Module 05: Multiphysics

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Overview

- Workbench Introduction
- Coupling Capabilities
- Workshop 05.1: Electromagnetic – Mechanical Coupling
- Workshop 05.2: Electromagnetic – Thermal Coupling
- Workshop 05.3: Electromagnetic Eddy Current – Thermal Coupling

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

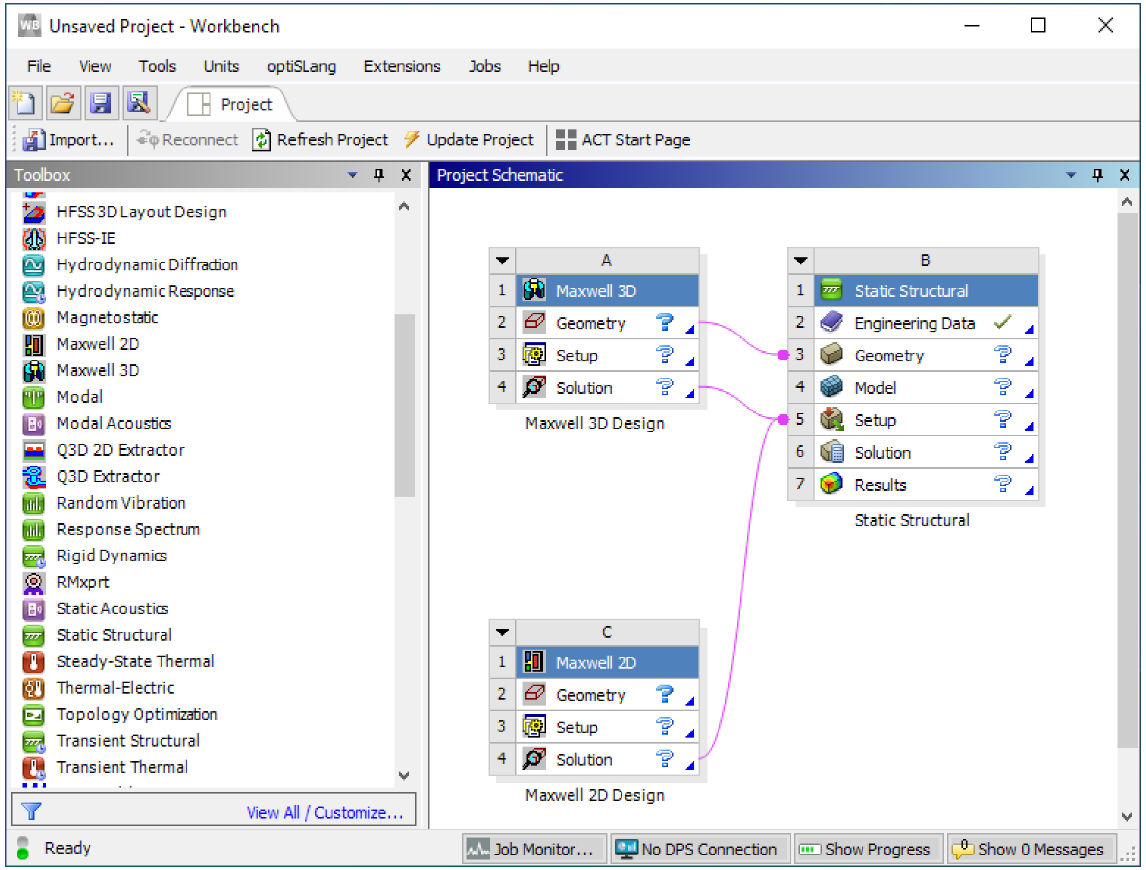
Workbench Introduction

Ansys

ANSYS Workbench

- **ANSYS Workbench is a project-management tool. It can be considered as the top-level interface linking all our software tools**
- **Workbench handles the passing of data between ANSYS Geometry/Mesh/Solver/Postprocessing tools**
- **This greatly helps project management. You do not need worry about the individual files on disk (geometry, mesh etc). Graphically, you can see at-a-glance how a project has been built**

  Because Workbench can manage the individual applications AND pass data between them, it is easy to automatically perform design studies (parametric analyses) for design optimisation

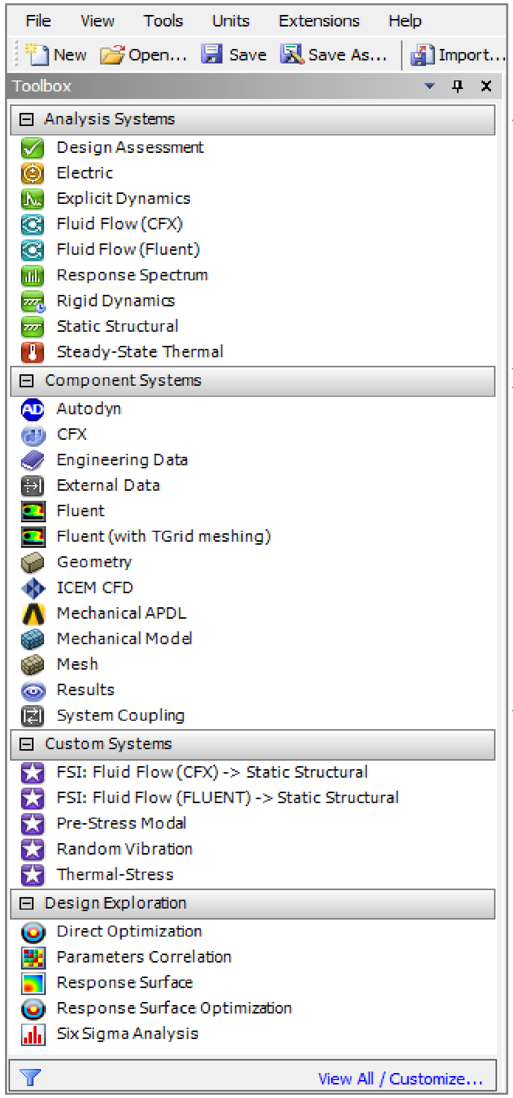


©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Workbench Overview

The options visible in the Toolbox show all the products (systems) you have licenses for.

TIP: If this list appears empty, you have a problem with your licensing!



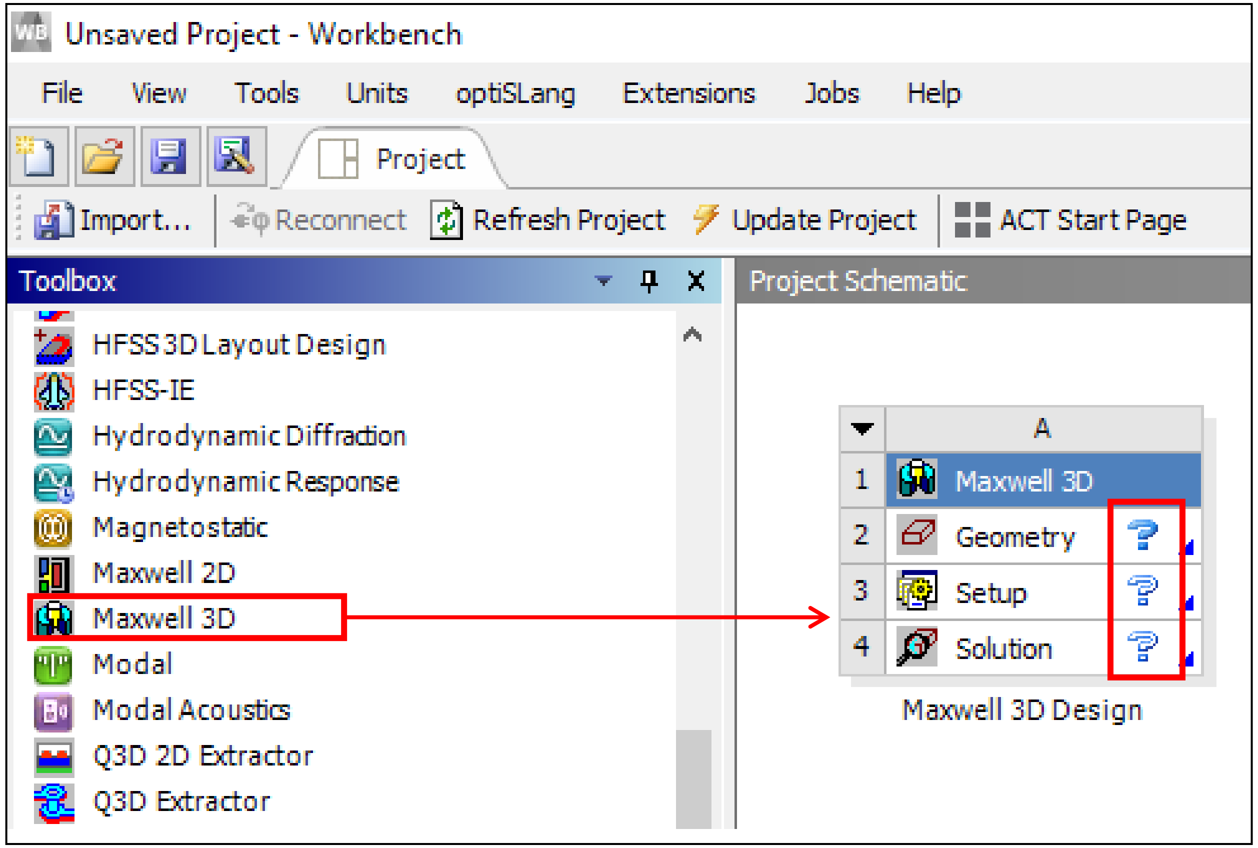
"**Analysis Systems**" are ready-made stencils that include all the individual systems (applications) needed for common analyses *(for example Geometry + Mesh + Solver + Post-Processor)*

"**Component Systems**" are the individual building-blocks for each stage of the analysis

"**Design Exploration**" provides tools for optimising designs and understanding the parametric response.

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Basic Workflow



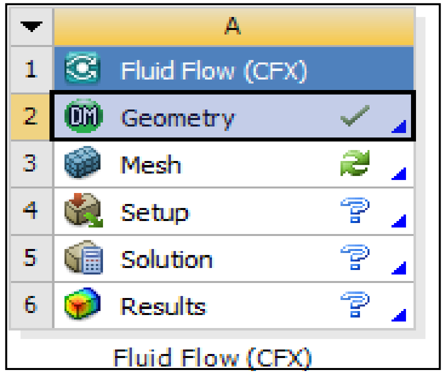
- **Dragging an Analysis System onto the Project Schematic lays out a workflow, comprising all the steps needed for a typical analysis.**
- **Workflow is from top to bottom. As each stage is complete, the icon at the right-hand side changes**

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

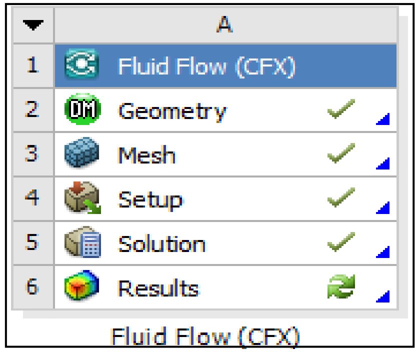
Cell States

As each stage in the model-build is completed, the state of the cell changes.

