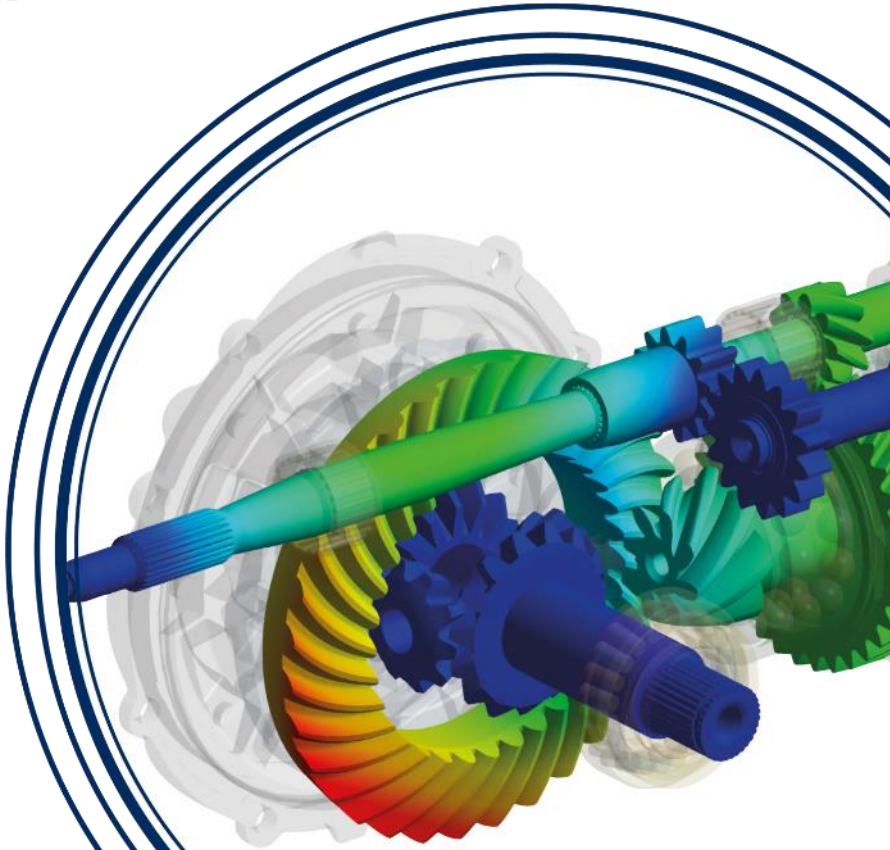
Fortgeschrittene multiphysikalische Simulation mit System Coupling

Fokus auf Elektrisch-Thermisch

René Fuger, CADFEM (Austria) GmbH



CADFEM Ansys CERTIFIED ELITE CHANNEL PARTNER

Motivation

Multiphysics simulation allows engineers and designers to design and create virtual prototypes of their products operating under real-world conditions where the interactions between fluid dynamics, structural mechanics, and electromagnetics are key.

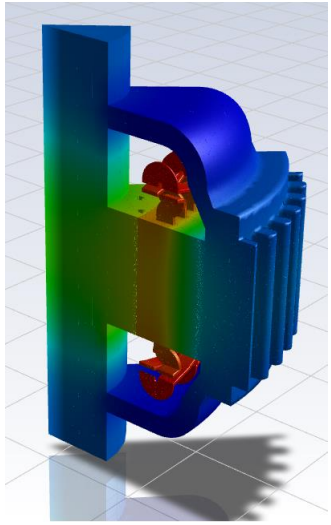
Multiphysics simulations are becoming increasingly important in modern industrial Computer-Aided Engineering (CAE) Applications.

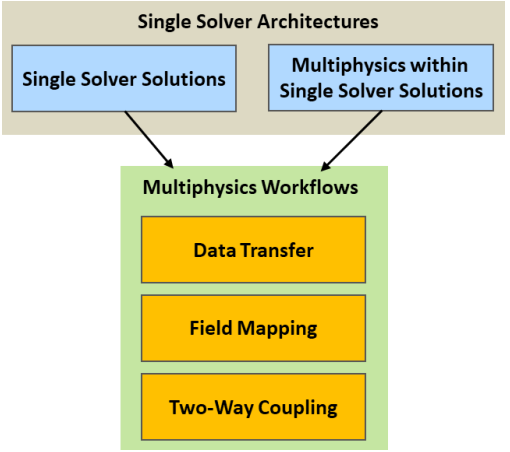
Due to the interactions between more than one physics disciplines, multiphysics problems such as those encountered for example in aerospace, biomedical, civil, and car engineering domains tend to be extremely challenging to simulate.

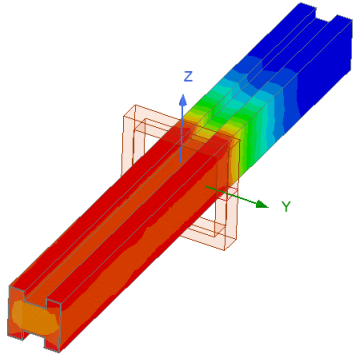
Complex problems in such industries involve physics (corresponding to different spatial as well as temporal scales) from multiple disciplines such as electrical, electromagnetics and fluid-mechanical.

Agenda

- **Multiphysics Methods**
- **Ansys System Coupling**
- **Example: Electric Motor Steady-State (Maxwell-Fluent Coupling)**
- **Example: Inducting Heating of a H-Beam (Maxwell-Mechanical Coupling)**







Multiphysics Methods

There are primarily two different approaches that researchers have adopted to conduct multiphysics simulations: monolithic and partitioned.

Multiphysics within Single Solver Solutions

(monolithic)

Equations concerning all physics involved in a problem are solved together as a single matrix system

Multiphysics Workflows

(partitioned)

Separate equations are solved within different single-physics software and coupled with a suitable mapping method.

Multiphysics Methods

There are primarily two different approaches that researchers have adopted to conduct multiphysics simulations: monolithic and partitioned.

Multiphysics within Single Solver Solutions

(monolithic)

Ansys Mechanical APDL Multiphysics Elements

Equations concerning all physics involved in a problem are solved together as a single matrix system

Multiphysics Workflows

(partitioned)

Separate equations are solved within different single-physics software and coupled with a suitable mapping method.

Multiphysics Methods

There are primarily two different approaches that researchers have adopted to conduct multiphysics simulations: monolithic and partitioned.

Multiphysics within Single Solver Solutions

(monolithic)

Ansys Mechanical APDL Multiphysics Elements

Equations concerning all physics involved in a problem are solved together as a single matrix system

Multiphysics Workflows

(partitioned)

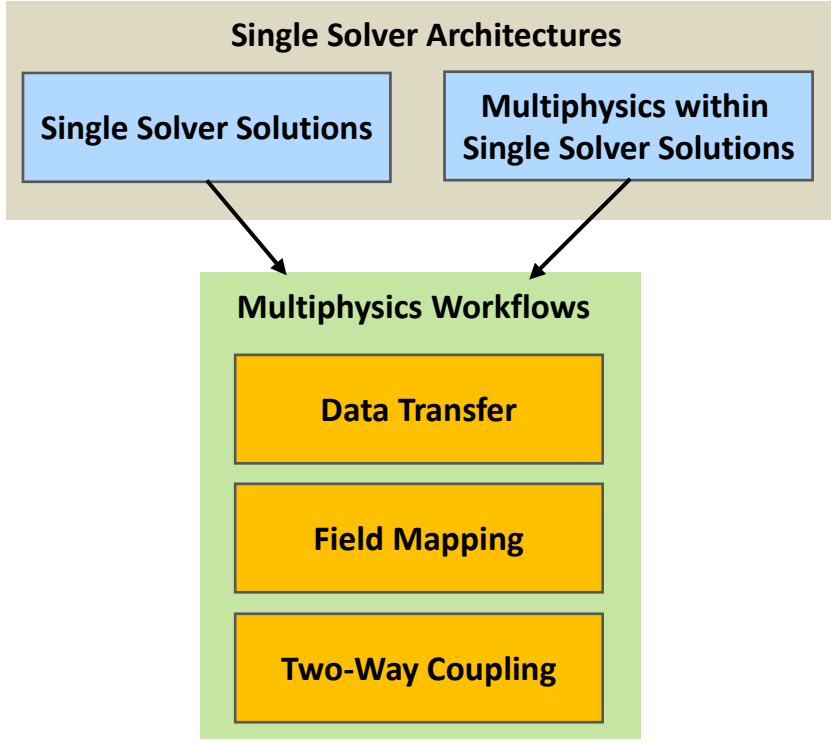
System Coupling

Separate equations are solved within different single-physics software and coupled with a suitable mapping method.

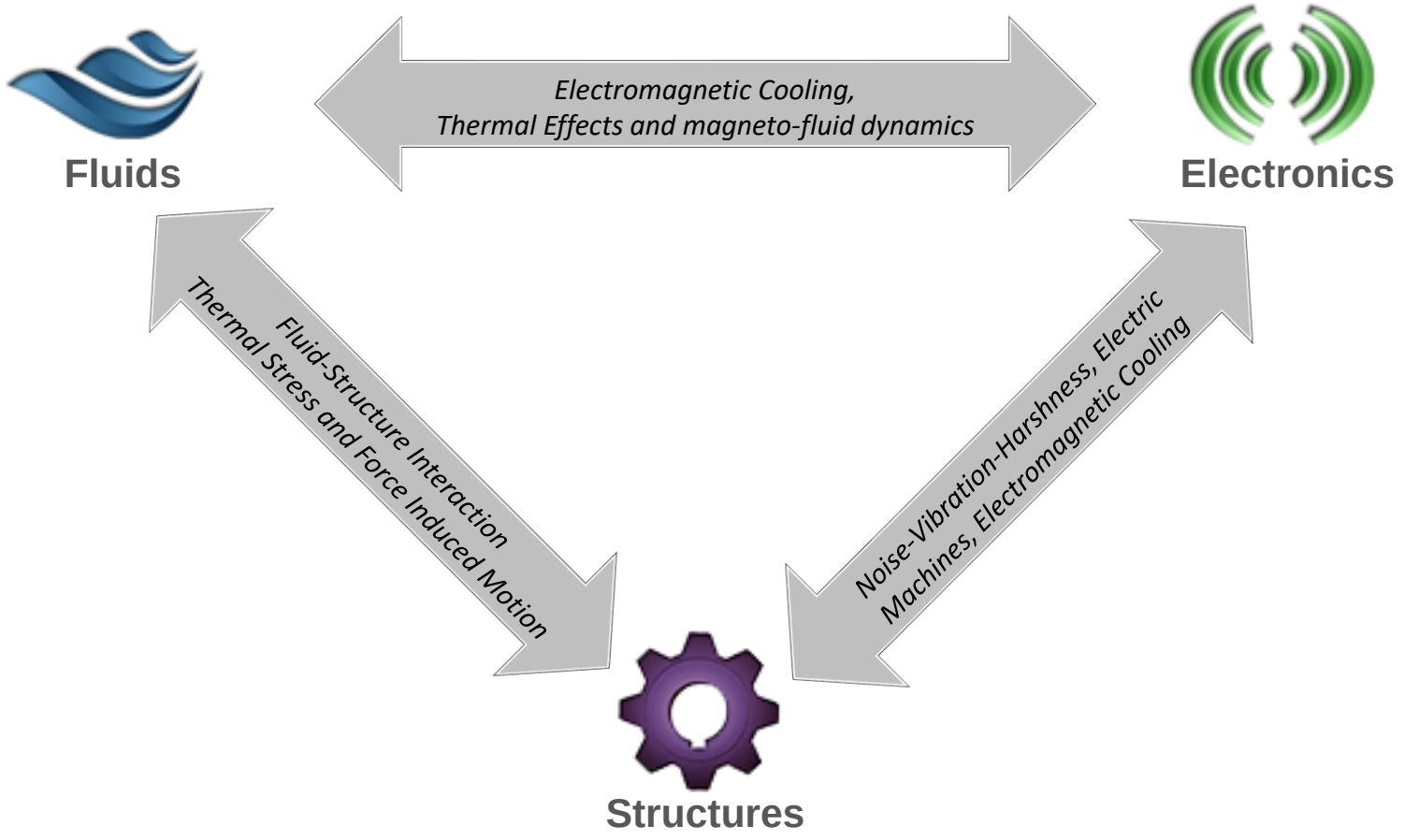
Multiphysics Workflows

Integrate single solver solutions, 3rd party software or experimental data, for high fidelity simulations

