\mathbf{E}\cdot\mathbf{E}^{*} - j\omega\varepsilon\mathbf{E}\cdot\mathbf{E}^{*}\right)$$

- Tissue exposed to elevated temperature can damage healthy tissue
- Design devices that do not produce this heating in MRI fields

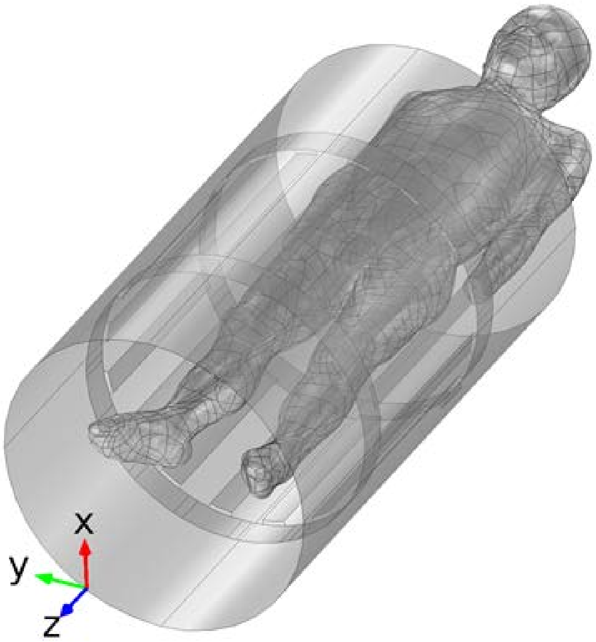


Previous Methods for MR Compatibility

- Experimentation
  - ASTM F2182
  - Requires MRI to conduct testing
  - Gel phantom without convective heat transfer effects
- Finite Different Time Domain (FDTD)
  - Simple to implement
  - Must solve in time domain
  - Highly refined grid is necessary to provide an accurate solution
  - Difficult to represent devices with small features
  - Typically linked to heat transfer via specific absorption rate (SAR)

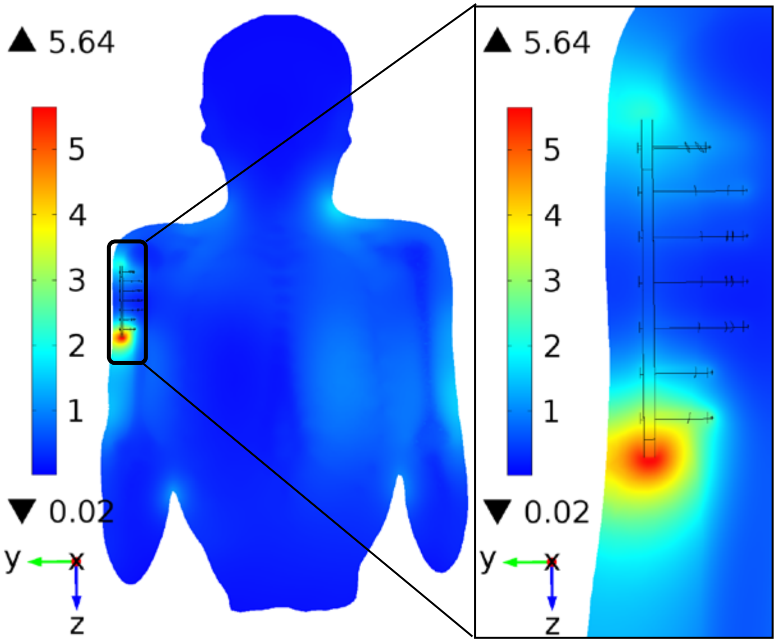
Multiphysics Modeling of MRI Heating

- Finite element based solution of Maxwell Equations
- Frequency domain solution available
- Direct calculation of EM heating
- Inclusion of blood flow