Icon	Meaning
	Up to Date
	Refresh required. Upstream data has changed
	Update required. Local data has changed
	Unfulfilled. Upstream data does not exist
	Attention Required
	Solving
	Update Failed
	Update Interrupted
	Changes pending (was up-to-date, but upstream data has changed)



Status after creating Geometry in A2, not yet opened mesh in A3

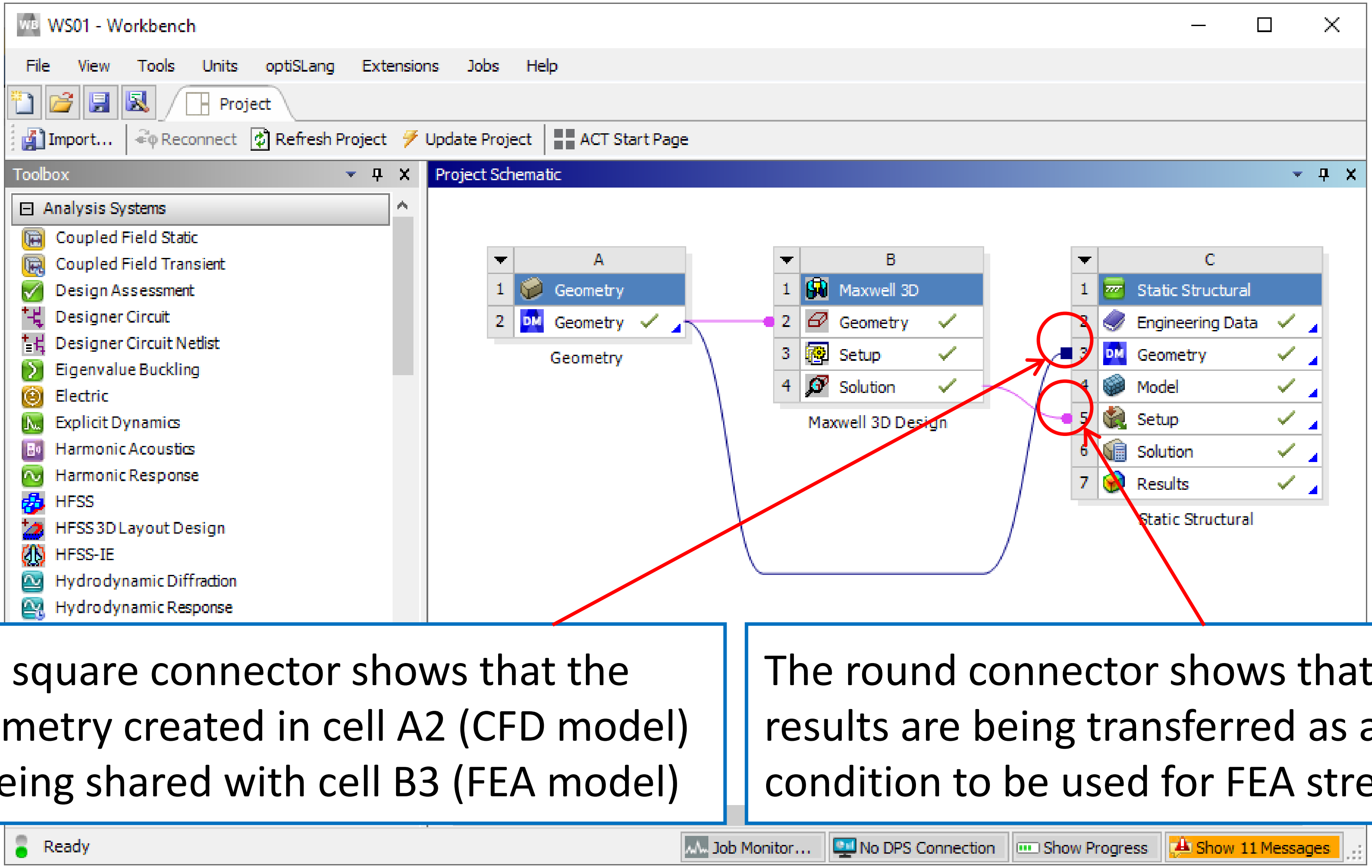


Status after model has converged, waiting for post-processing

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Sharing Data between Different Solvers

- **Workbench can be used to transfer data between solvers. In this 1-way FSI (Fluid-Structure-Interaction) example, we transfer the loads from a Fluent CFD simulation over to a Mechanical system to perform a stress analysis**



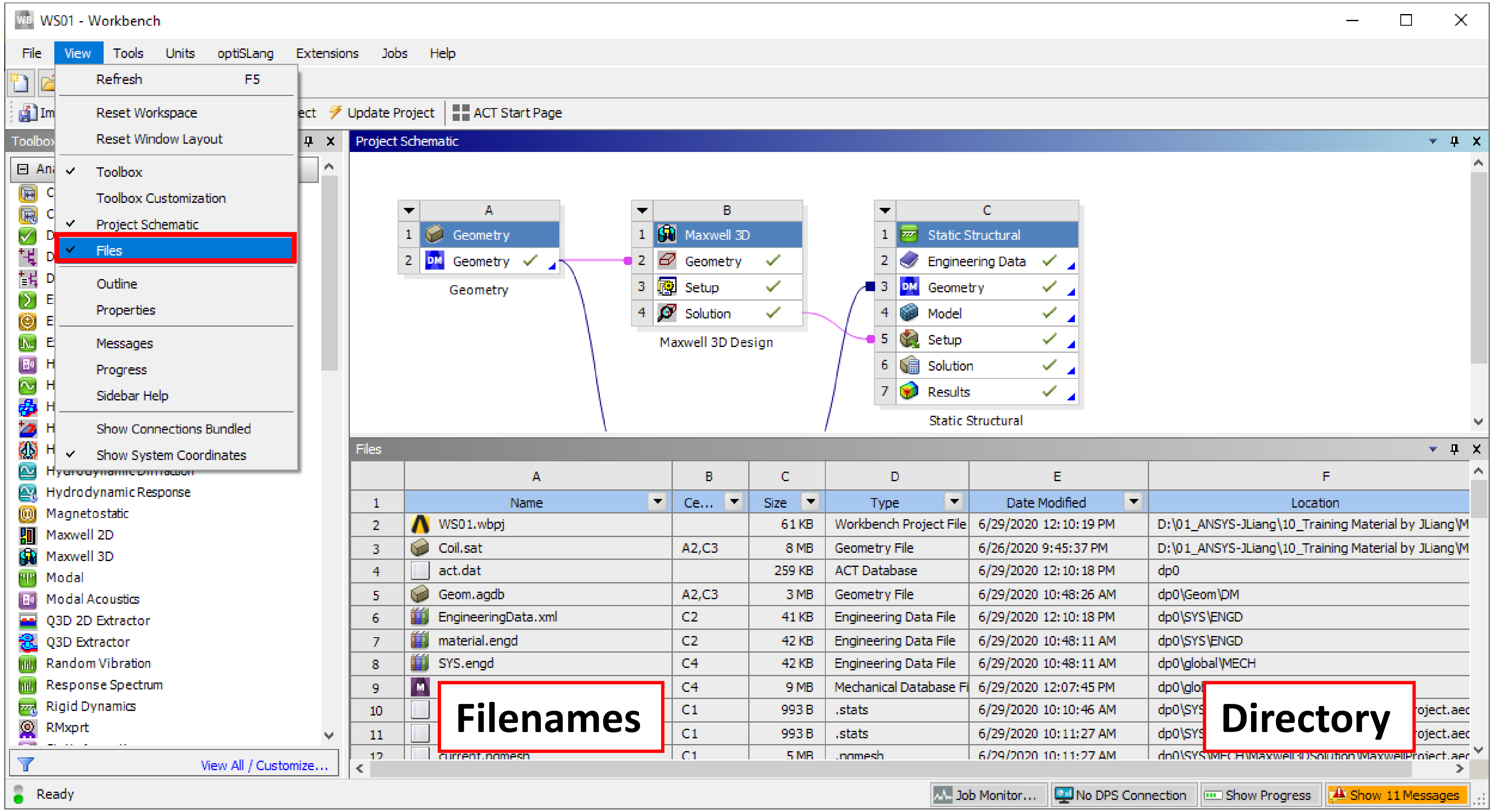
The square connector shows that the geometry created in cell A2 (CFD model) is being shared with cell B3 (FEA model)

The round connector shows that the CFD results are being transferred as a Setup (input) condition to be used for FEA stress analysis

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

File Location on the Disk

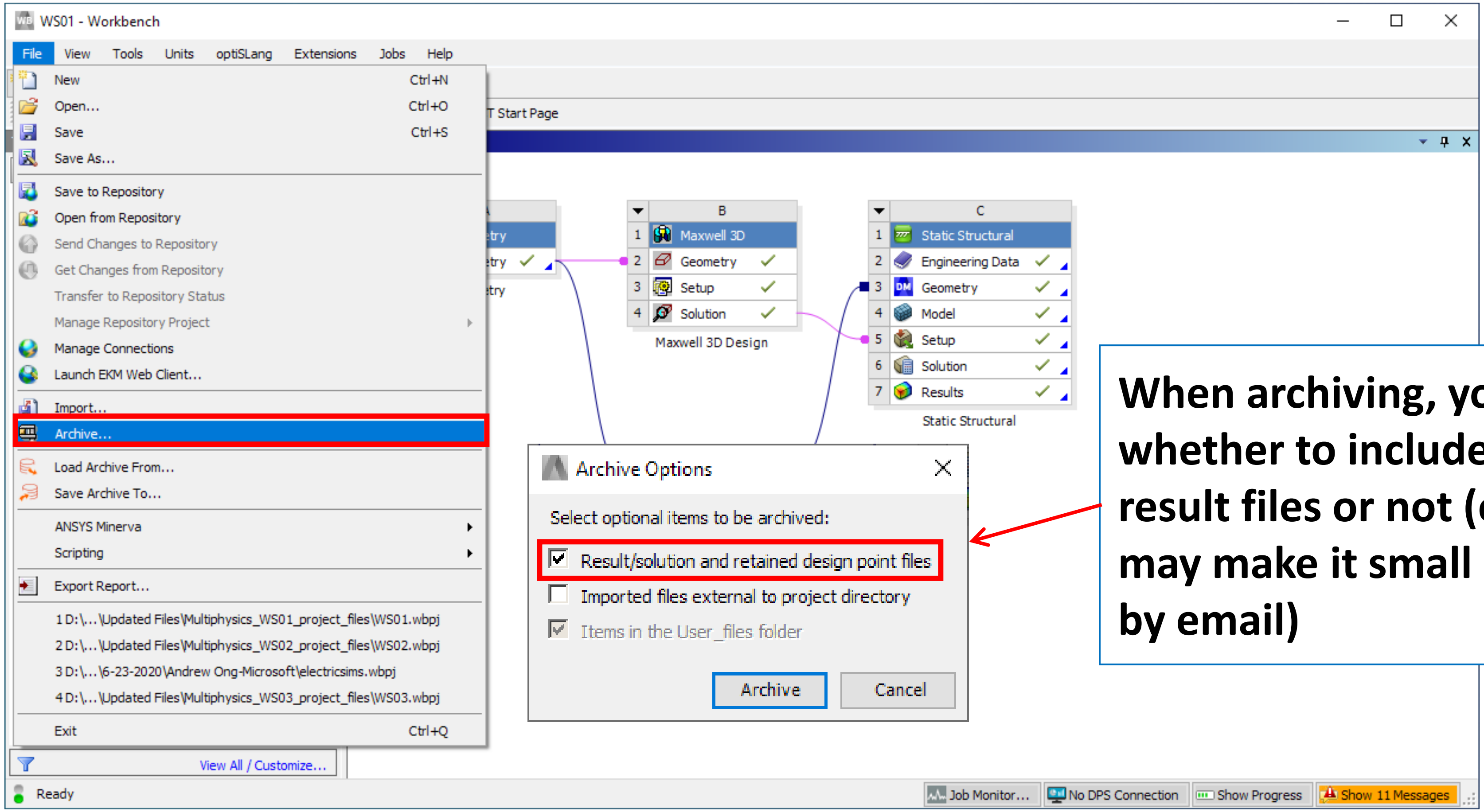
Should you need to identify the individual files on your disk for each stage of the project, these can be found by enabling *View → Files*. The resulting table will cross-reference the directory and filename with the project cells



