

# ANSYS: Software for Finite Element Analysis (FEA)

Paul J. Atzberger

206D: Finite Element Methods  
University of California Santa Barbara

# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.



# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to



# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling,



# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling, (ii) select correct material parameters,



# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling, (ii) select correct material parameters, (iii) perform meshing and refinements,



# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling, (ii) select correct material parameters, (iii) perform meshing and refinements, (iv) set up studies and quantities of interest,



# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling, (ii) select correct material parameters, (iii) perform meshing and refinements, (iv) set up studies and quantities of interest, and (v) perform parametric studies.



# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling, (ii) select correct material parameters, (iii) perform meshing and refinements, (iv) set up studies and quantities of interest, and (v) perform parametric studies.

**Aims:** Finite Element Analysis (FEA) used to obtain insights into systems before physically building them.



# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling, (ii) select correct material parameters, (iii) perform meshing and refinements, (iv) set up studies and quantities of interest, and (v) perform parametric studies.

**Aims:** Finite Element Analysis (FEA) used to obtain insights into systems before physically building them.

**Current Practice:** FEA is a common part of the engineering design process allowing for rapid prototyping and insights to help reduce costs.



# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling, (ii) select correct material parameters, (iii) perform meshing and refinements, (iv) set up studies and quantities of interest, and (v) perform parametric studies.

**Aims:** Finite Element Analysis (FEA) used to obtain insights into systems before physically building them.

**Current Practice:** FEA is a common part of the engineering design process allowing for rapid prototyping and insights to help reduce costs. While physical builds are still needed for validation, may reduce number of trials.



# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling, (ii) select correct material parameters, (iii) perform meshing and refinements, (iv) set up studies and quantities of interest, and (v) perform parametric studies.



**Aims:** Finite Element Analysis (FEA) used to obtain insights into systems before physically building them.

**Current Practice:** FEA is a common part of the engineering design process allowing for rapid prototyping and insights to help reduce costs. While physical builds are still needed for validation, may reduce number of trials.

**ANSYS:** We will focus on this package. Currently widely used in mechanical, electrical, and other fields of engineering and the sciences.

# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling, (ii) select correct material parameters, (iii) perform meshing and refinements, (iv) set up studies and quantities of interest, and (v) perform parametric studies.



**Aims:** Finite Element Analysis (FEA) used to obtain insights into systems before physically building them.

**Current Practice:** FEA is a common part of the engineering design process allowing for rapid prototyping and insights to help reduce costs. While physical builds are still needed for validation, may reduce number of trials.

**ANSYS:** We will focus on this package. Currently widely used in mechanical, electrical, and other fields of engineering and the sciences.

**Availability:** ANSYS is installed on university computer labs in engineering and you can request access. You also can download a student edition.

# Software for Finite Element Analysis (FEA)

**Software:** Many software solvers and interfaces developed encapsulating Finite Element Methods.

**Features:** Provide ways to (i) perform geometric modeling, (ii) select correct material parameters, (iii) perform meshing and refinements, (iv) set up studies and quantities of interest, and (v) perform parametric studies.



**Aims:** Finite Element Analysis (FEA) used to obtain insights into systems before physically building them.

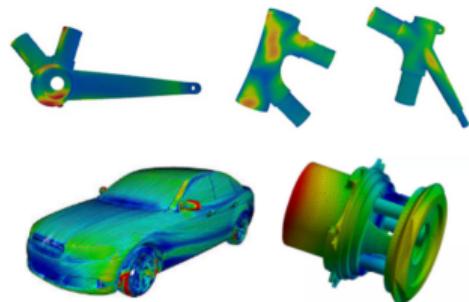
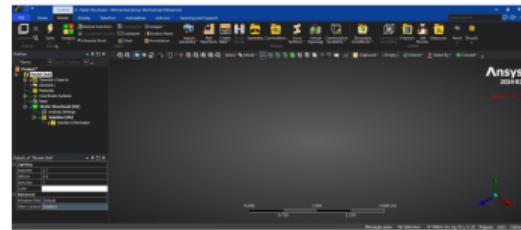
**Current Practice:** FEA is a common part of the engineering design process allowing for rapid prototyping and insights to help reduce costs. While physical builds are still needed for validation, may reduce number of trials.