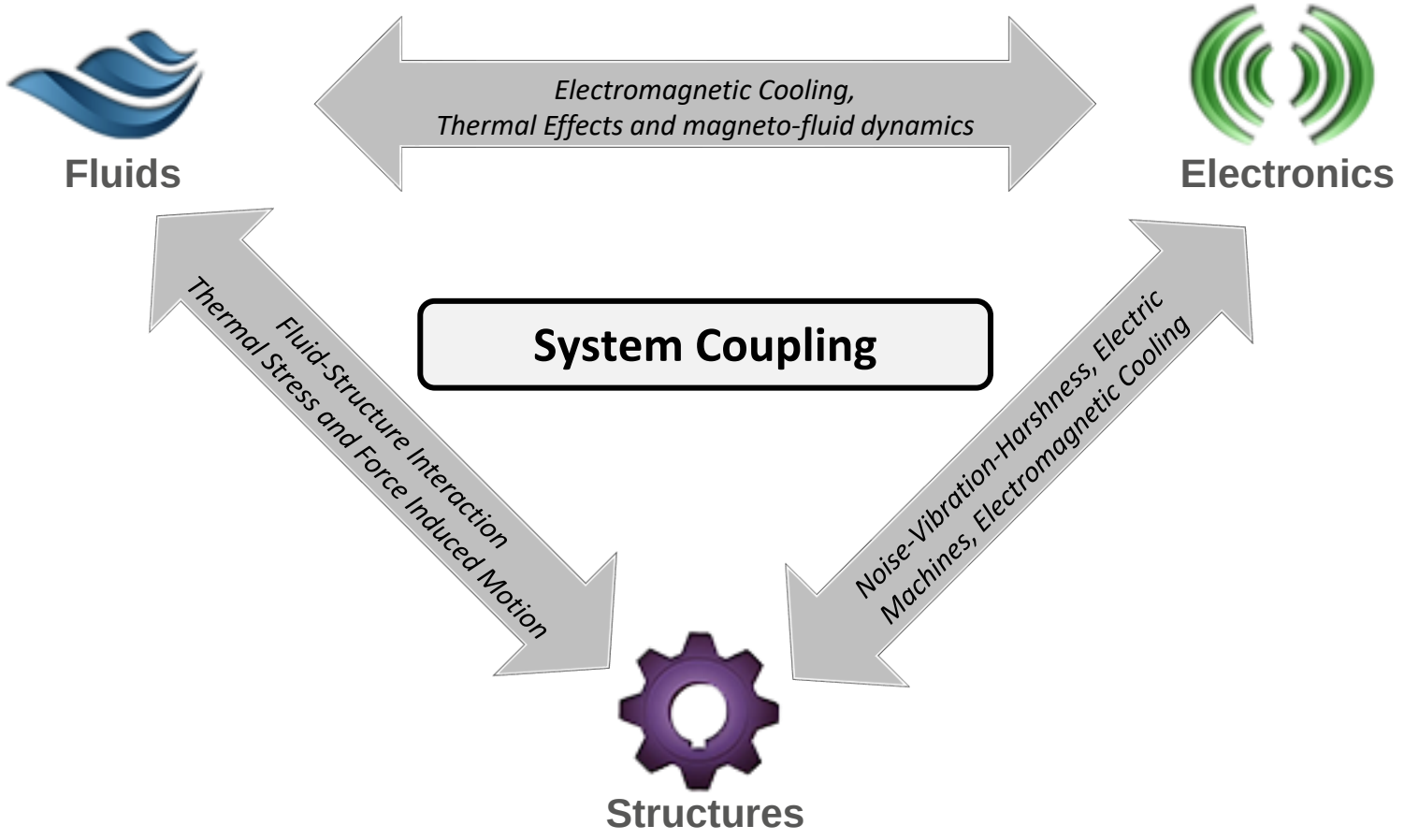
- **Data transfer:** Transfer and map data from experiments and other software packages for initial or boundary conditions
- **One-way couplings:** Automatic results transfer for when one physics solution is dependent on another
- **Two-way couplings:** Tight integration for when different physics solutions heavily impact each other