Ansys

Use of Archive / Restore

The workbench project comprises many files and directories. If you need to either archive the project, or bundle it to send to us for a Technical Support query, use the 'Archive' tool. This generates a single zipfile of the entire project.

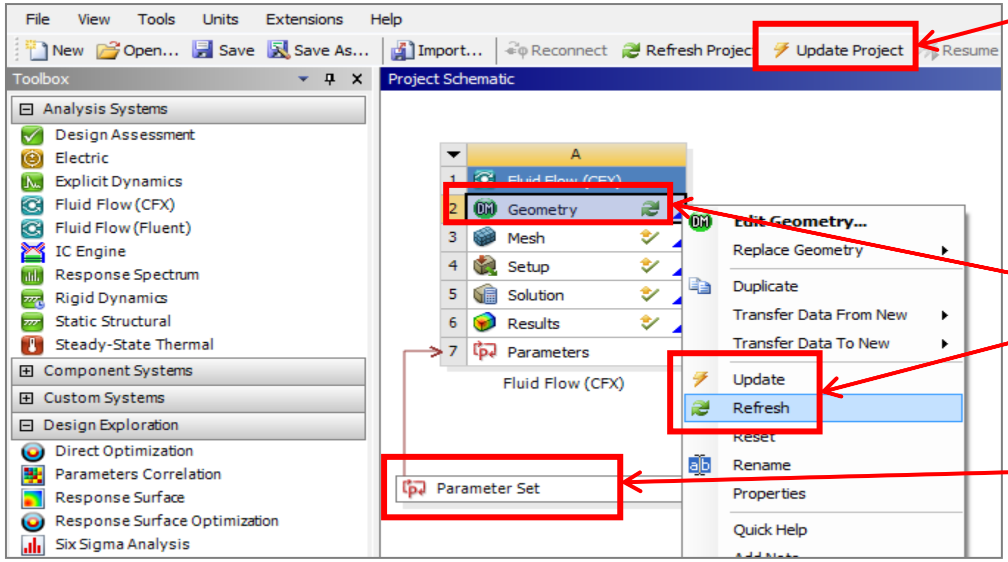


When archiving, you can choose whether to include the computed result files or not (omitting these may make it small enough to send by email)

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Working with Parameters / Refresh and Update

- To make changes, you can manually open up a component cell (e.g. geometry)
- Or: most Workbench applications will let you specify key quantities as a parameter. A new object 'Parameter Set' appears on the Project schematic
- You then need to update your model. From Workbench you can choose to then update the entire project, or just a single cell
- Refresh: Reads upstream data, but will not do any lengthy operation like solving or meshing
- Update: Performs both a Refresh, AND generates the new output



2] "Update Project" will then work through each component in turn (geometry > mesh > solver) to compute the new design point.

3] However you may want to update an individual component (eg to preview the new geometry before proceeding). Right-click on a individual cell.

1] Clicking here will let you modify the parameters centrally, without having to open the individual application.

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Workbench Summary

- **ANSYS Workbench** is a convenient way of managing your simulation projects.

- Workbench is used to launch the individual software components, and used to transfer data between them.

- It is easy to see at-a-glance how a model has been built, and determine which files were used for a particular simulation (pairing geometry files to solver runs)

- Workbench also makes it straightforward to perform parametric analyses (without the user needing to manually launch each application in turn), and makes it easy to simulate multi-physics scenarios like fluid-structure interaction.

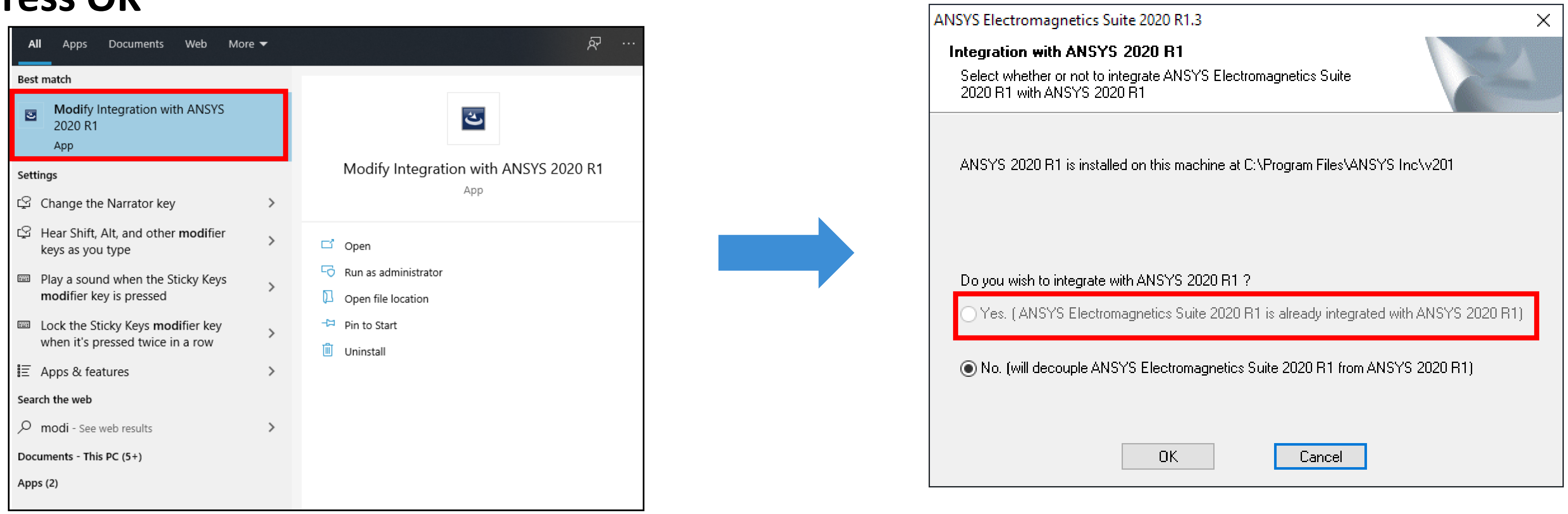
©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Maxwell Integration in the Workbench

Ansys

Maxwell Integration in the Workbench

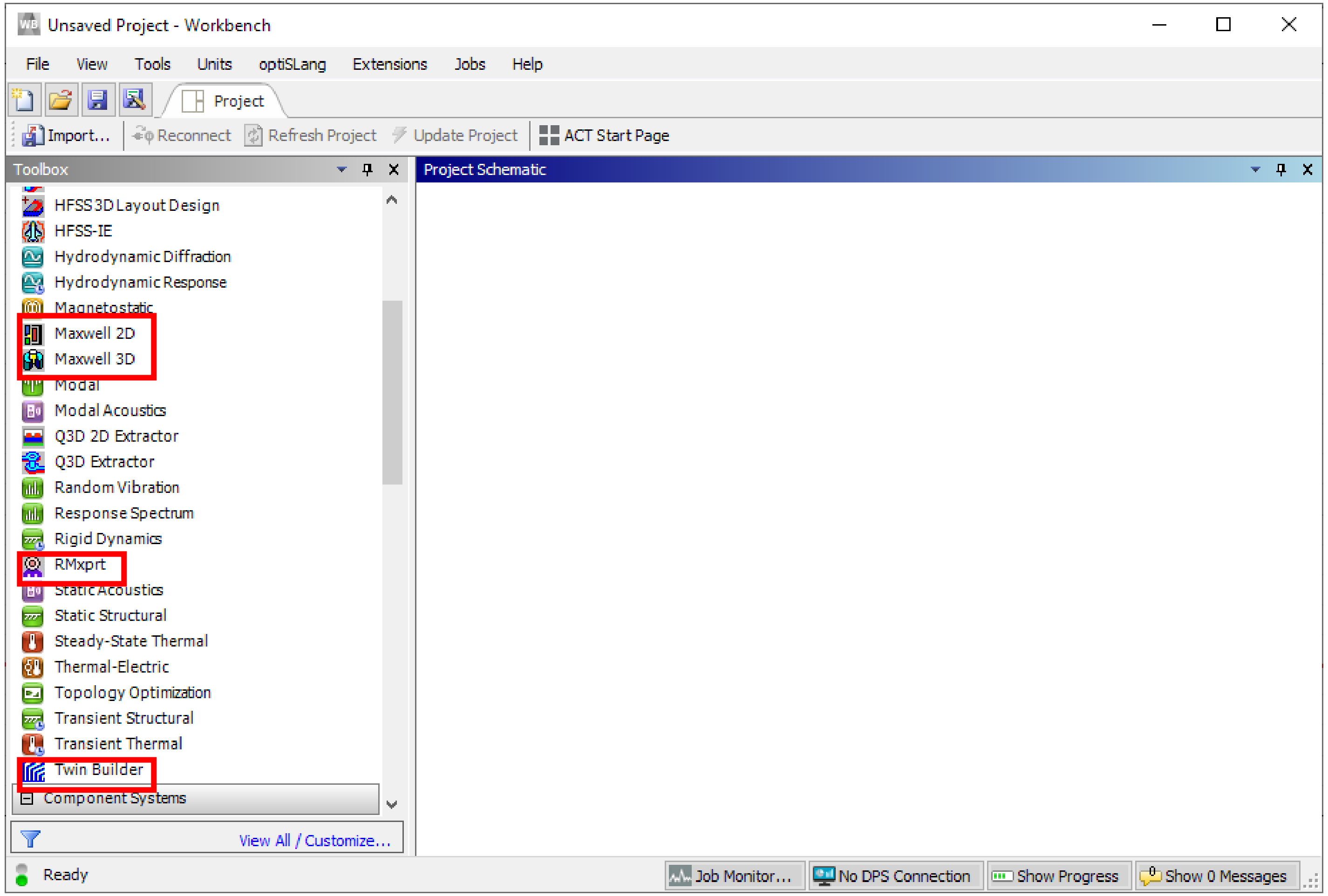
- Integration in the Workbench
  - The Electronics Desktop and therefore Maxwell has to be integrated in the Workbench
  - Ansys Electronics Desktop 2020 R1 has to be integrated in Ansys Workbench 2020 R1
  - From the menu Start of Windows, select All Programs
    - Scroll till finding Ansys Electromagnetics Suite 2020 R1
    - Expand the folder
    - Select “Modify Integration with ANSYS 2020 R1”
    - The installation wizard pops-up, choose “Yes”
    - Press OK