**ANSYS:** We will focus on this package. Currently widely used in mechanical, electrical, and other fields of engineering and the sciences.

**Availability:** ANSYS is installed on university computer labs in engineering and you can request access. You also can download a student edition.

# ANSYS: Software

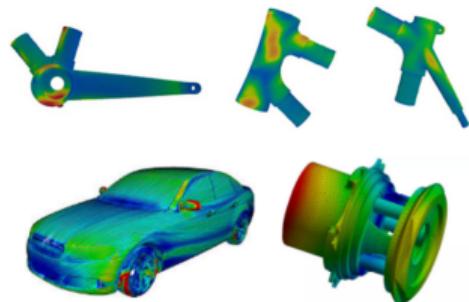
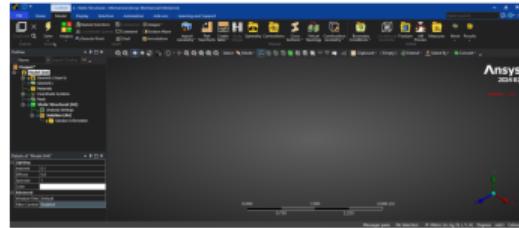
Provides software for design, modeling, and analysis to aid in engineering systems.



# ANSYS: Software

Provides software for design, modeling, and analysis to aid in engineering systems.

Comprised of several components and modules

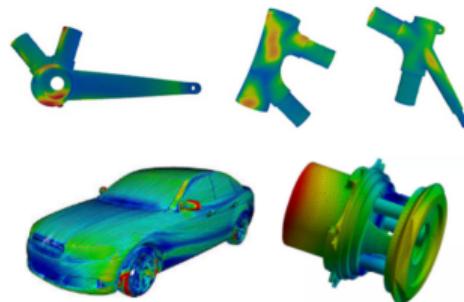


# ANSYS: Software

Provides software for design, modeling, and analysis to aid in engineering systems.

Comprised of several components and modules

- **Workbench:** general workflow for design, analysis, and coordinating between tasks.

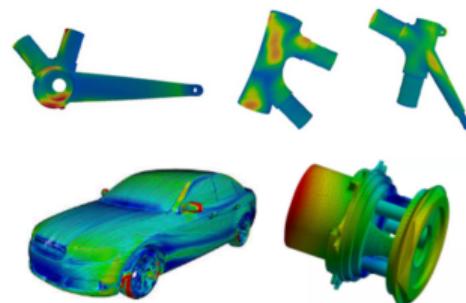


# ANSYS: Software

Provides software for design, modeling, and analysis to aid in engineering systems.

Comprised of several components and modules

- **Workbench:** general workflow for design, analysis, and coordinating between tasks.
- **Mechanical:** structural and thermal analysis.

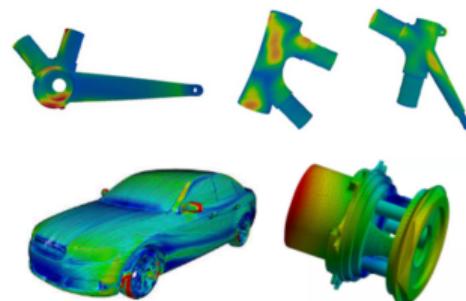
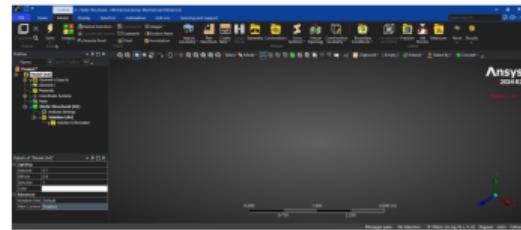


# ANSYS: Software

Provides software for design, modeling, and analysis to aid in engineering systems.