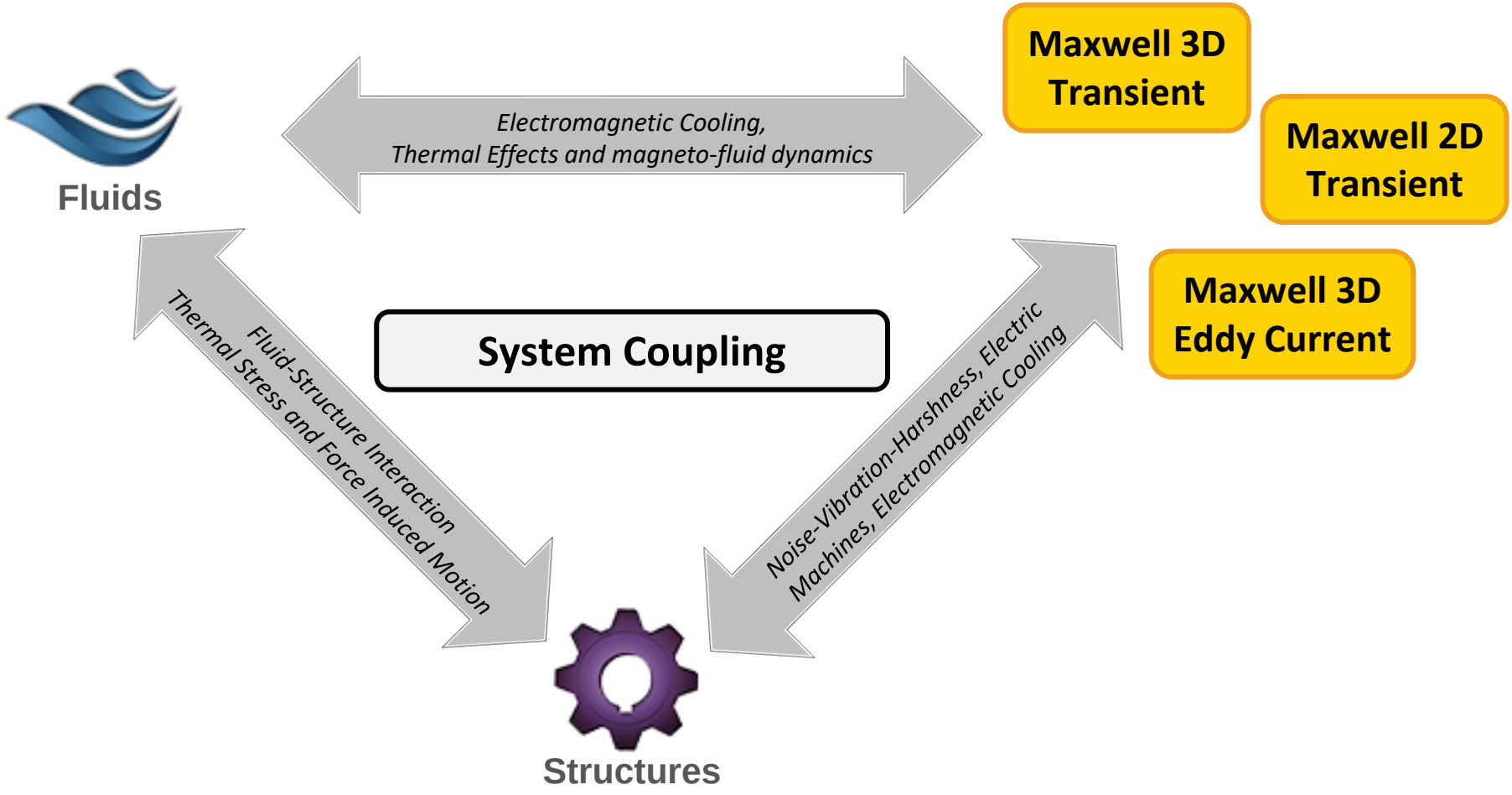
Ansys Multiphysics



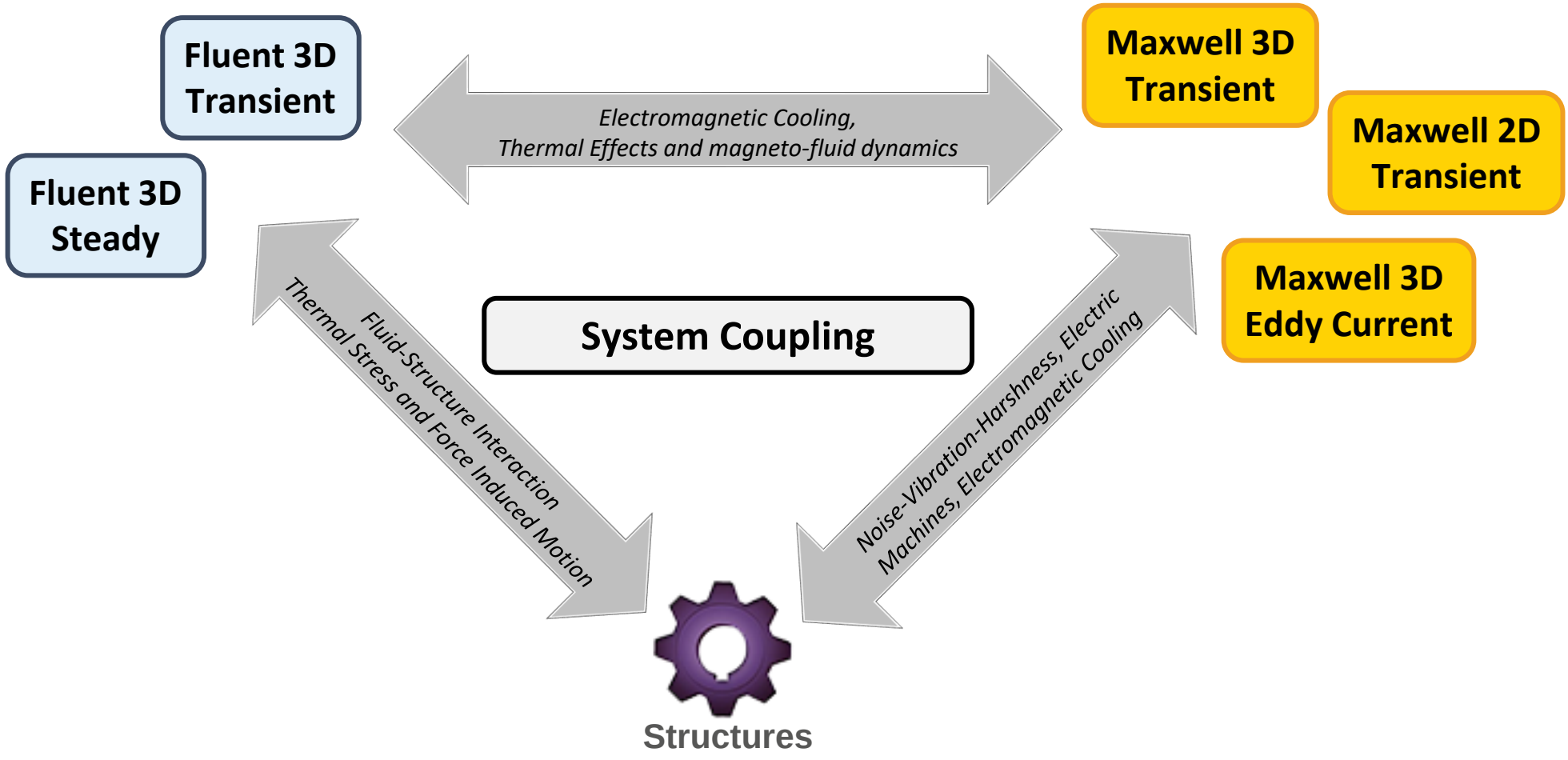
Ansys Multiphysics



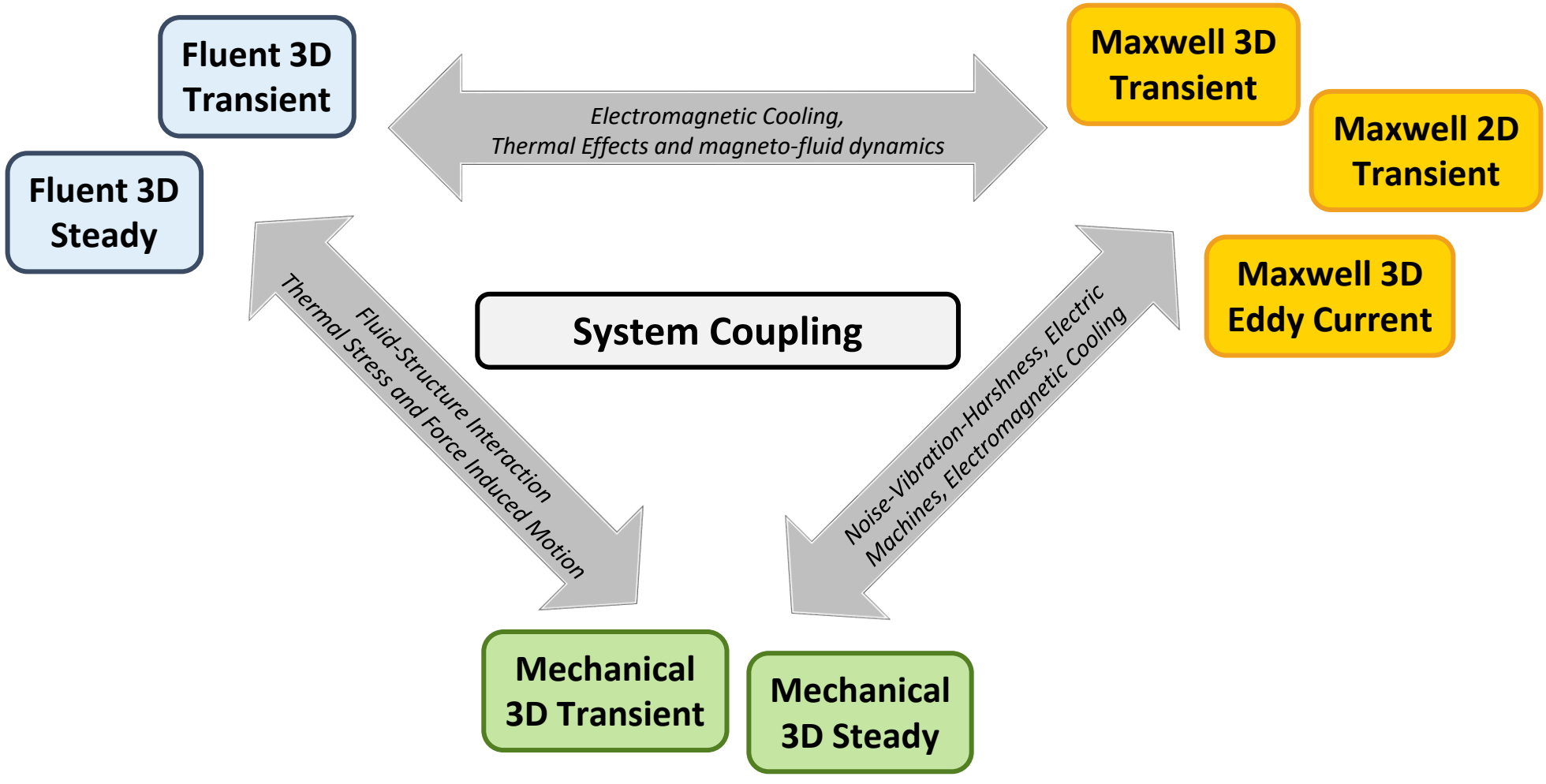
Ansys Multiphysics



Ansys Multiphysics



Ansys Multiphysics

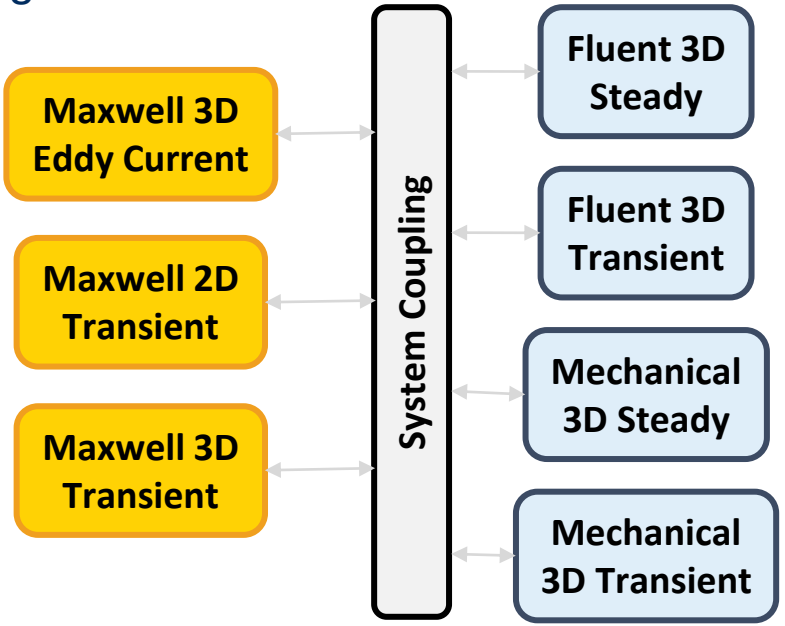


System Coupling Overview

A Framework to Couple ANSYS Tools

System Coupling extend capabilities for coupling

- Dissimilar mesh
- Multiple time-scales
- Scalable
- HPC-capable
- Cross-platform



System Coupling Overview

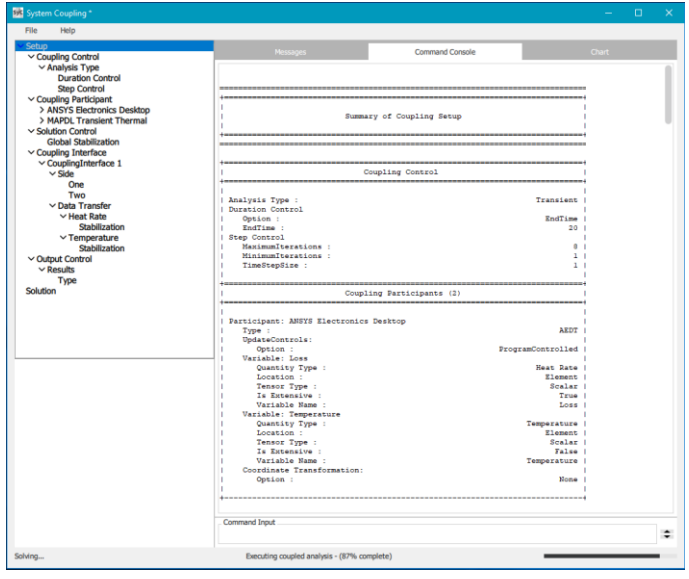
- System Coupling can integrate *multiple individual analyses*, enabling you to leverage different physics solvers and/or static external data sources in a single multiphysics simulation.

- System Coupling can be performed using:
  - the System Coupling *graphical user interface* (GUI)
  - the System Coupling *command-line interface* (CLI), or
  - within *ANSYS Workbench* (WB).

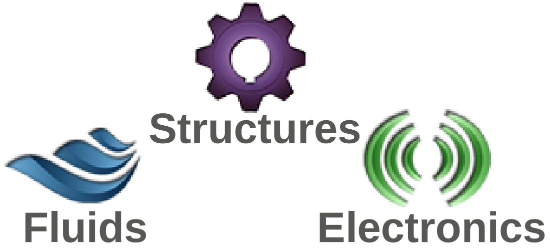
- System Coupling manages the execution of simulations between *coupling participants*, which are the applications or data sources that send and/or receive data in a coupled analysis.

System Coupling Overview

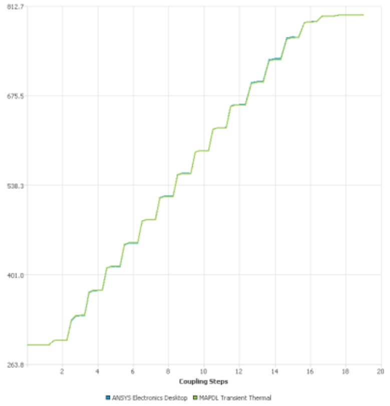
- System Coupling can integrate *multiple individual analyses*, enabling you to leverage different physics solvers and/or static external data sources in a single multiphysics simulation.
- System Coupling can be performed using:
  - the System Coupling *graphical user interface* (GUI)
  - the System Coupling *command-line interface* (CLI), or
  - within *ANSYS Workbench* (WB).
- System Coupling manages the execution of simulations between *coupling participants*, which are the applications or data sources that send and/or receive data in a coupled analysis.

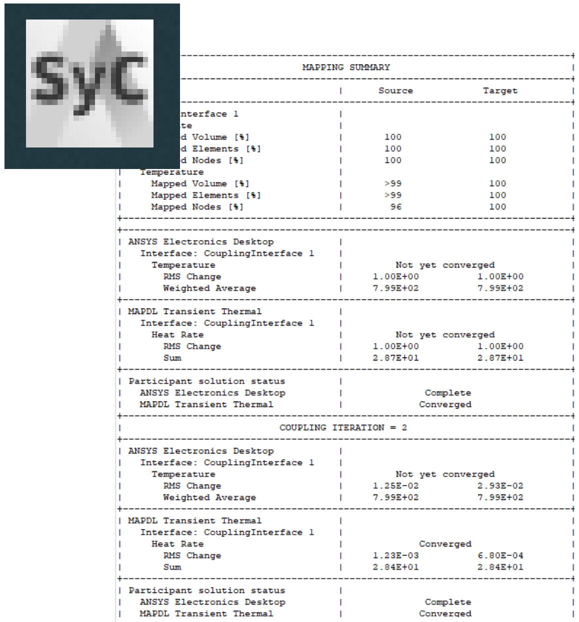


Steps of a System Coupling Analysis



1. **Set up the physics for coupling participants**
   For each participant, set up data transfer variables and regions to be coupled, as well as coupling-related settings that enable a coupled analysis.
2. **Set up System Coupling**
   Set up the System Coupling part of the analysis, specifying analysis settings and defining data transfers.
3. **Run the coupled analysis**
   Run the coupled analysis by starting System Coupling and each of the participant solvers.
4. **During the coupled analysis**
   - Monitor solution progress
   - Stop and resume the analysis
5. **After the coupled analysis**
   - Extend and restart the analysis
   - Debug your coupled analysis
6. **Review the analysis output**





```
MAPPING SUMMARY
                                 |     Source        Target
nterface 1                       |
te                               |
d Volume [%]                     |      100           100
d Elements [%]                   |      100           100
d Nodes [%]                      |      100           100
  Temperature                    |
    Mapped Volume [%]            |      >99           100
    Mapped Elements [%]          |      >99           100
    Mapped Nodes [%]             |       96           100

ANSYS Electronics Desktop        |
  Interface: CouplingInterface 1 |
    Temperature                  |      Not yet converged
      RMS Change                 |   1.00E+00      1.00E+00
      Weighted Average           |   7.99E+02      7.99E+02
MAPDL Transient Thermal          |
  Interface: CouplingInterface 1 |
    Heat Rate                    |      Not yet converged
      RMS Change                 |   1.00E+00      1.00E+00
      Sum                        |   2.87E+01      2.87E+01
Participant solution status      |
  ANSYS Electronics Desktop      |          Complete
  MAPDL Transient Thermal        |          Converged
                 COUPLING ITERATION = 2
ANSYS Electronics Desktop        |
  Interface: CouplingInterface 1 |
    Temperature                  |      Not yet converged
      RMS Change                 |   1.25E-02      2.93E-02
      Weighted Average           |   7.99E+02      7.99E+02
MAPDL Transient Thermal          |
  Interface: CouplingInterface 1 |
    Heat Rate                    |          Converged
      RMS Change                 |   1.23E-03      6.80E-04
      Sum                        |   2.84E+01      2.84E+01
Participant solution status      |
  ANSYS Electronics Desktop      |          Complete
  MAPDL Transient Thermal        |          Converged
```

System Coupling Settings

- **Analysis settings**

  Define the type of analysis and controls for the initialization, duration, and steps of the analysis.
- **Solution settings**

  Define analysis-level solution controls, such as simultaneous execution of participant solutions.
- **Execution control settings**

  Define details about the execution of the coupled analysis.
- **Output control settings**

  Define the frequency with which coupling output (specifically, restart points) is generated.
- **Participant settings**

  Define a name for each participant, regions from and to which data can be transferred, the input and output data transfer variables available for each region, and the frequency with which the participant is updated.
- **Data transfer settings**

  Define a source and a target for each data transfer. Both source and target are defined by a coupling participant, along with a region and variable defined for that participant. Additional data transfer settings control how specified data transfers are executed. These can include convergence target, relaxation factor, ramping, and Quasi-Newton solution stabilization options.
- **Expert settings**

  Provide additional controls for the coupled analysis.