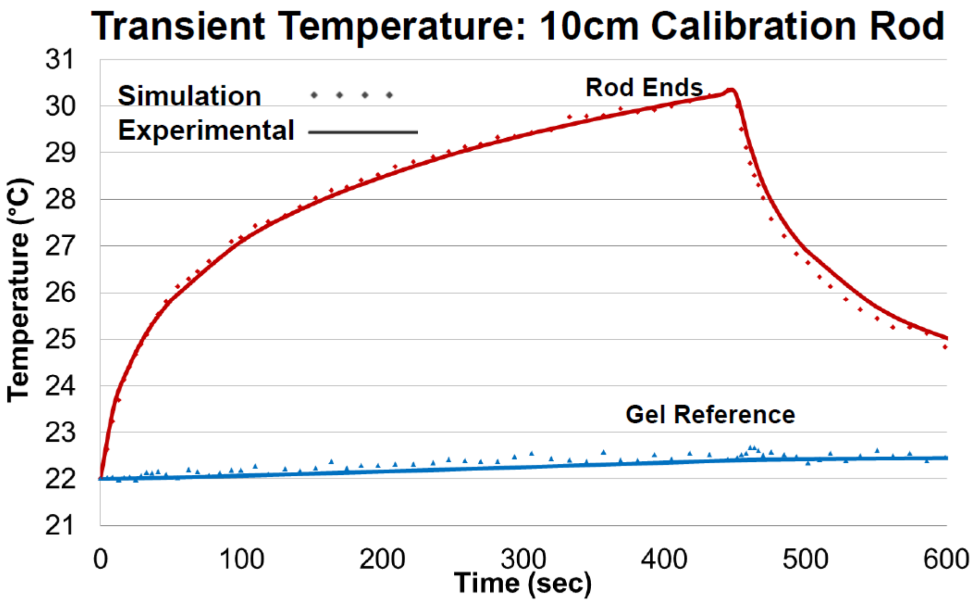
Orthopedic Insert Example

- Generic humerus locking plate
- Model includes body cavity and skeleton
- Temperature rise calculated as 5.6 °C



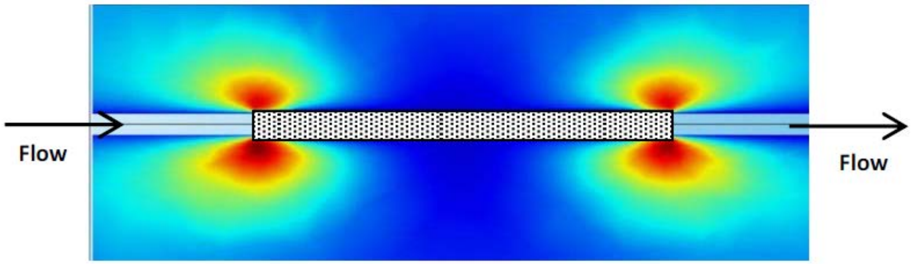
RF Induced Heating During MRI: Evaluation of a Passive Implant in an Anatomical Model using Coupled Multiphysics FEA, Gopal, S., et al. , BMES Conf, May 2015

Comparison with Experimental Data

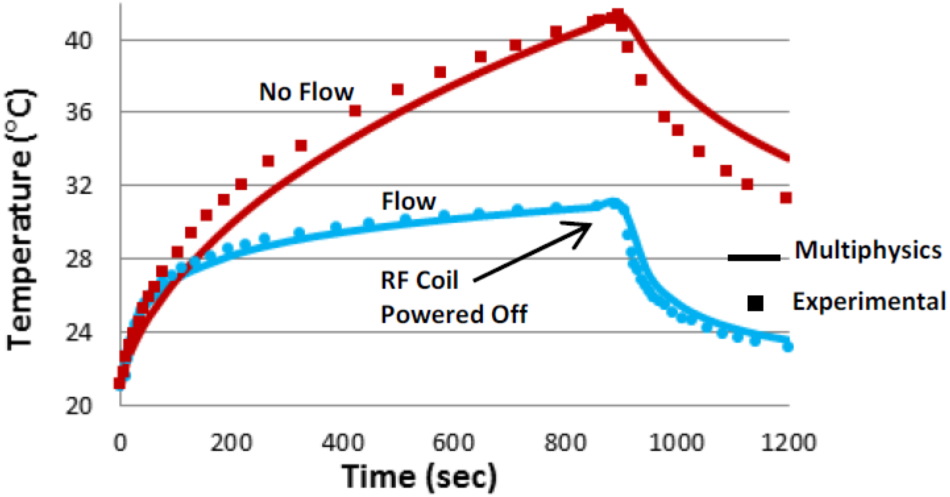


- ASTM F2182 calibration rod – 10 cm, titanium
- Temperature measured at ends of the rod
- Coupled EM-thermal simulation