Comprised of several components and modules

- **Workbench:** general workflow for design, analysis, and coordinating between tasks.
- **Mechanical:** structural and thermal analysis.
- **Fluent:** fluid mechanics (CFD).

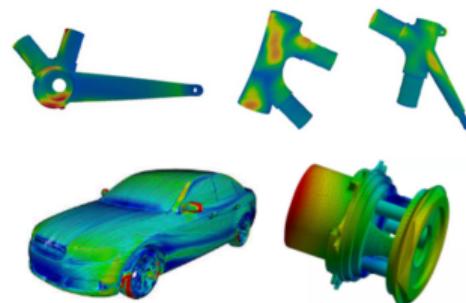
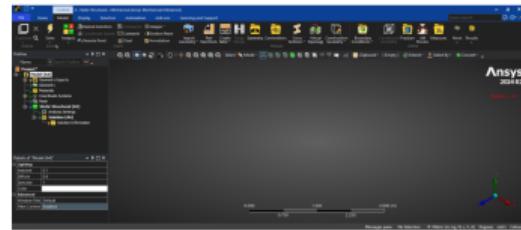


# ANSYS: Software

Provides software for design, modeling, and analysis to aid in engineering systems.

Comprised of several components and modules

- **Workbench:** general workflow for design, analysis, and coordinating between tasks.
- **Mechanical:** structural and thermal analysis.
- **Fluent:** fluid mechanics (CFD).
- **HFSS:** high-frequency electromagnetics.

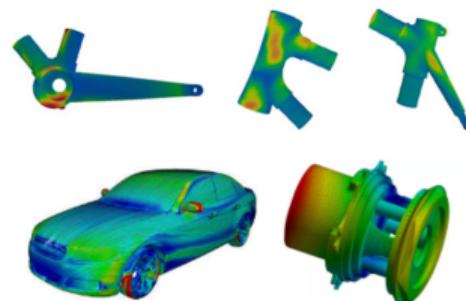
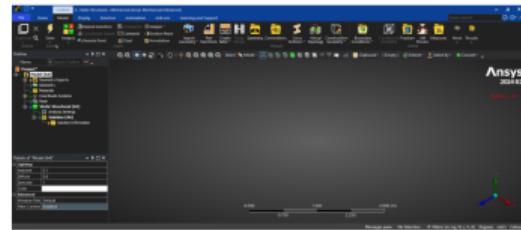


# ANSYS: Software

Provides software for design, modeling, and analysis to aid in engineering systems.

Comprised of several components and modules

- **Workbench:** general workflow for design, analysis, and coordinating between tasks.
- **Mechanical:** structural and thermal analysis.
- **Fluent:** fluid mechanics (CFD).
- **HFSS:** high-frequency electromagnetics.
- and many other components.



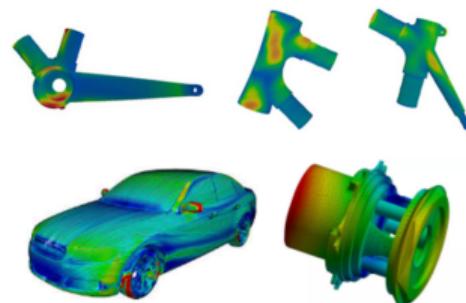
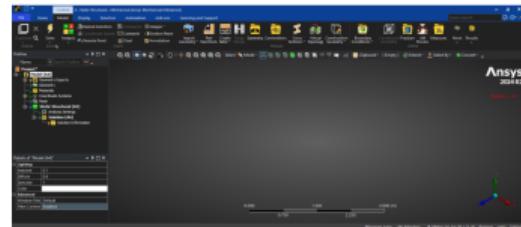
# ANSYS: Software

Provides software for design, modeling, and analysis to aid in engineering systems.

Comprised of several components and modules

- **Workbench:** general workflow for design, analysis, and coordinating between tasks.
- **Mechanical:** structural and thermal analysis.
- **Fluent:** fluid mechanics (CFD).
- **HFSS:** high-frequency electromagnetics.
- and many other components.