Coupling Interfaces and Data Transfers

Coupling Interfaces

A coupling interface defines two sets of regions, each set belonging to one participant, between which data can be transferred during the coupled analysis. A single coupling interface encompasses all transfers in either direction between a given set of regions on the two sides of the interface.

Interfaces support two-way data transfers, so a given participant can be both a source and a target in the same interface. Because of this, both the interface sides and their associated participants and regions are referred to as "side One" and "side Two," instead of "source" and "target.“

Data Transfers

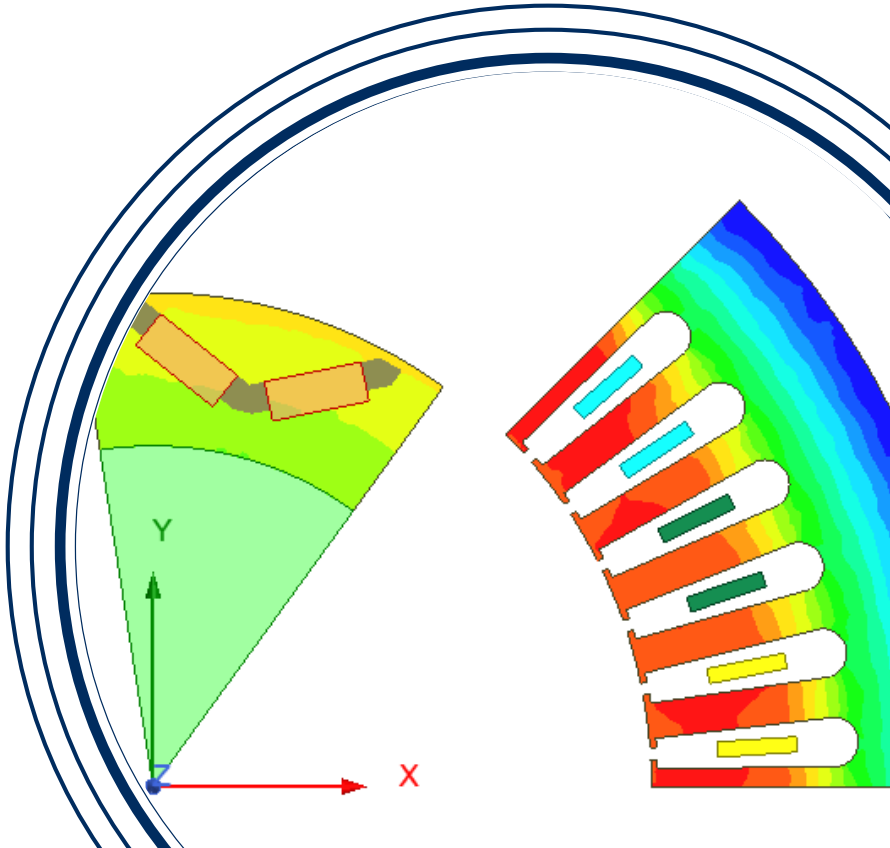
A data transfer is the transfer of a single quantity type in one direction between the two sides of the interface.

Because data transfers go in only one direction, associated participants and regions may still be referred to as "source" and "target" when discussed in relation to a data transfer.

Electric Motor

Task:

A steady electromagnetic-thermal co-simulation of a permanent magnet electric motor.



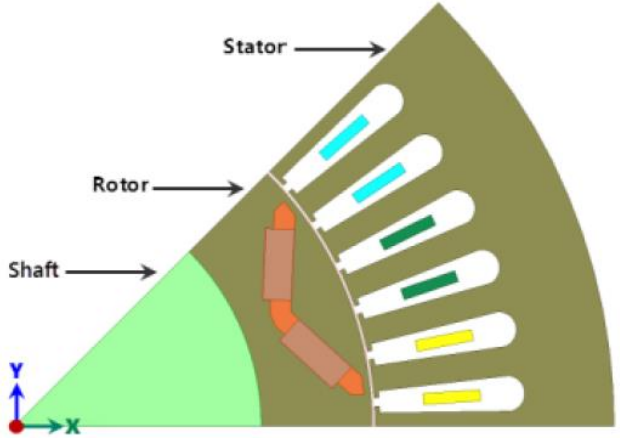
CADFEM Ansys CERTIFIED ELITE CHANNEL PARTNER

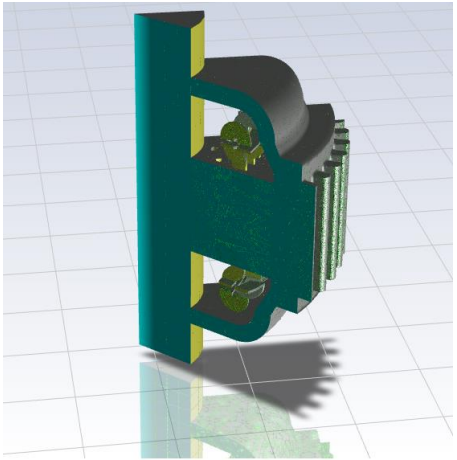
Electric Motor

- **Machine Type**: Internal Permanent Magnet Machine

- **Maxwell 2D Design**
  - It can be assumed a uniform power-loss distribution along the axial direction.
  - Reduce Maxwell's computation time by simulating only an axial slice.

- **Fluent 3D Design**
  - Fluent simulates the thermal coupling between the rotor and stator components. The rotor and stator are separated by two air-like solid zones.





Coupled Simulation Process

Physics

- Electro-magnetic and CHT coupled physics

Time Scales

- Maxwell performs a 2D transient electromagnetic solution. The solution values sent to System Coupling are time-averaged over the duration of the co-simulation.
- Fluent performs a 3D steady thermal solution. Based on this, the co-simulation also has a steady analysis type.

Data Transfer

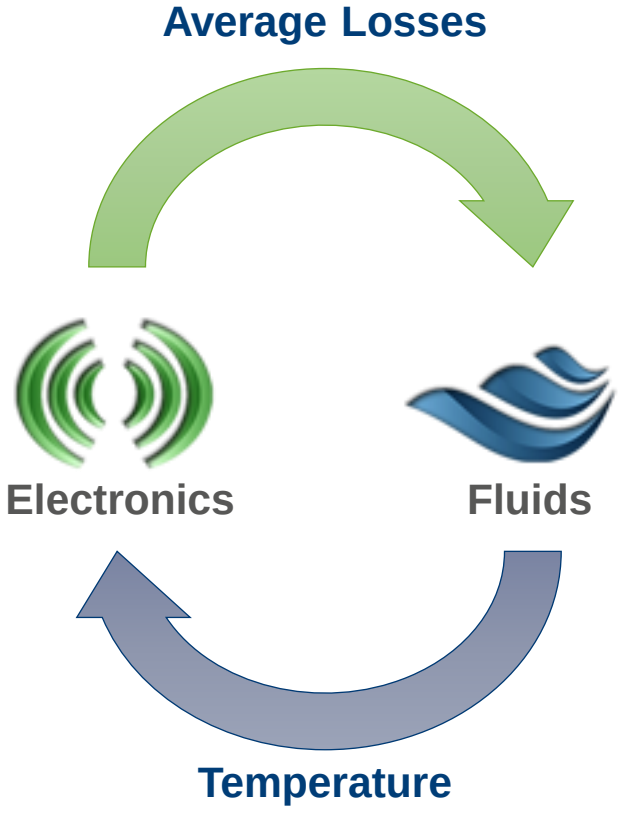
- System Coupling coordinates the simultaneous execution of their solvers and the data transfers between them.
- Data are transferred between planar surface (Maxwell) and volume (Fluent) topologies.

Physics

Maxwell

- Transient Solver
  - Movement of the rotor
  - (Power supply wave form)
- Materials
  - Temperature dependent properties
- Data transfer
  - Import temperature
  - Export average losses
- System Coupling Setup
  - Averaging time period

«Include Temperature Dependence» and «Enable Feedback» should be turned on



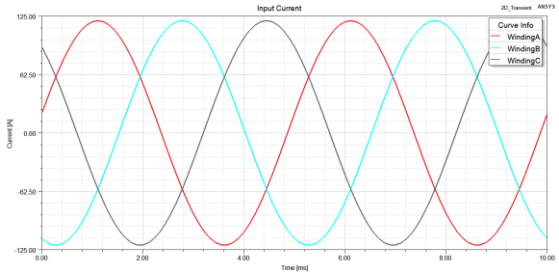
Fluent

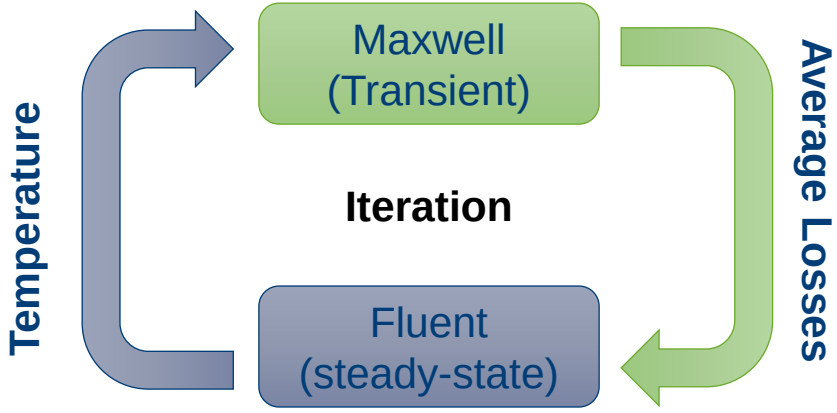
- Steady-state
  - Temperature (Solving Energy Equations)
  - (Flow)
- Materials
  - Temperature dependent properties
- Data transfer
  - Import heat rate
  - Export temperature

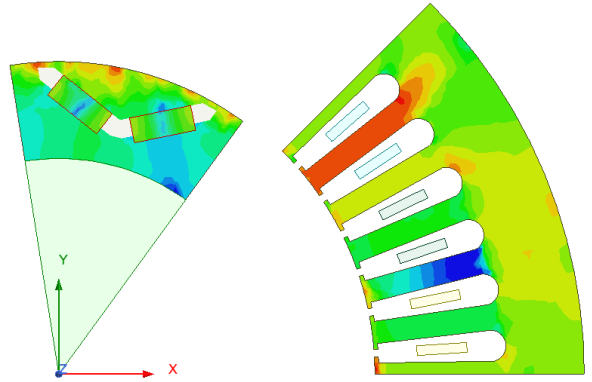
Temperature dependent properties (permeability, el. conductivity, magnetic coercivity, core losses)

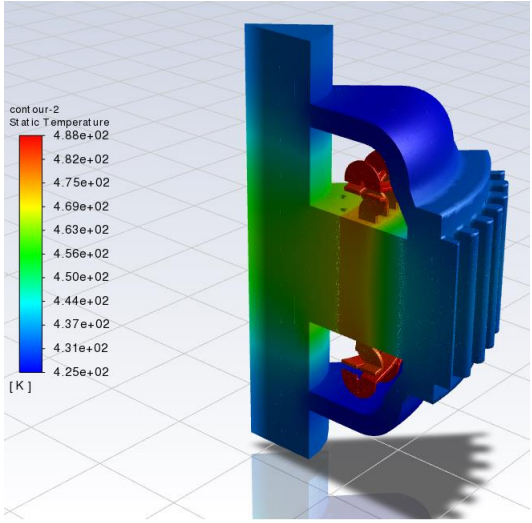
Data Transfer

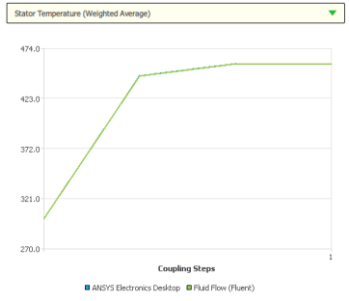
Several electromagnetic and thermal simulations are performed until the quantities converge.