©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

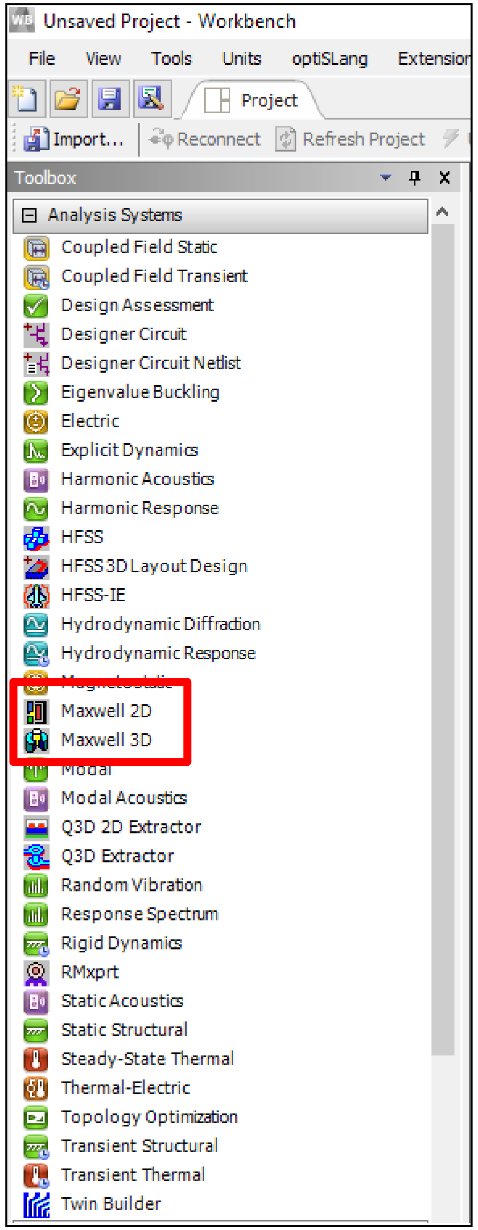
Maxwell Integration in the Workbench

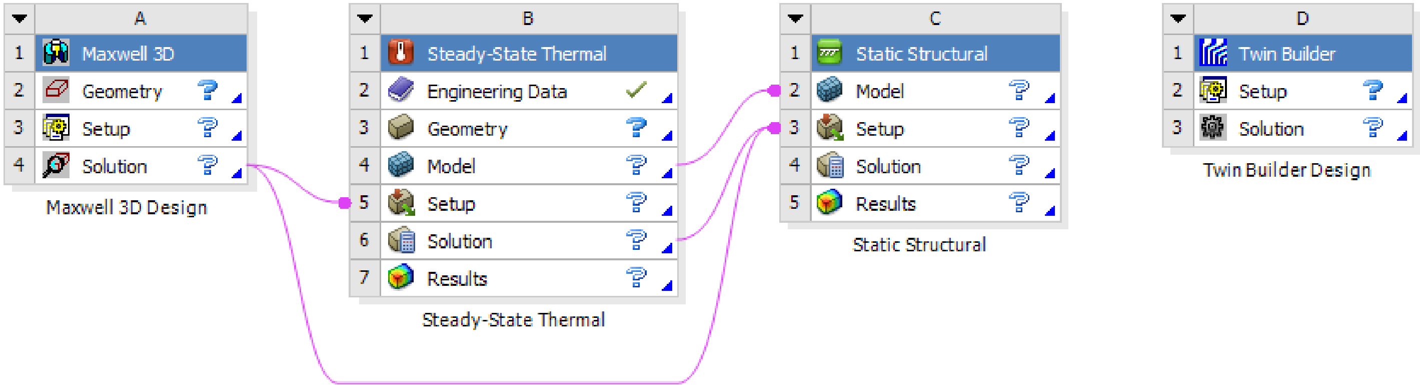
- Integration in the Workbench
  - Launch the Workbench 2020 R1
  - In the Toolbox List the Electromagnetics Products are now present

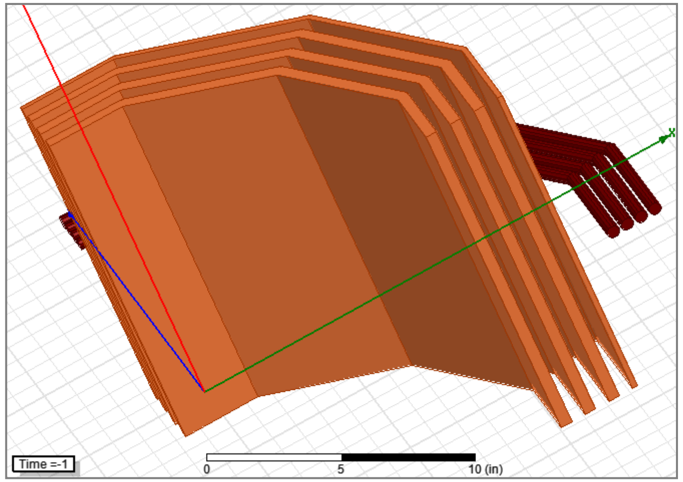


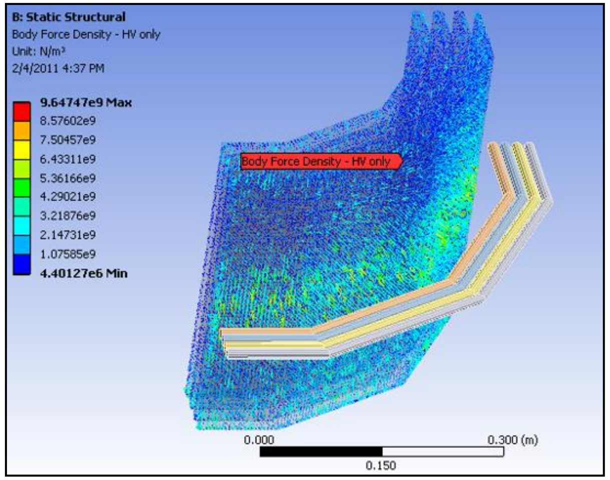
©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Workbench Coupling Technology









©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

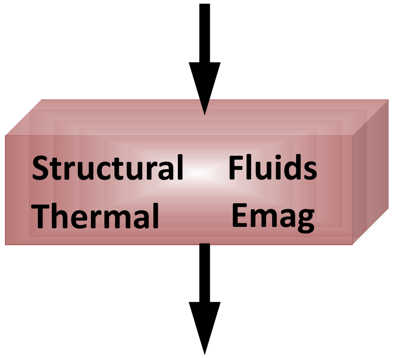
Ansys

Coupled-Physics Solutions – How?

Multi-Field Analysis

Direct Coupling (outside of Maxwell)

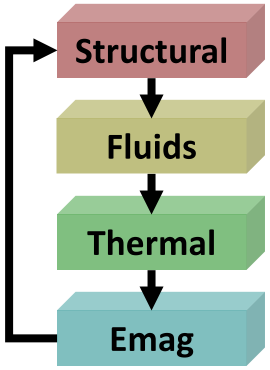
- A single analysis that uses coupled-field elements containing all necessary degrees-of-freedom to solve the coupled-field problem



- Physics solved simultaneously
- Highly coupled physics
- Single model & mesh

Load Transfer (system coupling)

- Two or more analyses coupled by applying results from one analysis as loads to another



- Physics solved sequentially
- Separate model & mesh
- Separation of expertise

Note: examples of Direct Coupling are Joule Heating (CFD, AIM, Mechanical), Thermal-Structural and Piezo-electric. For further details please refer to specific material on Ansys Customer Portal or Ansys Learning Hub

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Coupled-Field Elements vs. System Coupling

Coupled-Field Element

Advantages

- **Ease of use**
  - one element type, one model and mesh, one set of results
- **Robust and accurate solution**
  - Designed for strongly coupled-physics
    - Piezoelectric, electromagnetic, thermo-electric, thermo-elastic
  - Highly nonlinear coupled-field analyses
    - Geometric nonlinearities
    - Multiphysics contact

Disadvantages

- **Large model sizes**
- **Can produce nonsymmetric matrices**

System Coupling

Advantages

- **Solution efficiency**
  - dissimilar mesh interface
    - Independent mesh for each physics
    - Surface and volumetric load transfer
    - 2D to 3D transfer
    - Periodic load transfer
  - independent solver options for each analysis
    - Different time-steps in transient analyses
    - Load transfer between harmonic and static/transient analyses
  - Collaboration between physics experts

Disadvantages

- **Analysis set up**
  - Involves multiple commands and options
- **Possible slow convergence**
- **Possible need of multiple executables**
  - licensing, debugging, ...