**ANSYS Mechanical** finite element methods will be our primary focus. Other components are used similarly.



# ANSYS: Software

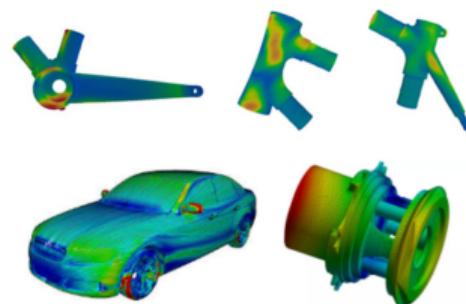
Provides software for design, modeling, and analysis to aid in engineering systems.

Comprised of several components and modules

- **Workbench:** general workflow for design, analysis, and coordinating between tasks.
- **Mechanical:** structural and thermal analysis.
- **Fluent:** fluid mechanics (CFD).
- **HFSS:** high-frequency electromagnetics.
- and many other components.

**ANSYS Mechanical** finite element methods will be our primary focus. Other components are used similarly.

Provides many integrated tools for design, modeling, simulation, analysis, and other tasks.



# ANSYS: Software

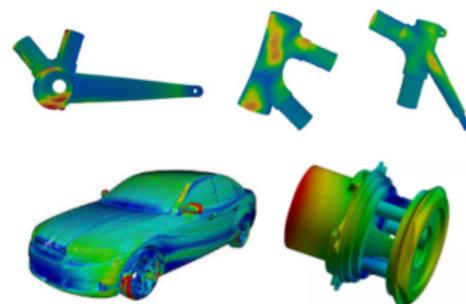
Provides software for design, modeling, and analysis to aid in engineering systems.

Comprised of several components and modules

- **Workbench:** general workflow for design, analysis, and coordinating between tasks.
- **Mechanical:** structural and thermal analysis.
- **Fluent:** fluid mechanics (CFD).
- **HFSS:** high-frequency electromagnetics.
- and many other components.

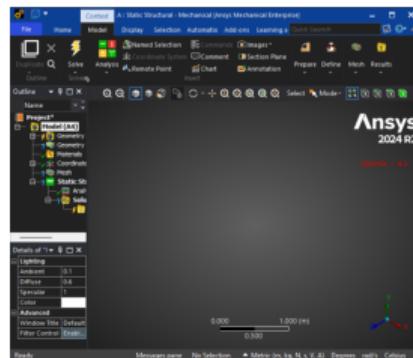
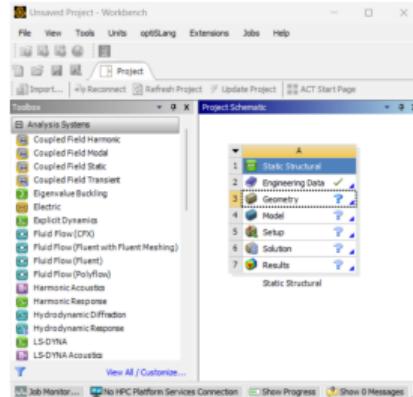
**ANSYS Mechanical** finite element methods will be our primary focus. Other components are used similarly.

Provides many integrated tools for design, modeling, simulation, analysis, and other tasks.



# ANSYS Mechanical: Setting Up FEA Studies

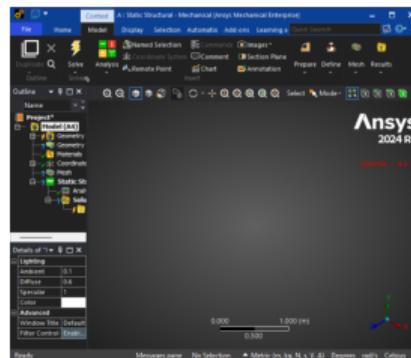
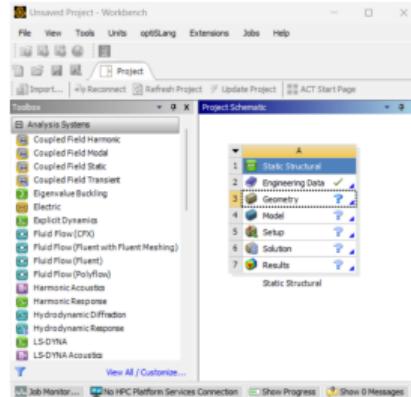
**Outline:** Creating a new FEA study follows these steps:



# ANSYS Mechanical: Setting Up FEA Studies

**Outline:** Creating a new FEA study follows these steps:

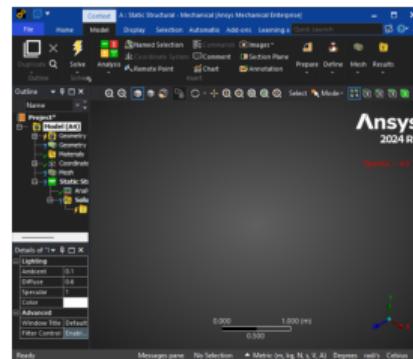
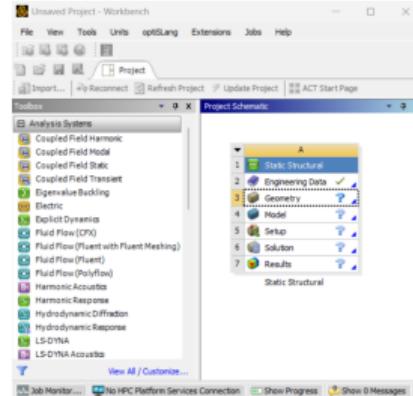
1. **Open Workbench:** Select "Static Structural" and drag to the workflow space.



# ANSYS Mechanical: Setting Up FEA Studies

**Outline:** Creating a new FEA study follows these steps:

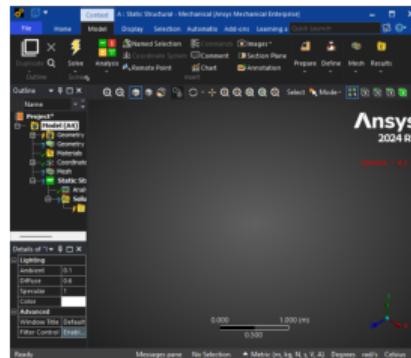
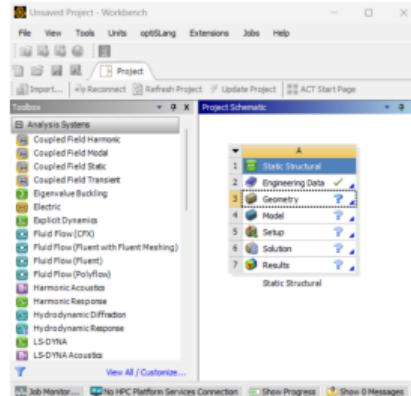
- 1. Open Workbench:** Select "Static Structural" and drag to the workflow space.
- 2. Engineering Data:** Specify material properties and other settings.



# ANSYS Mechanical: Setting Up FEA Studies

**Outline:** Creating a new FEA study follows these steps:

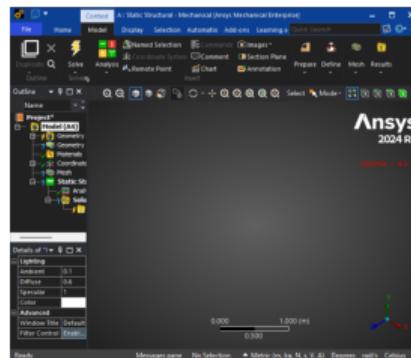
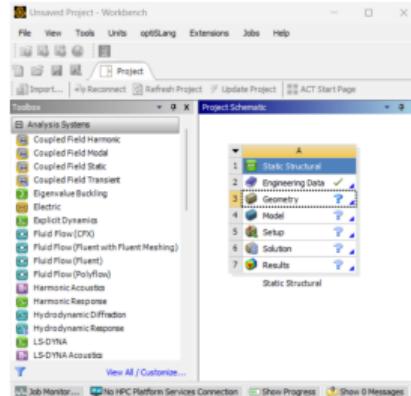
- 1. Open Workbench:** Select "Static Structural" and drag to the workflow space.
- 2. Engineering Data:** Specify material properties and other settings.
- 3. Geometry:** Build the geometry and designate properties.