Vascular Flow Effects



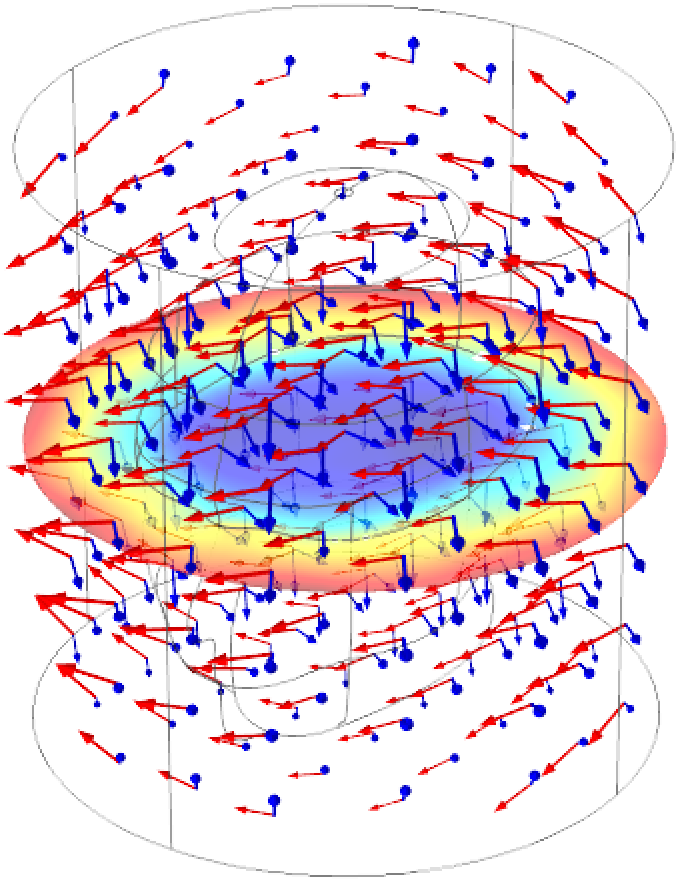
Vascular Flow Effects on RF Heating of Passive Implants: The use of a Flow Modified ASTM F2182 Phantom in a Siemens Tim Trio 3T Scanner, Leewood, A., et al., ISMRM, April 2013



- Titanium implant, 10 cm
- 3T Scanner, whole body SAR of 4 W/kg
- Temperature increase of 20 °C
- Temperature rise of 10 °C with 2 L/min water flow

Live Demo: MRI Coil

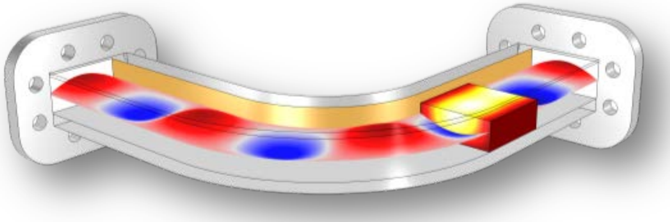
- Develop model of birdcage coil in 3D
- Develop solution in frequency domain
- Include coil capacitors using lumped elements
- Tune magnetic field with capacitors included in model
- Use perfect electrical conductor to represent coil surface and shield
- Quadrature excitation via lumped ports
- Include gel phantom and calibration rod from ASTM F2182-11a
- Calculate temperature rise in gel due to induction current losses