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Coupling Methods

Coupling

Strong (matrix, simultaneous)

$$\begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix}$$

- Coupled effects are accounted for by the off-diagonal matrices K_{12} and K_{21}
- Coupled solution achieved in one iteration
- Can produce nonsymmetric matrices
- Allow linear harmonic and modal analyses
- Can be reduced to weak coupling

Weak (load vector, sequential)

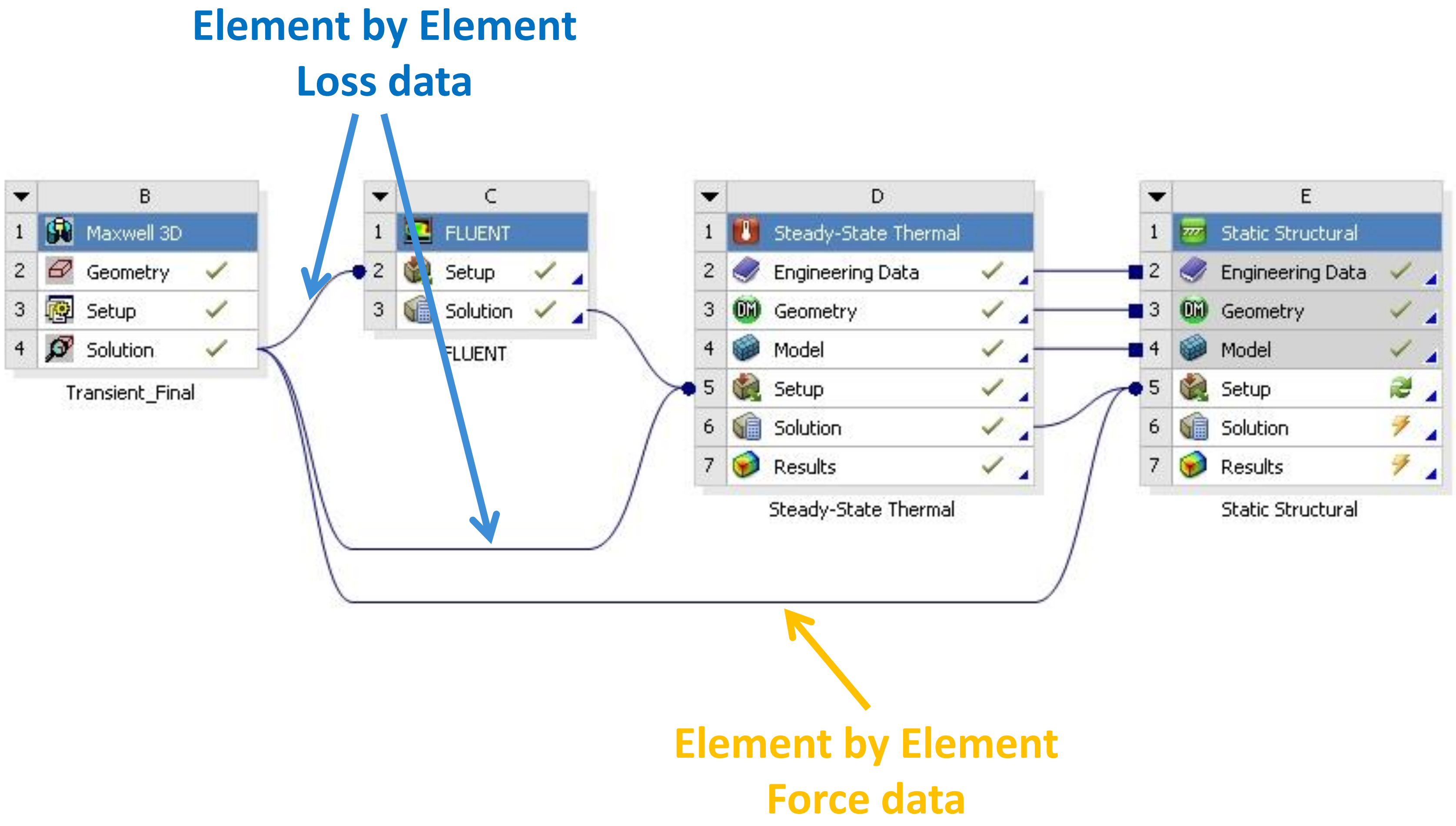
$$\begin{bmatrix} K_{11}(X_1,X_2) & 0 \\ 0 & K_{22}(X_1,X_2) \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix} = \begin{Bmatrix} F_1(X_1,X_2) \\ F_2(X_1,X_2) \end{Bmatrix}$$

- Coupled effects are accounted for as dependencies of K_{11} and F_1 on X_2 and K_{22} and F_2 on X_1
- At least two iterations are required to achieve a coupled response
- Matrices are typically symmetric
- Only static and transient analyses are allowed

Note: taking as example an Ohmic Loss application, F1 would be Voltage and F2 Ohmic Loss, while X1 and would be, respectively, DC current and Temperature

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Automatic field coupling

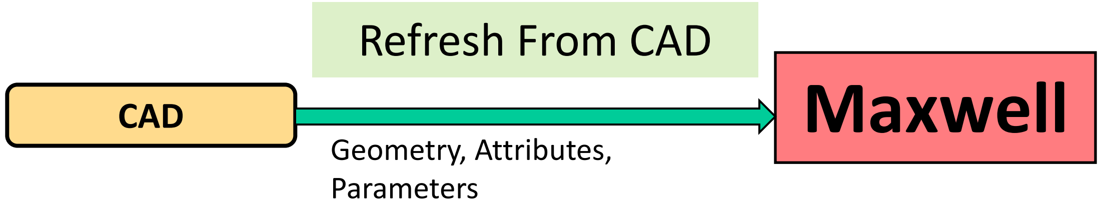


©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

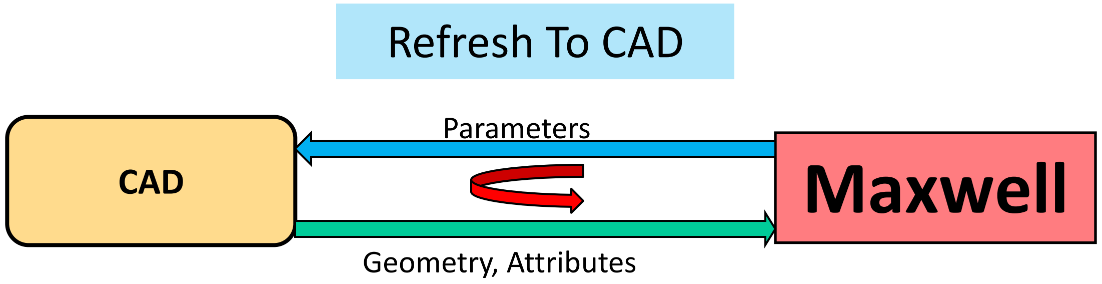
Ansys

Bi-directional CAD integration

- User makes an edit in CAD application and runs ‘Refresh’ on Maxwell Cell
- Refresh pulls the current state of CAD model (geometry, parameters, materials etc) and updates the corresponding data in Maxwell project



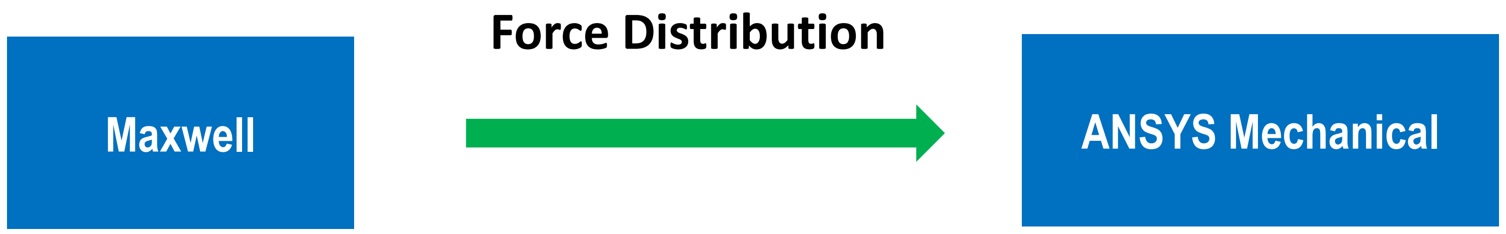
- Maxwell to CAD: User edits CAD parameters in Maxwell application and runs ‘Refresh To Source’ command



©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Maxwell – ANSYS Stress Coupling

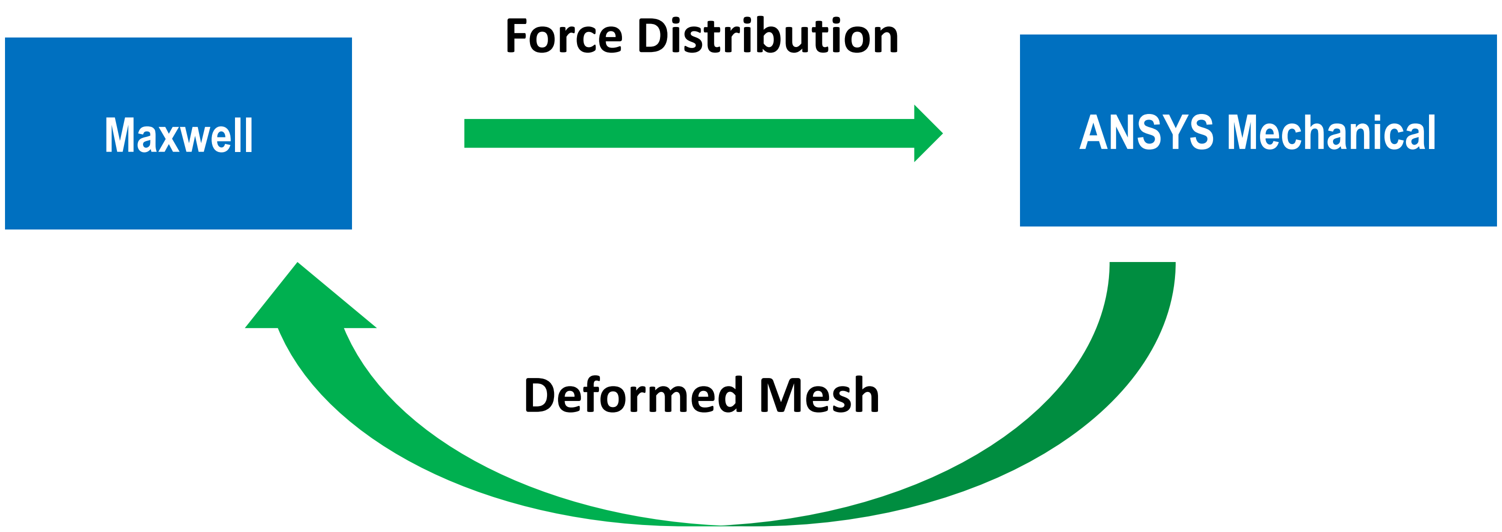
- One way coupling available (2D to 3D, 3D to 3D, static to static, static to transient, transient to transient)
- Approach:
- Maxwell upstream
- The Force distribution is transferred as load into ANSYS Mechanical



©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Maxwell – ANSYS Stress Coupling