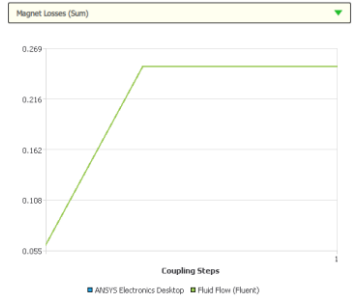


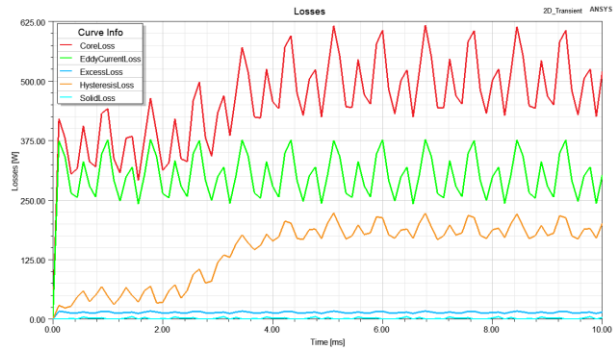






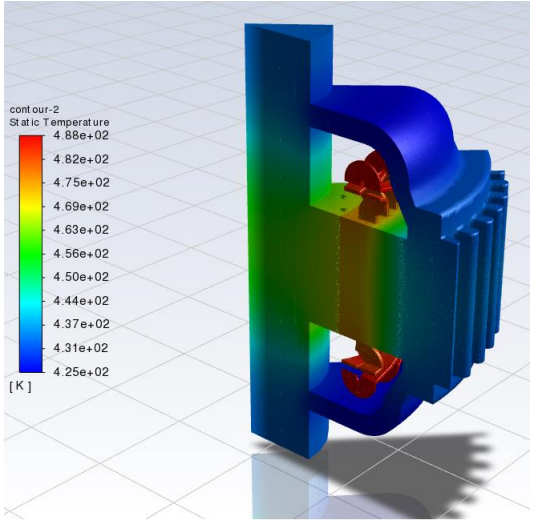


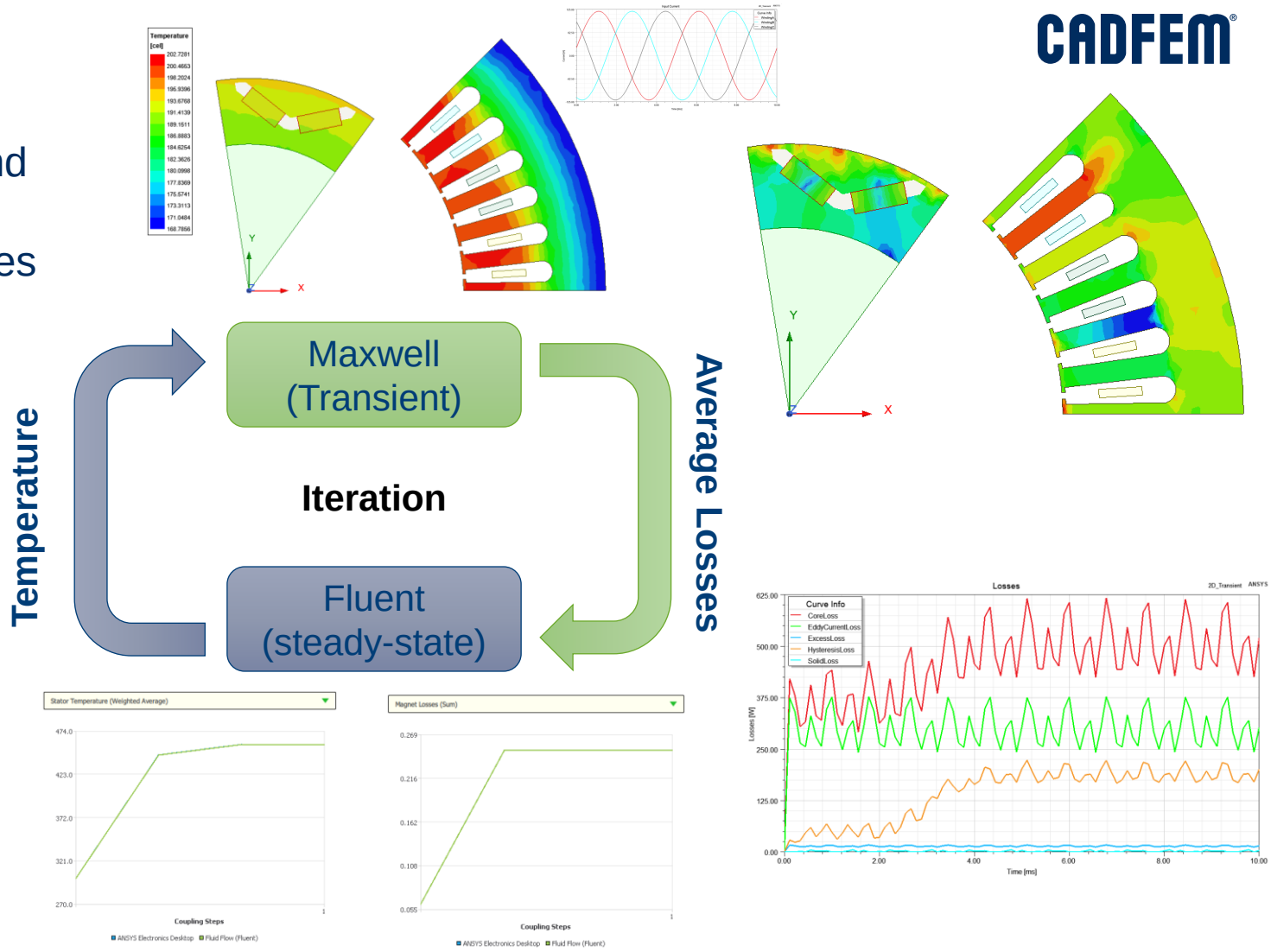




Data Transfer

Several electromagnetic and thermal simulations are performed until the quantities converge.