ALTASIM TECHNOLOGIES
REALIZING TOMORROW'S TECHNOLOGY

COMSOL
Certified Consultant

Q&A Session



Product Suite – COMSOL® 5.1

COMSOL Multiphysics®

COMSOL Server™

ELECTRICAL
- AC/DC Module
- RF Module
- Wave Optics Module
- Ray Optics Module
- MEMS Module
- Plasma Module
- Semiconductor Module

MECHANICAL
- Heat Transfer Module
- Structural Mechanics Module
- Nonlinear Structural Materials Module
- Geomechanics Module
- Fatigue Module
- Multibody Dynamics Module
- Acoustics Module

FLUID
- CFD Module
- Mixer Module
- Microfluidics Module
- Subsurface Flow Module
- Pipe Flow Module
- Molecular Flow Module

CHEMICAL
- Chemical Reaction Engineering Module
- Batteries & Fuel Cells Module
- Electrodeposition Module
- Corrosion Module
- Electrochemistry Module

MULTIPURPOSE
- Optimization Module
- Material Library
- Particle Tracing Module

INTERFACING
- LiveLink™ *for* MATLAB®
- LiveLink™ *for* Excel®
- CAD Import Module
- Design Module
- ECAD Import Module
- LiveLink™ *for* SOLIDWORKS®
- LiveLink™ *for* Inventor®
- LiveLink™ *for* AutoCAD®
- LiveLink™ *for* Revit®
- LiveLink™ *for* PTC® Creo® Parametric™
- LiveLink™ *for* PTC® Pro/ENGINEER®
- LiveLink™ *for* Solid Edge®
- File Import *for* CATIA® V5

COMSOL

Try COMSOL Multiphysics®

North America

- Memphis, TN
- Youngstown, OH
- San Mateo, CA
- Charlottesville, VA
- College Station, TX
- Novi, MI
- Minneapolis, MN
- Fayetteville, AR
- Madison, WI

Europe

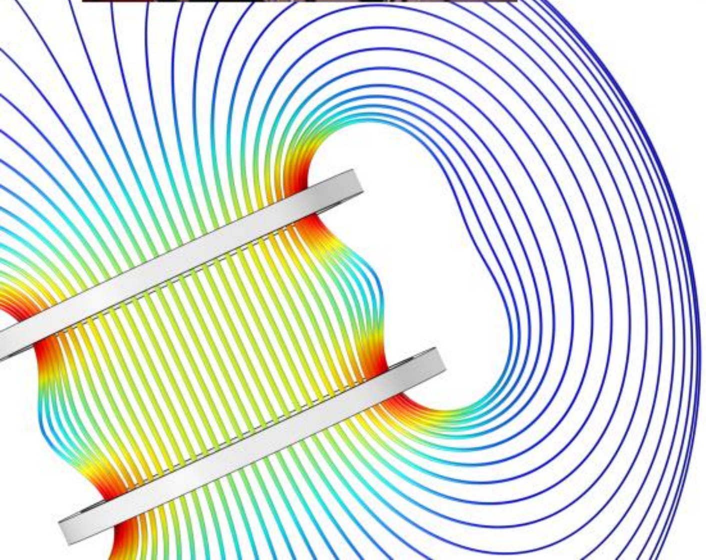
- Kgs. Lyngby, Denmark
- Stockholm, Sweden
- Paris, France
- Bremen, Germany
- Oslo, Norway

Free hands-on workshops

REGISTER TODAY
www.comsol.com/events

COMSOL

COMSOL
CONFERENCE
2015



Join the leaders in Multiphysics Simulation at the COMSOL Conference 2015


Boston
October 7-9


Grenoble
October 14-16


Pune
October 29-30


Beijing
November 5-9


Curitiba
November 5-6


Taipei
November 13


Seoul
November 27


Kuala Lumpur
TBD


Tokyo
December 3-4

REGISTER TODAY!

www.comsol.com/conference

COMSOL

Contact Us

- Questions?
  www.comsol.com/contact

- www.comsol.com
  - User Stories
  - Videos
  - Application Gallery
  - Discussion Forum
  - Blog
  - Product News