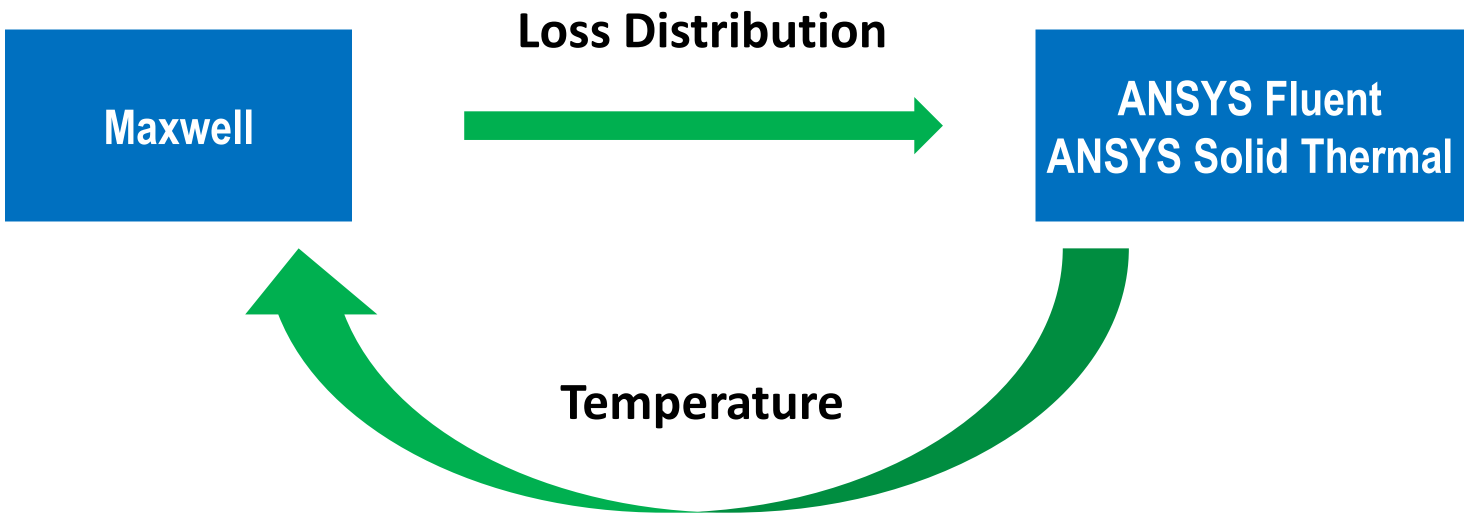
- Two way coupling Maxwell non-transient solver and ANSYS stress solver is possible
- Approach:
- The Force distribution is transferred as load into ANSYS Mechanical
- The node displacement information is sent back to Maxwell as deformed mesh



©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Maxwell – Fluent Two-Way Coupling
Maxwell – ANSYS Mechanical Coupling (solid Thermal solvers)

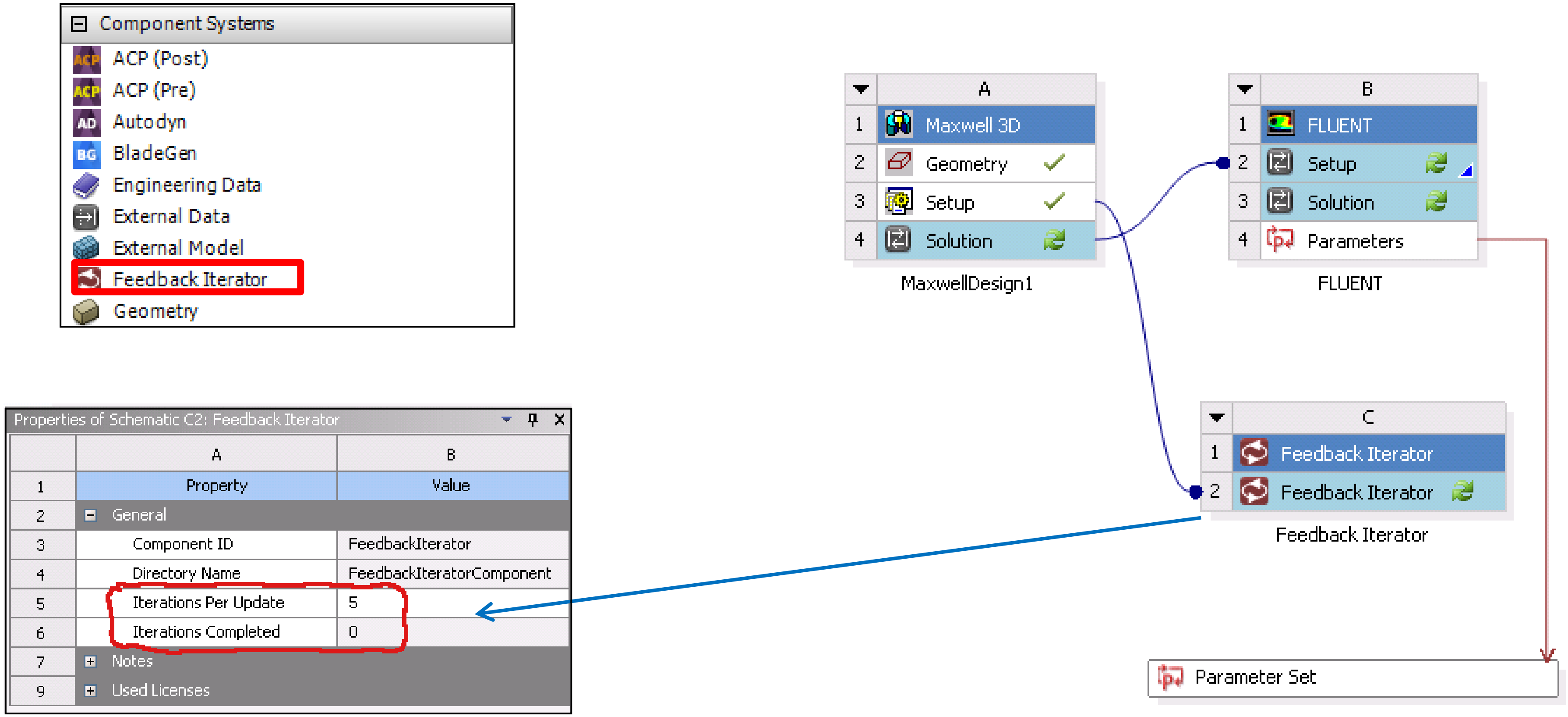
- **Approach:**
- **The Loss distribution is transferred as load into Thermal solver**
- **The Temperature distribution is sent back to Maxwell**



©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Automation of Feedback Couplings

- **Support a ‘Feedback Iterator’ WB system to automate the two-way coupling iterations**



DesignXplorer of coupled system

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Multiphysics Capabilities

- The following table shows the coupling capabilities implemented in 2020 R1
- The symbol → represents a "one way" coupling (Maxwell upstream), while the symbol ⟷ represents a "two ways" coupling, with the possibility to iterate the process

Maxwell Solver	Structural	Thermal	Harmonic Stress	CFD (*)
Magnetostatic	⟷	⟷		⟷
Eddy Current	⟷	⟷	→	⟷
Magnetic Transient	→	⟷	→	
Electrostatic	⟷			
DC Conduction		⟷		
Electric Transient		⟷		

(*) Enable Electric Arc simulations or advanced Induction Heating simulations

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Loss Densities From Maxwell

- **Volumetric Losses**
  - **Ohmic loss (including eddy loss)**
  - **Iron losses (hysteresis loss, eddy loss, excess loss)**
- **Surface Loss**
  - **Losses on impedance boundary**
- **Time-Average Loss from Transient**
  - **In transient solver, the loss is integrated by** $\dot{Q}(t) = \int_0^t \dot{q}\,dt$
  - **At given time points, the solver dumps integrated losses** $\dot{Q}(0), \dot{Q}(t_1), \dot{Q}(t_2) \cdots\cdots \dot{Q}(t_N)$
  - **The average loss rate is given by** $\dot{q}_n = \dfrac{\dot{Q}(t_n) - \dot{Q}(t_{n-1})}{t_n - t_{n-1}}$

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Maxwell Magnetic force calculations

Magnetostatic

Lorentz Force

$$\boldsymbol{f}_V = \boldsymbol{J} \times \boldsymbol{B}$$

The Lorentz force is proportional to the current and to the magnetic lines cutting through it. The direction of this force is perpendicular to current and field

Reluctance Force

$$\vec{f}_s = \left(\vec{n} \bullet \vec{B}_s\right)\frac{\vec{B}_s}{2\mu_0} + \left(\vec{n} \times \frac{\vec{B}_s}{2\mu_0}\right) \times \vec{B}_s$$

Eddy Current

Lorentz Force

$$\boldsymbol{f}_{V\_DC} = \frac{1}{2}\mathrm{Re}\{\boldsymbol{J} \times \boldsymbol{B}^*\}$$

$$\boldsymbol{f}_{V\_AC} = |\boldsymbol{J} \times \boldsymbol{B}|_{\omega t}$$

The AC force has a DC component and an AC component at twice the electrical frequency. This applies for all driven and induced currents

Reluctance Force

$$\vec{f}_{s\_DC} = \frac{1}{2}\mathrm{Re}\left\{\left(\vec{n} \bullet \vec{B}_s\right)\frac{\vec{B}_s^{\,*}}{2\mu_0} + \left(\vec{n} \times \frac{\vec{B}_s}{2\mu_0}\right) \times \vec{B}_s^{\,*}\right\}$$

Transient

Lorentz Force

$$\boldsymbol{f}_V(t) = \boldsymbol{J} \times \boldsymbol{B}$$

The instantaneous force is directly defined by the instantaneous current and flux density.

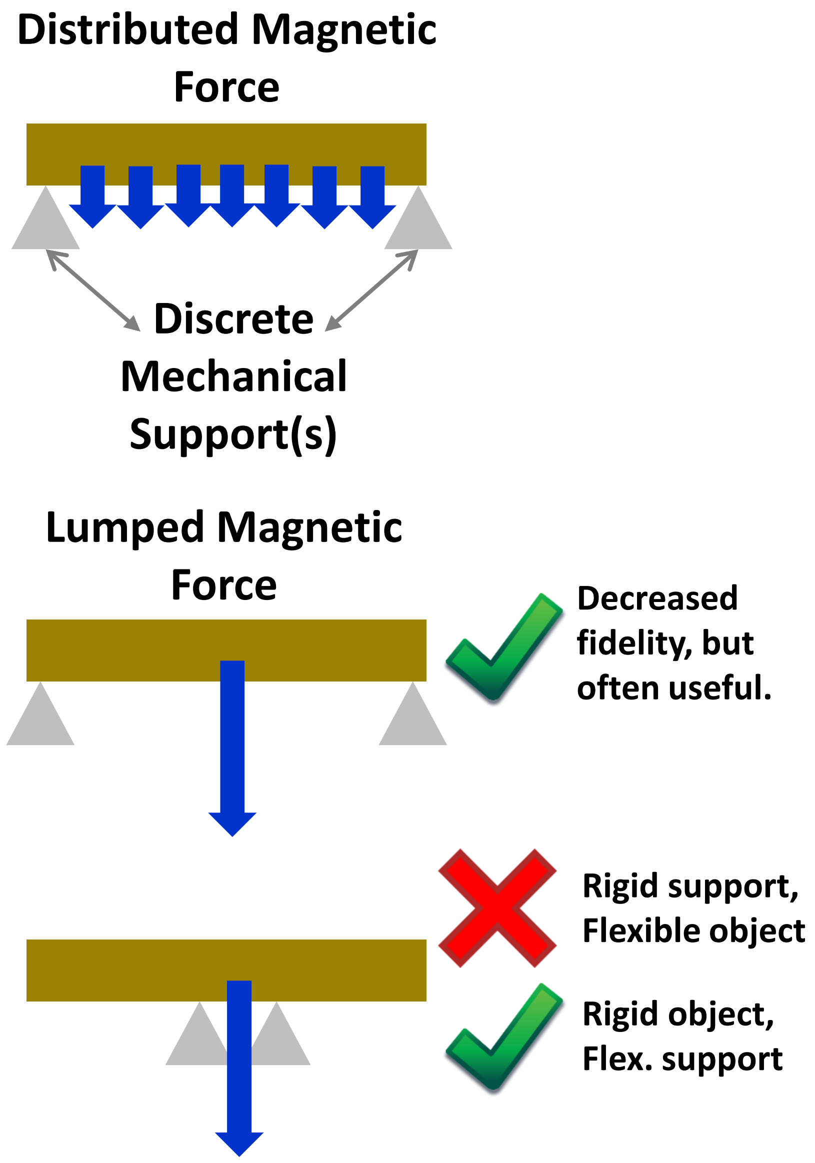
Reluctance Force

$$\vec{f}_s(t) = \left(\vec{n} \bullet \vec{B}_s\right)\frac{\vec{B}_s}{2\mu_0} + \left(\vec{n} \times \frac{\vec{B}_s}{2\mu_0}\right) \times \vec{B}_s$$

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Integrated and lumped forces

- The magnetic force in Maxwell is spatially distributed (volumetric or surface density).
- In many situations it is useful to integrate the force and consider a lumped force on the object(s).
- Whether to use a distributed or lumped force depends on different factors:
  - How stiff is the object (stiff ⇒ lumped)
  - Location of the mechanical supports with respect to the object C.M.
  - Distribution of Magnetic Force – largely varying may require a distributed force.
  - Can use moments of the force distribution for better representation of distribution:
    0th moment: Lumped Force
    1st moment: Lumped Torque around C.M.