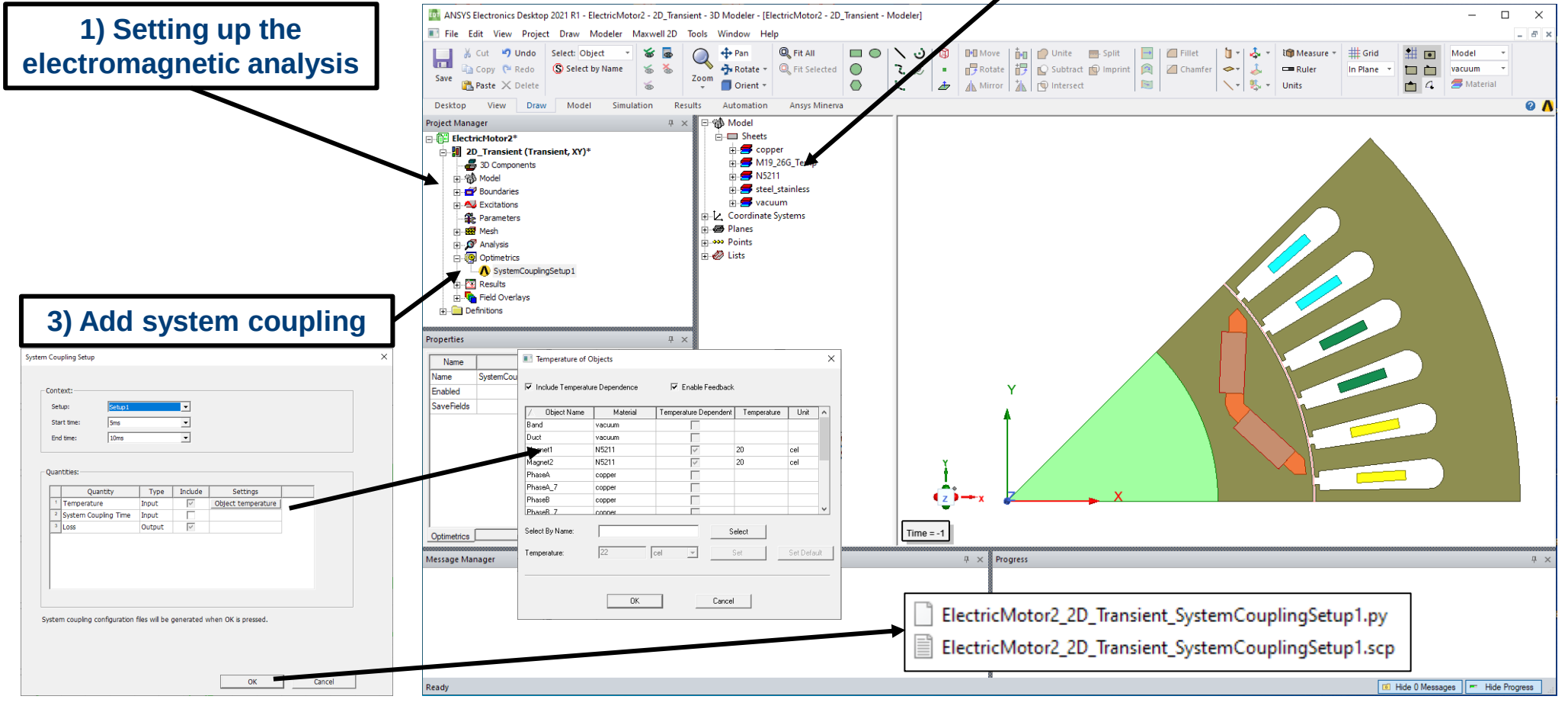


Maxwell

1) Setting up the electromagnetic analysis

2) Defining the Materials

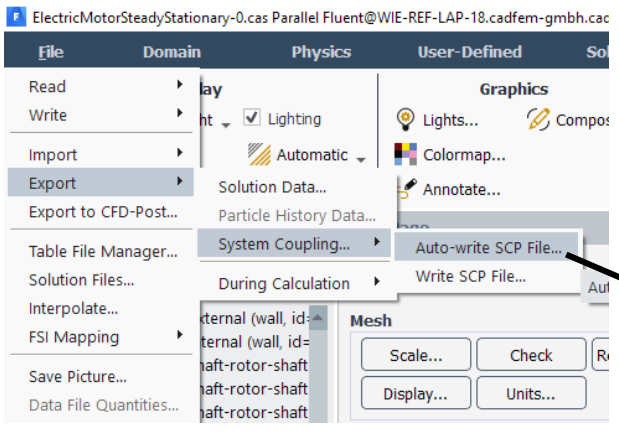
3) Add system coupling

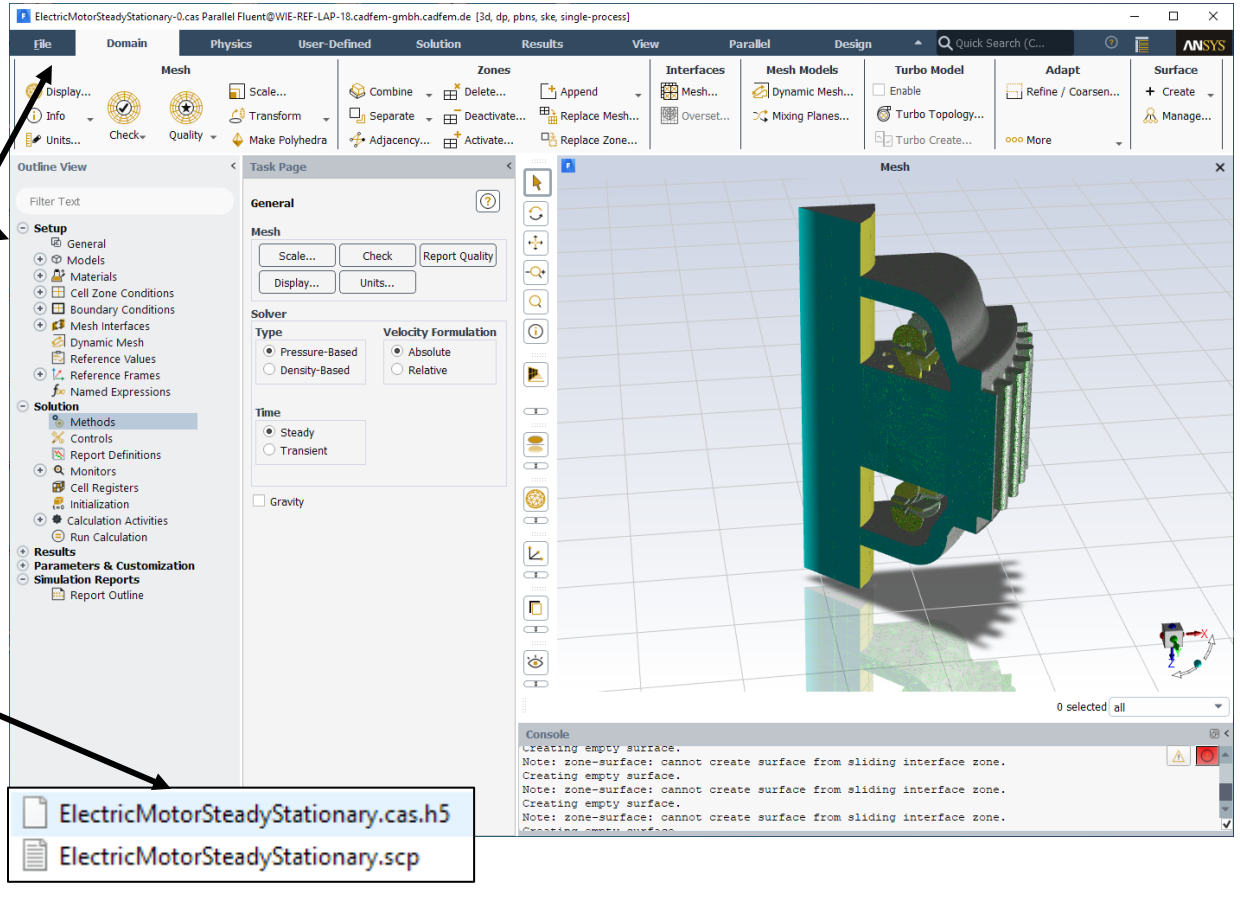


Fluent

1) Setting up the electromagnetic analysis

2) Export SCD File





System Coupling

1) Add Coupling Participants

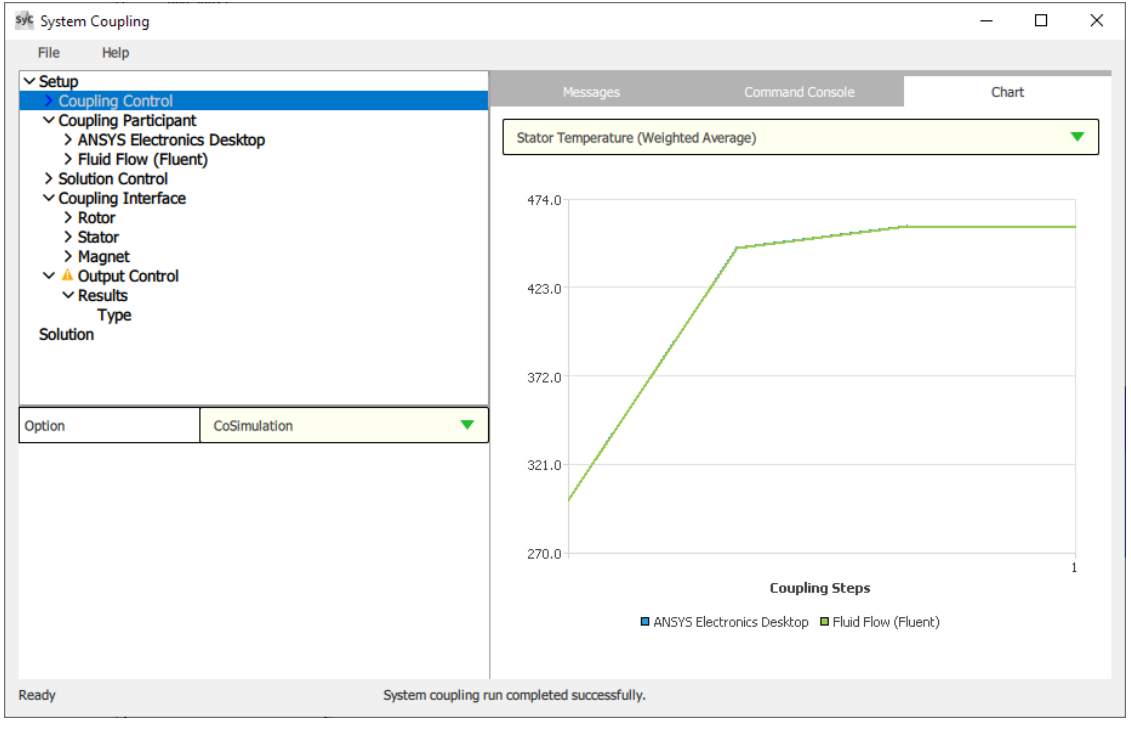
2) Set Coupling Control Settings

3) Add Coupling Interfaces

3b) Add Data Transfer

4) Solve the Co-Simulation

5) Postprocessing System Coupling Results



Postprocessing System Coupling's Results

- Evaluate Mapping Quality
  - Review Mapping in the Transcript
  - Visualize Temperature Mapping Quality in EnSight

MAPPING SUMMARY

	Source	Target
Rotor		
Rotor Losses		
Mapped Area/Volume [%]	100	100
Mapped Elements [%]	100	100
Mapped Nodes [%]	100	100
Rotor Temperature		
Mapped Area/Volume [%]	91	100
Mapped Elements [%]	84	100
Mapped Nodes [%]	60	98
Stator		
Stator Losses		
Mapped Area/Volume [%]	100	100
Mapped Elements [%]	100	100
Mapped Nodes [%]	100	100
Stator Temperature		
Mapped Area/Volume [%]	90	>99
Mapped Elements [%]	72	>99
Mapped Nodes [%]	51	80
Magnet		
Magnet Losses		
Mapped Area/Volume [%]	100	100
Mapped Elements [%]	100	100
Mapped Nodes [%]	100	100
Magnet Temperature		
Mapped Area/Volume [%]	100	100
Mapped Elements [%]	100	100
Mapped Nodes [%]	100	100

Postprocessing System Coupling's Results

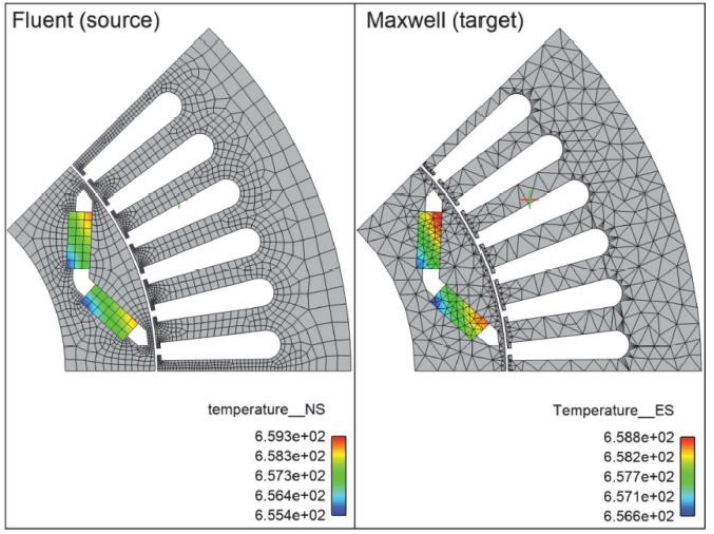
- Evaluate Mapping Quality
  - Review Mapping in the Transcript
  - Visualize Temperature Mapping Quality in EnSight
- Verify the Application of Fluent-Generated Temperatures

MAPPING SUMMARY

	Source	Target
Rotor		
Rotor Losses		
Mapped Area/Volume [%]	100	100
Mapped Elements [%]	100	100
Mapped Nodes [%]	100	100
Rotor Temperature		
Mapped Area/Volume [%]	91	100
Mapped Elements [%]	84	100
Mapped Nodes [%]	60	98
Stator		
Stator Losses		
Mapped Area/Volume [%]	100	100
Mapped Elements [%]	100	100
Mapped Nodes [%]	100	100
Stator Temperature		
Mapped Area/Volume [%]	90	>99
Mapped Elements [%]	72	>99
Mapped Nodes [%]	51	80
Magnet		
Magnet Losses		
Mapped Area/Volume [%]	100	100
Mapped Elements [%]	100	100
Mapped Nodes [%]	100	100
Magnet Temperature		
Mapped Area/Volume [%]	100	100
Mapped Elements [%]	100	100
Mapped Nodes [%]	100	100

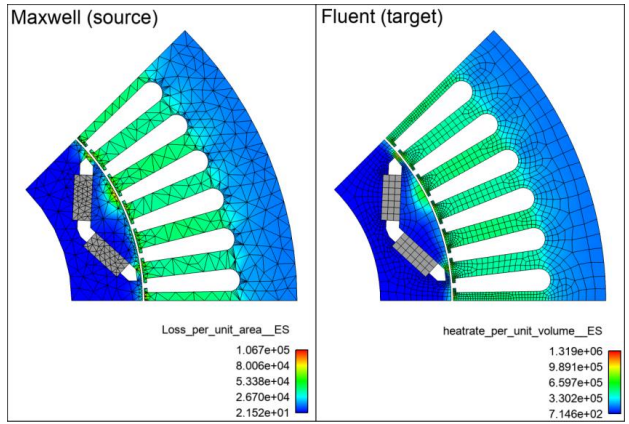
COUPLING ITERATION = 4

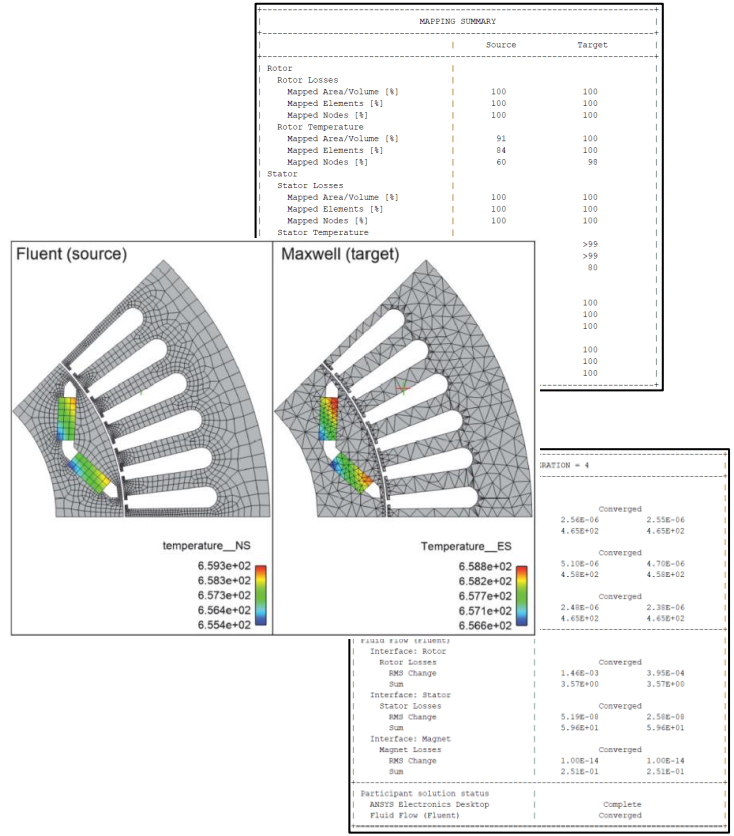
ANSYS Electronics Desktop		
Interface: Rotor		
Rotor Temperature	Converged	
RMS Change	2.56E-06	2.55E-06
Weighted Average	4.65E+02	4.65E+02
Interface: Stator		
Stator Temperature	Converged	
RMS Change	5.10E-06	4.70E-06
Weighted Average	4.58E+02	4.58E+02
Interface: Magnet		
Magnet Temperature	Converged	
RMS Change	2.48E-06	2.38E-06
Weighted Average	4.65E+02	4.65E+02
Fluid Flow (Fluent)		
Interface: Rotor		
Rotor Losses	Converged	
RMS Change	1.46E-03	3.95E-04
Sum	3.57E+00	3.57E+00
Interface: Stator		
Stator Losses	Converged	
RMS Change	5.19E-08	2.58E-08
Sum	5.96E+01	5.96E+01
Interface: Magnet		
Magnet Losses	Converged	
RMS Change	1.00E-14	1.00E-14
Sum	2.51E-01	2.51E-01
Participant solution status		
ANSYS Electronics Desktop	Complete	
Fluid Flow (Fluent)	Converged	



Postprocessing System Coupling's Results

- Evaluate Mapping Quality
  - Review Mapping in the Transcript
  - Visualize Temperature Mapping Quality in EnSight
- Verify the Application of Fluent-Generated Temperatures
- Verify the Application of Maxwell-Generated Losses

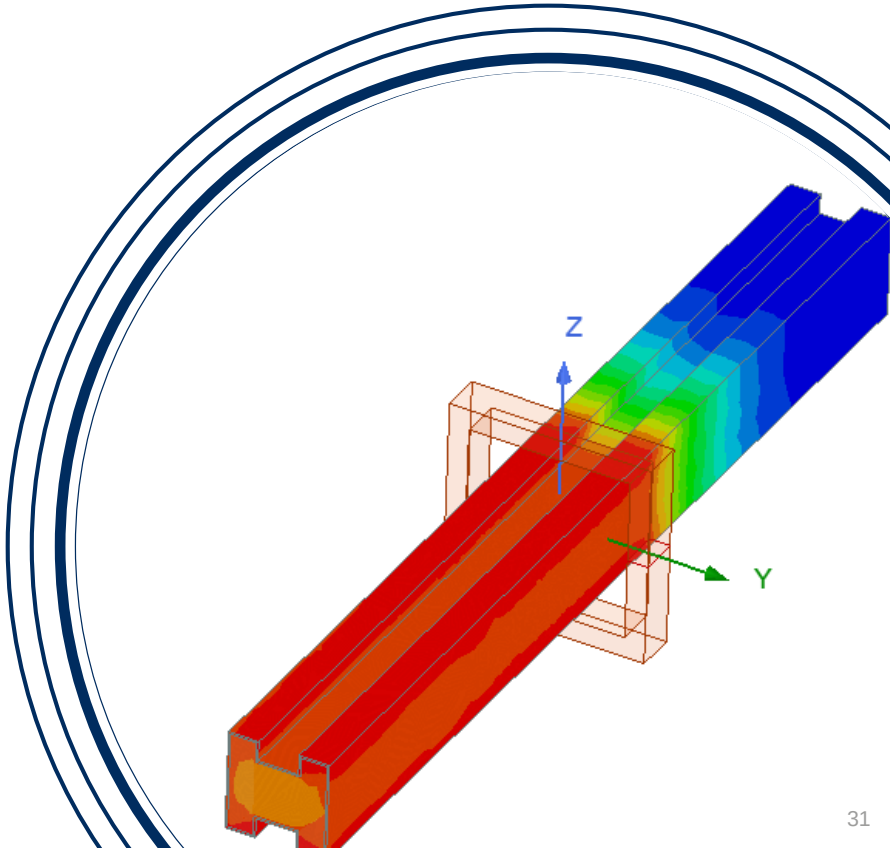




Inducting Heating of a H-Beam

Task:

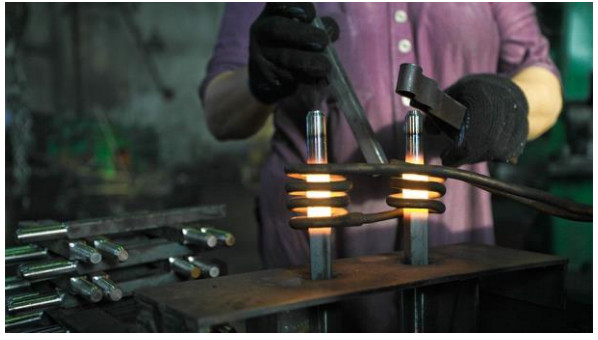
An electrically conductive H-beam is moved through an excitation coil which induced eddy currents in the beam. The eddy currents leads to a heating process in the beam.

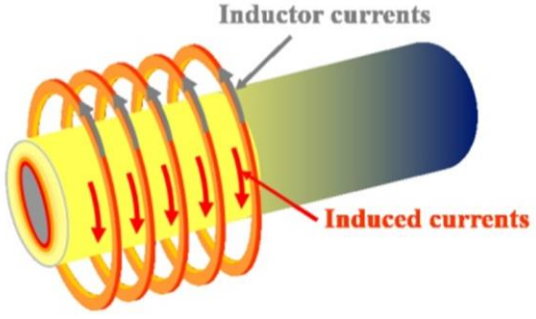


CADFEM Ansys CERTIFIED ELITE CHANNEL PARTNER

Background – Induction Heating

- Use of high frequency AC currents to induce eddy current losses in conducting objects
  - For industrial applications, typical frequency is in the range of kHz (up to few hundreds).
- Several advantages vs. conventional heating sources:
  - Fast and selective heating cycles
  - Energy savings
  - Reduced oxidation and cleaning
  - Process repeatability
- The process works only with conductive materials (usually metals)
  - Materials with high permeability (i. e. ferromagnetic materials) are easier to heat
- Frequency used depends on the workpiece size, desired penetration depth and material type





ANSYS, Induction Heating Coupled Simulation Workflow, Davide Frigerio, Senior Engineer

Coupled Simulation Process

Physics

- Electro-magnetic and Thermo-mechanics must be considered as coupled physics

Time Scales

- Since the time scale of the phenomena is order of magnitudes different among physics, the most effective approach is to consider each of them as quasi-static

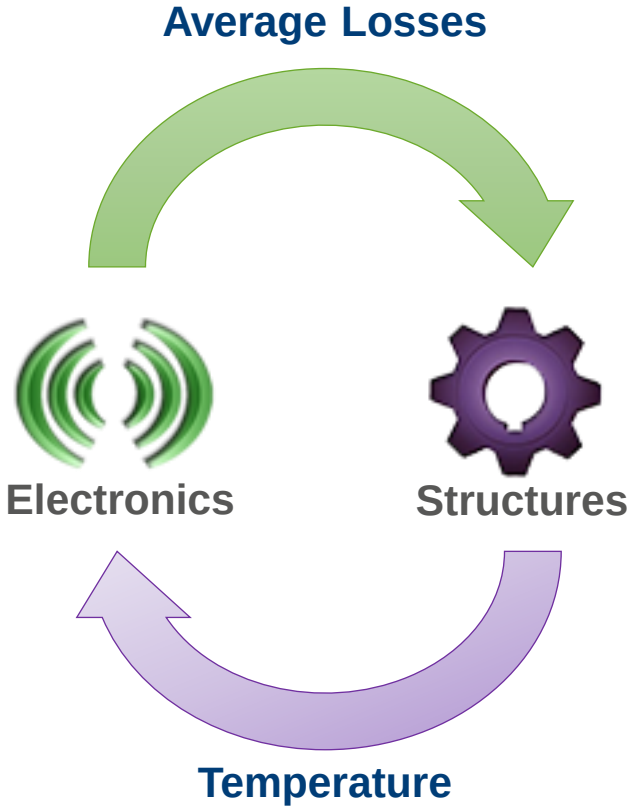
Data Transfer

- Massive data transfer and synchronization are required in order to complete successfully the analysis

Physics

Maxwell

- Quasi-static (Eddy Current Solver)
  - Sinus excitation
- Materials
  - Temperature dependent properties
  - Linear magnetic properties
- Data transfer
  - Import temperature
  - Export average losses
- System Coupling Setup
  - Steady results



Mechanical

- Transient thermal
  - Convection boundary
- Materials
  - Temperature dependent properties
- Enable data transfer
  - Import Losses
  - Export Temperature
  - Time Step

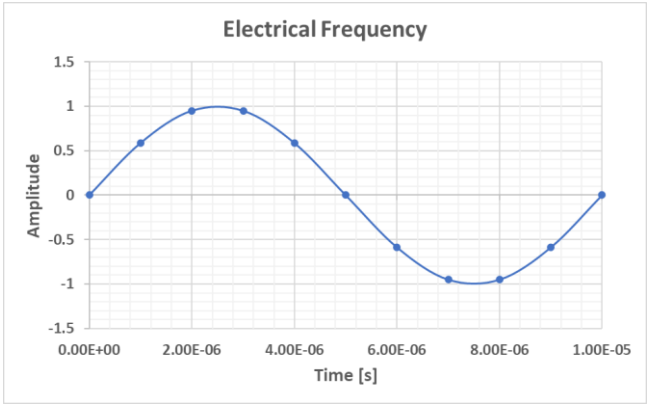
«Include Temperature Dependence» and «Enable Feedback» should be turned on

Temperature dependent properties (permeability, el. conductivity, thermal properties)