©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Energy and Virtual Work

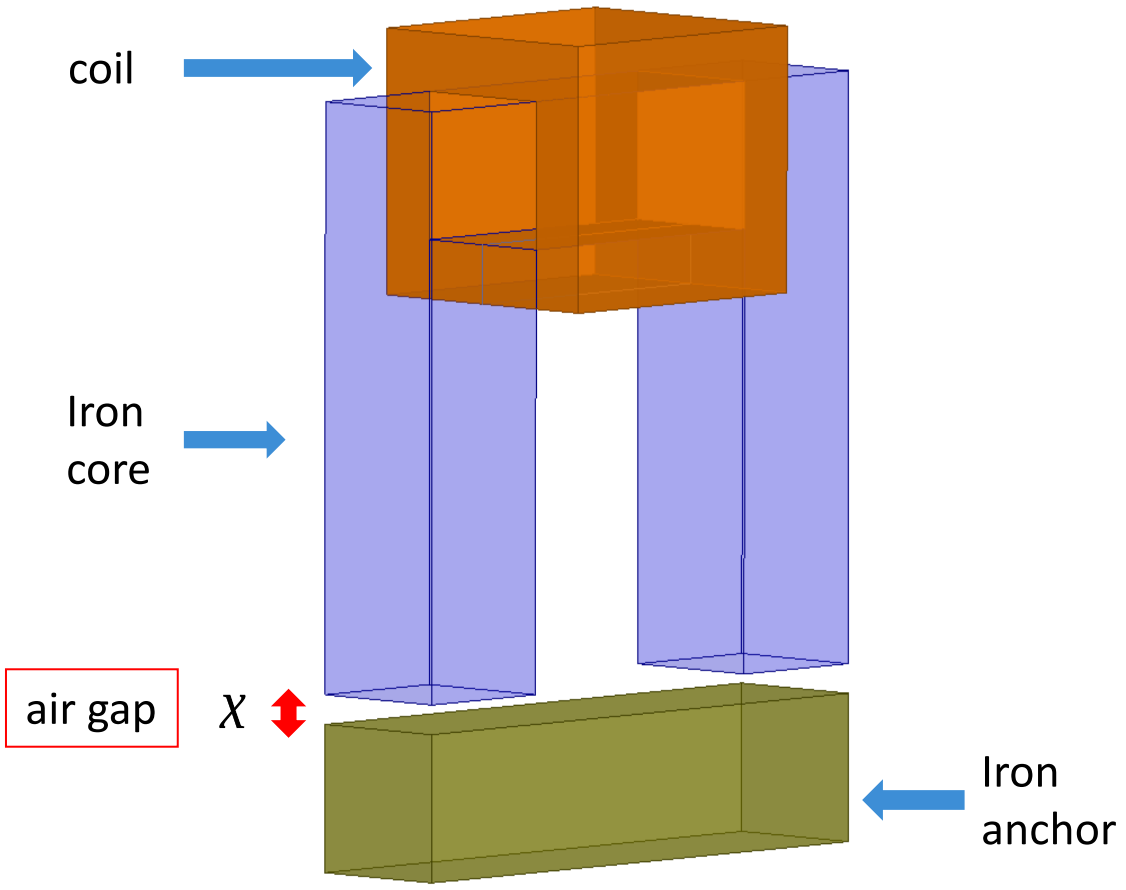
Principle of Virtual Work $F_x = \frac{\partial W(x)}{\partial x}$

$$H = \frac{I}{2x} \qquad B = \frac{\mu_0 I}{2x}$$

$$W = \frac{HB}{2} S \cdot 2x = \frac{\mu_0 I^2 S}{4x}$$

$$F = \frac{\partial W}{\partial x} = -\frac{\mu_0 I^2 S}{4x^2}$$

I – current crossing through the coil
S – iron core and iron anchor cross section



Note: Through this example it is possible to tune FEA results with analytical ones

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Forces and Maxwell Stress Tensor (MST)

– The force acting on an object can be expressed as

$$F_\alpha = \sum_\beta \int_V \frac{\partial}{\partial x_\beta} T_{\alpha\beta} dV$$

$T_{\alpha\beta}$ is the stress tensor - see (1) on the bottom right side

– Apply the divergence theorem to volume integral (3)

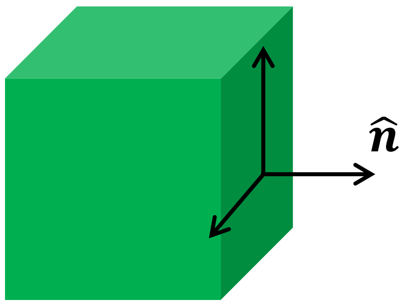
$$F_\alpha = \oint_S \sum_\beta T_{\alpha\beta} n_\beta ds$$

– The surface force: the force per unit area across the surface (2)

$$\sum_\beta T_{\alpha\beta} n_\beta$$

– For magnetic field, the explicit form of surface force density is

$$\sum_\beta T_{\alpha\beta} n_\beta = (\vec{B} \bullet \vec{n})\vec{H} - \frac{1}{2}\vec{B} \bullet \vec{H}\vec{n}$$



$\vec{f}_s$ is a surface force density
The normal component is a pressure.
The tangential components are shears.
$\vec{f}_s$ has units [N/m^2]; $\vec{f}_V$ has [N/m^3].

(1) $\vec{f}_V = \nabla \cdot \vec{\vec{T}}$

(2) $\vec{f}_s = \vec{\vec{T}} \cdot \hat{n}$

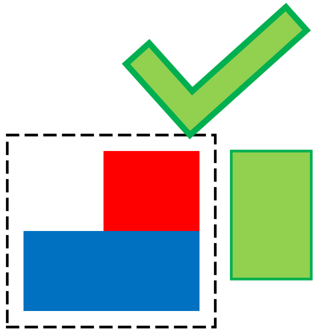
(3) $\int \nabla \cdot \vec{\vec{T}} dV = \oint \vec{\vec{T}} \cdot \hat{n} dS$

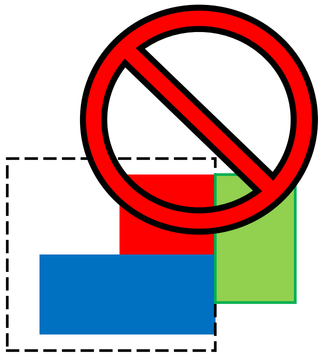
(4) $\int \vec{f}_V dV = \oint \vec{f}_s dS$

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Virtual Work, MST, Lorentz

– Virtual Work and Maxwell Stress Tensor are generally surface-density distributions, and are meant to be applied to object(s) with an airgap surrounding them – not touching other objects.
– At least, the interpretation has to be correct when using objects in contact with other objects.
– Virtual Work considers a “virtual displacement” of the surface elements of selected objects, and assumes air is surrounding, therefore a gap is required.
– MST is applied to surface of dashed line – i.e. surfaces in the air gap, and completely surrounding object(s).
– Lorentz Force is always a volumetric density of JxB.
– The integration of these volume/surface densities creates a lumped force. The 1st moment a torque.





©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Field Mapping from Maxwell 3D to Mechanical

- Surface Force Density
  Surface_Force_Density (Maxwell 3D) to Surface Force Density (Mechanical):
  Magnetic reluctance force, e.g. as two magnets, or flux from a coil that creates a force between two pieces of steel, e.g. an actuator.

  Note: If μ_r > 1, Surface force >> Volume force.

- Volume Force Density
  Volume_Force_Density (Maxwell 3D) to Body Force Density (Mechanical)
  Due to Lorentz Force, where a current in a conductor is impinged with a magnetic field, F = I x B on Coils and bus bars. If μ_r = 1, volume force only.

  Note: If μ_r > 1 and σ > 0 (and Eddy current activated) volume force will grow a lot. If the steel is fully saturated, or in a case of a magnet (μ_r ~1.05), the Surface and Volume force will be close, if you have heavy eddy currents, you may need to consider both.

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Field Mapping from Maxwell 2D to Mechanical

- Edge Force Density
  Edge_Force_Density (Maxwell 2D) to Surface Force Density (Mechanical) :
  Magnetic reluctance force, e.g. as two magnets, or flux from a coil that creates a force between two pieces of steel, e.g. an actuator.

  Note: If $\mu_r > 1$, Surface force >> Volume force.

- Surface Forces
  Surface_Force_Density (Maxwell 2D) to Body Force Density (Mechanical)
  Due to Lorentz Force, where a current in a conductor is impinged with a magnetic field, F = I x B on Coils and bus bars. If $\mu_r = 1$, surface force only.

  Note: If $\mu_r > 1$ and $\sigma > 0$ (and Eddy current activated) volume force will grow a lot. If the steel is fully saturated, or in a case of a magnet (μ_r ~1.05), the Surface and Volume force will be close, if you have heavy eddy currents, you may need to consider both.

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

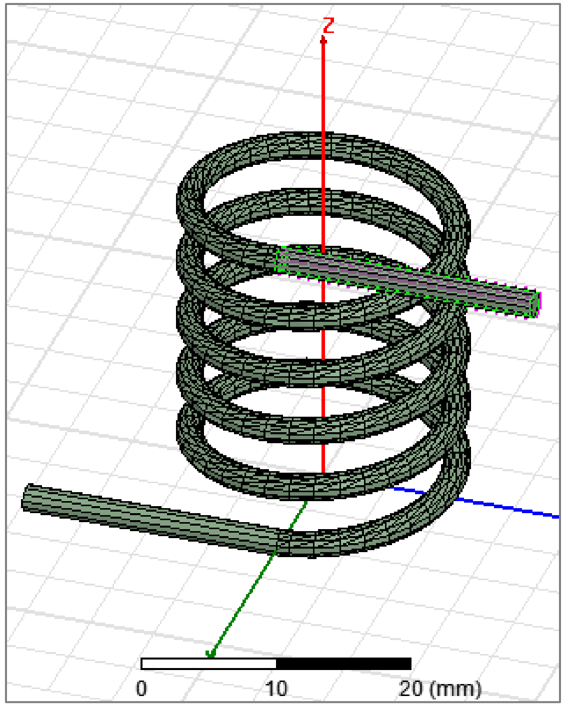
Summary

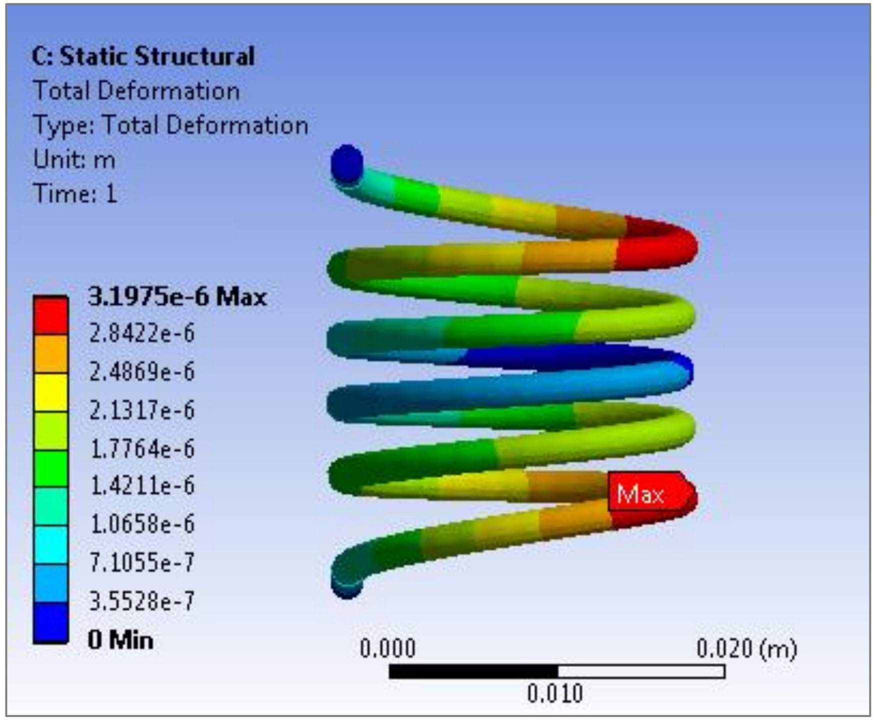
What have we learned in this session?

- Workbench concepts
- Maxwell integration in Workbench
- Coupling capabilities
- Force calculations

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

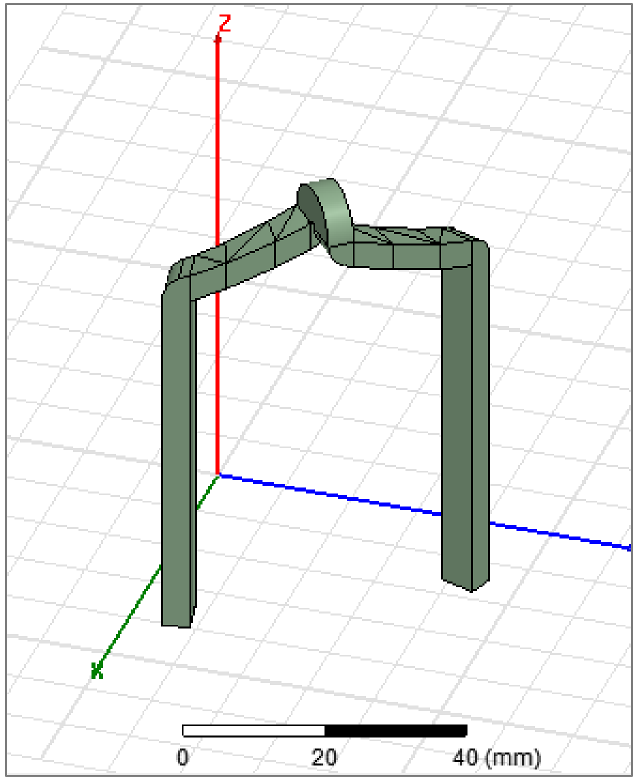
Workshop 05.1 – Electromagnetic – Mechanical coupling

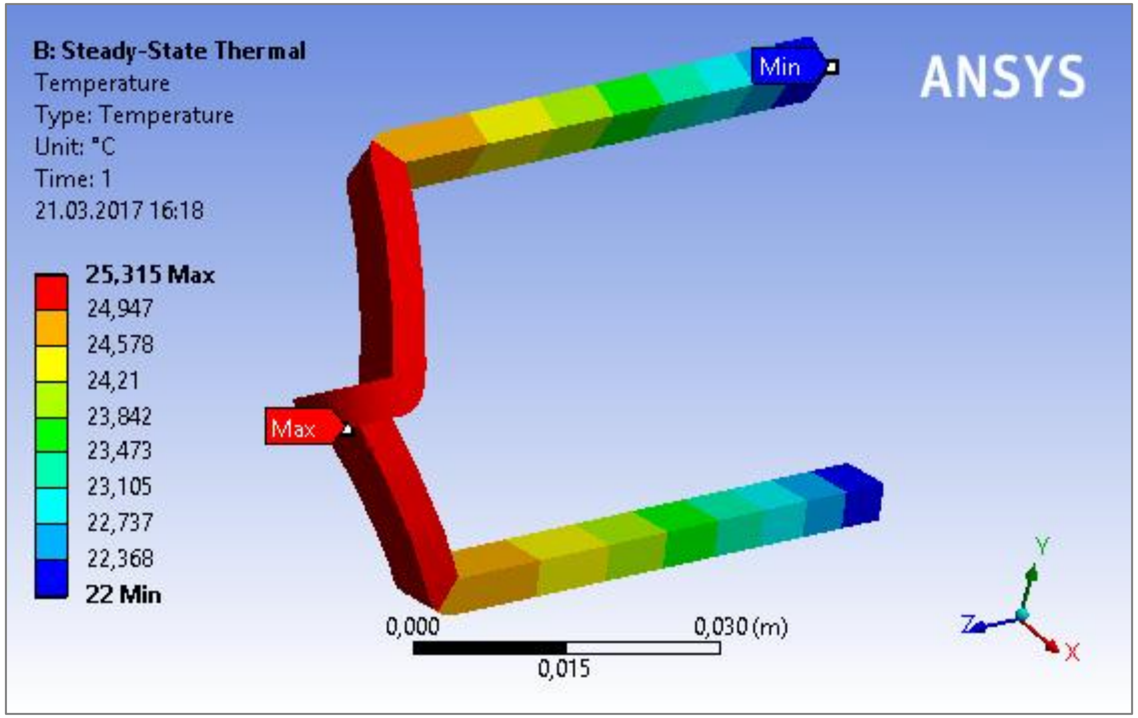




©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

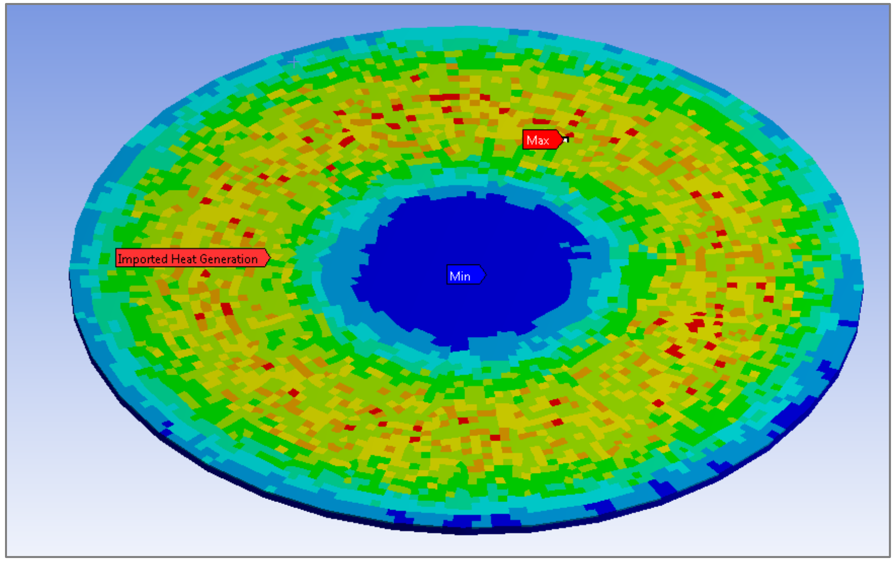
Workshop 05.2 – Electromagnetic – Thermal coupling

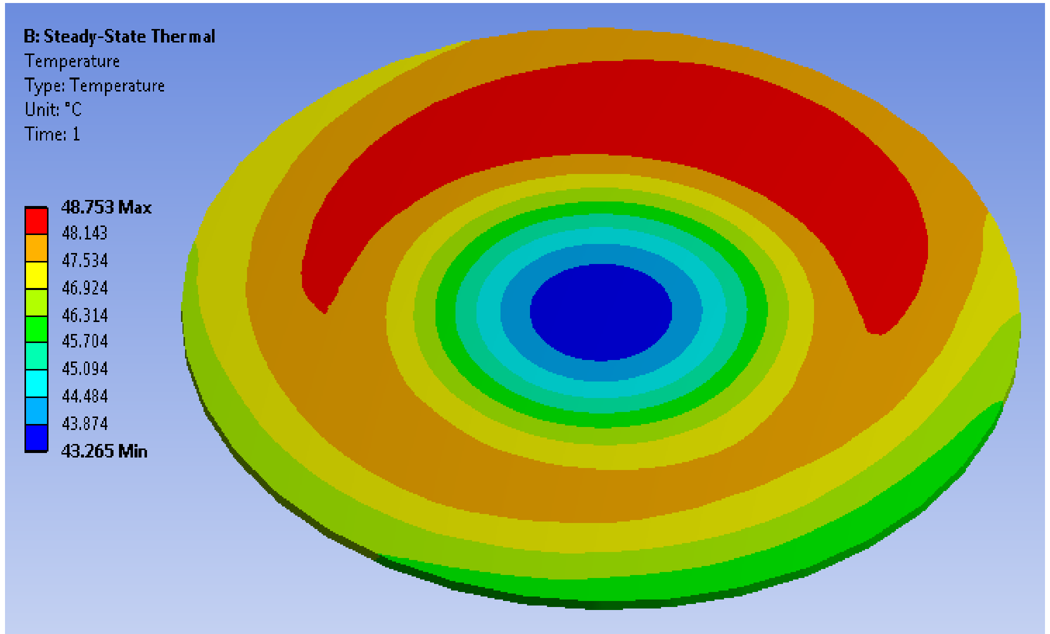




©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Workshop 05.3 – Electromagnetic Eddy Current – Thermal coupling





©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.