Time Scales

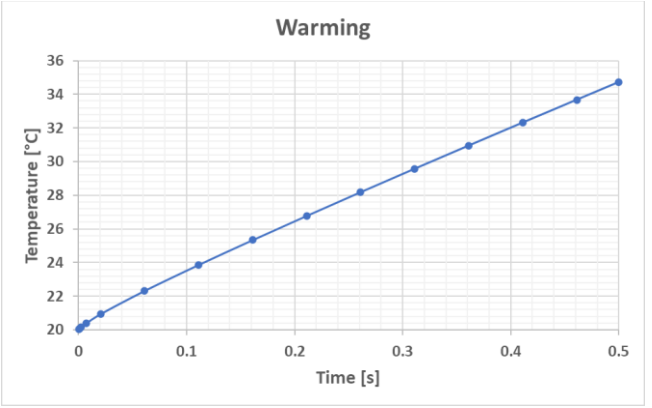
Electromagnetic

- Electric period in the range of µs



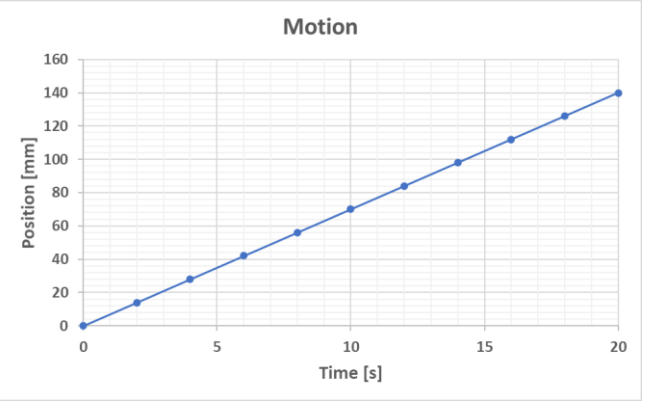
Thermal

- Warm-up rate of a few °C per 100ms



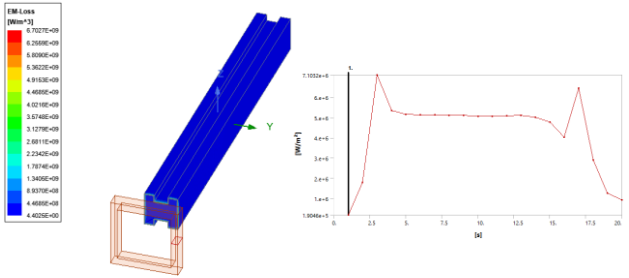
Motion

- Speed a few mm per s

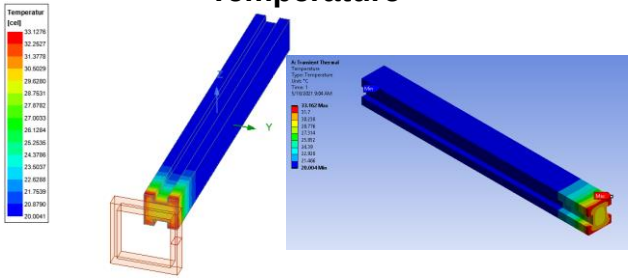


Data Transfer

Losses



Temperature

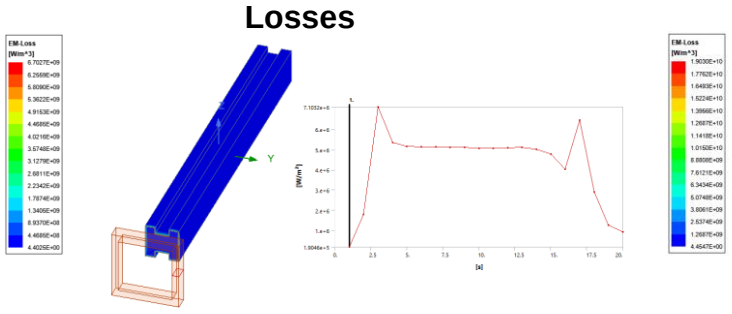


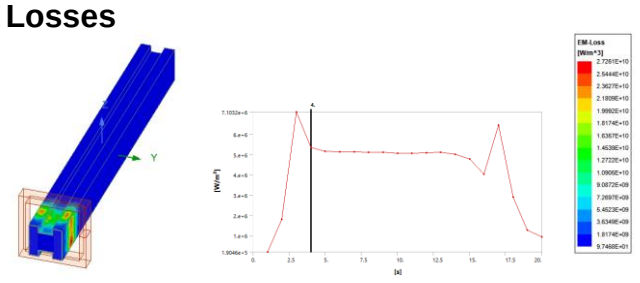
Maxwell → Mechanical

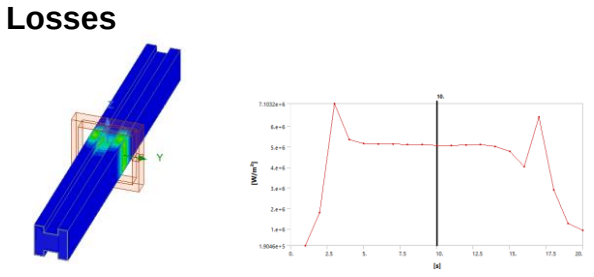
Stop

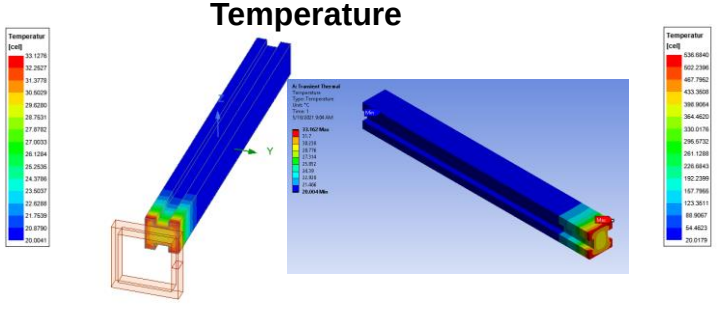
T_{Start}

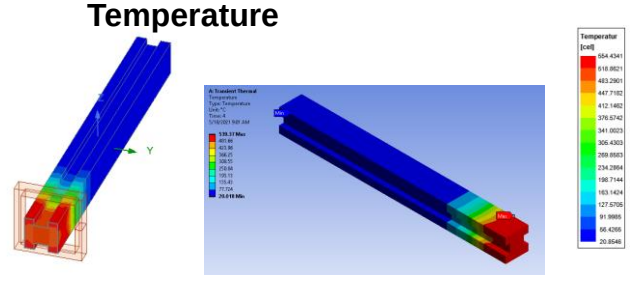
Data Transfer

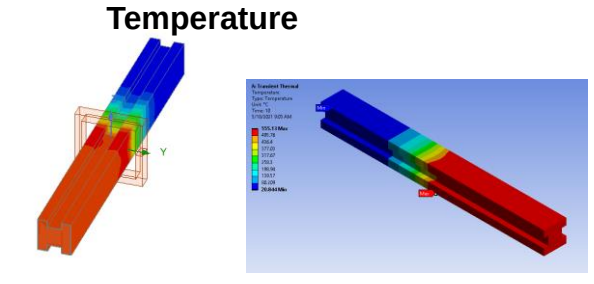


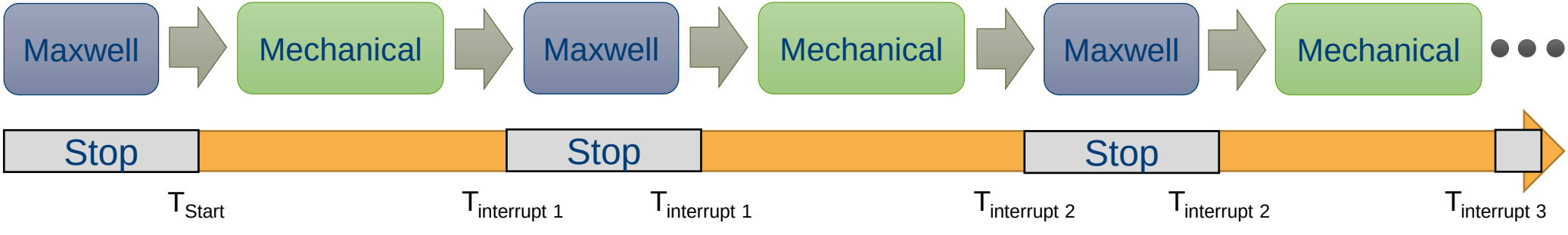






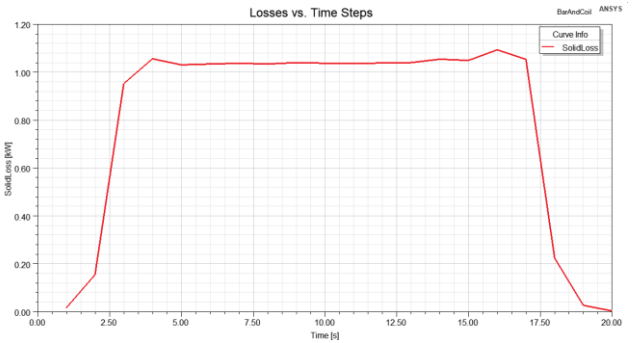


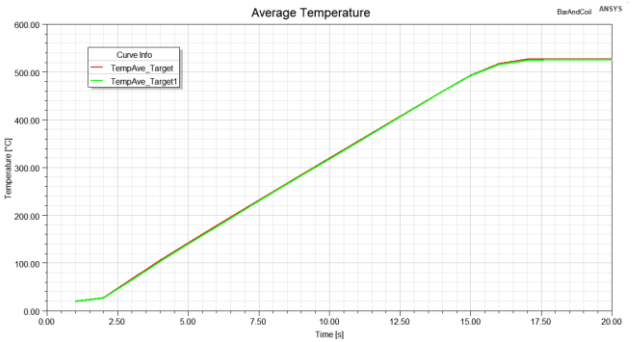


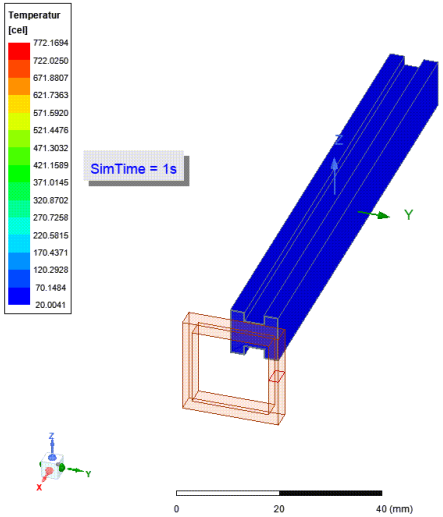


Results

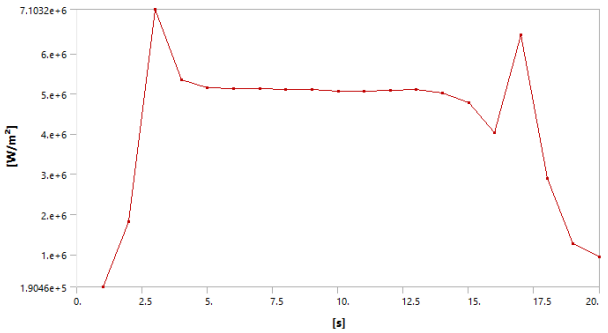
- Maxwell

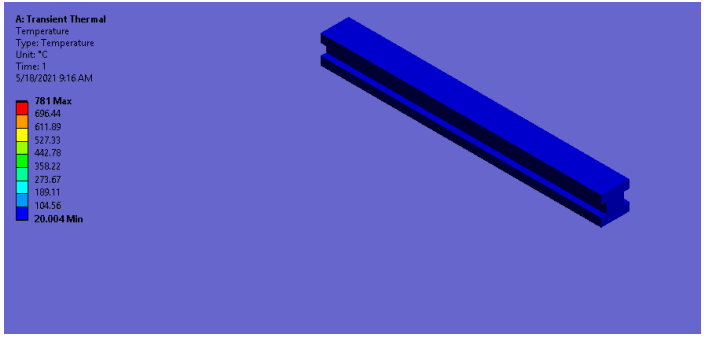






- Mechanical

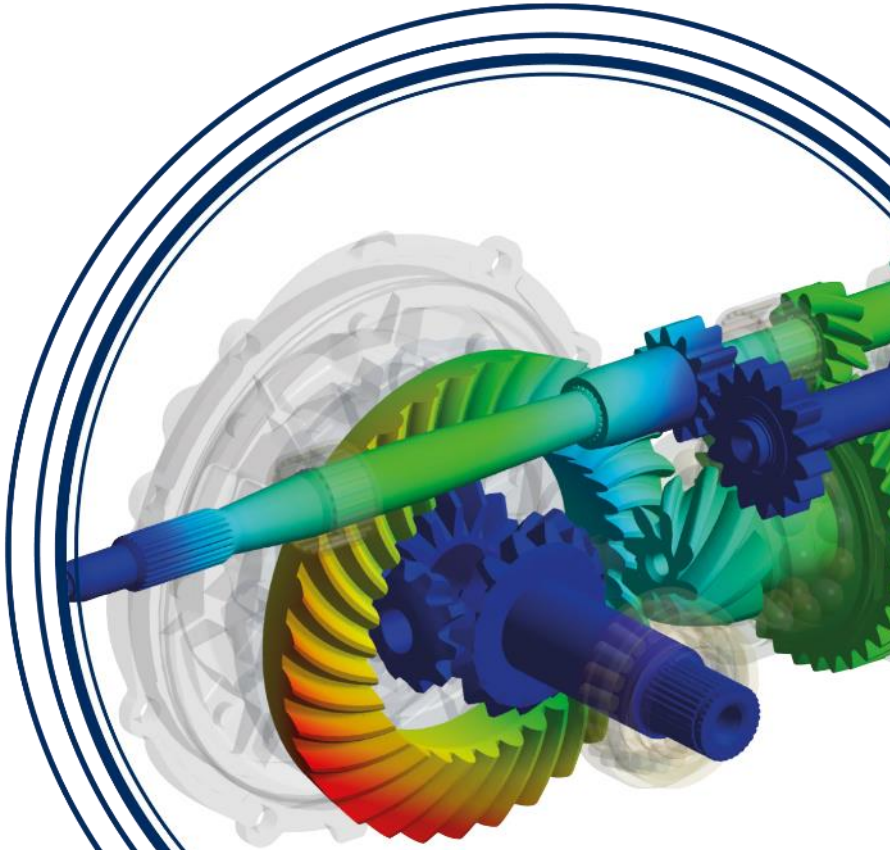




Summary & Outlook

CADFEM

Simulation is more than Software



www.cadfem.net

CADFEM Ansys CERTIFIED ELITE CHANNEL PARTNER