# ANSYS Mechanical: Setting Up FEA Studies

**Outline:** Creating a new FEA study follows these steps:

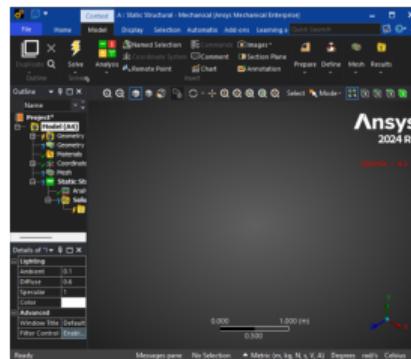
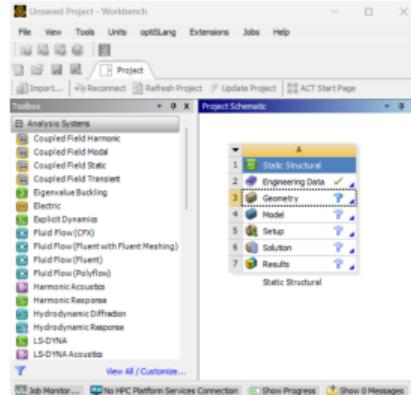
- 1. Open Workbench:** Select "Static Structural" and drag to the workflow space.
- 2. Engineering Data:** Specify material properties and other settings.
- 3. Geometry:** Build the geometry and designate properties.
- 4. Model:** Define boundary conditions and specify analysis to be performed.



# ANSYS Mechanical: Setting Up FEA Studies

**Outline:** Creating a new FEA study follows these steps:

1. **Open Workbench:** Select "Static Structural" and drag to the workflow space.
2. **Engineering Data:** Specify material properties and other settings.
3. **Geometry:** Build the geometry and designate properties.
4. **Model:** Define boundary conditions and specify analysis to be performed.
5. **Results:** Run calculations and visualize the outcomes.

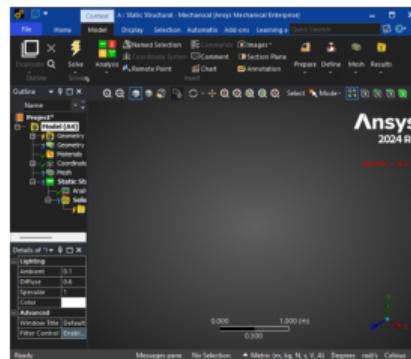
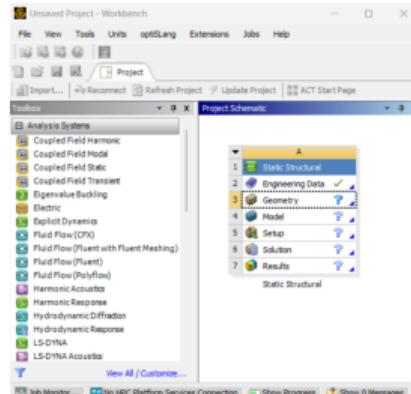


# ANSYS Mechanical: Setting Up FEA Studies

**Outline:** Creating a new FEA study follows these steps:

- 1. Open Workbench:** Select "Static Structural" and drag to the workflow space.
- 2. Engineering Data:** Specify material properties and other settings.
- 3. Geometry:** Build the geometry and designate properties.
- 4. Model:** Define boundary conditions and specify analysis to be performed.
- 5. Results:** Run calculations and visualize the outcomes.

More details will be given in our live demonstrations.



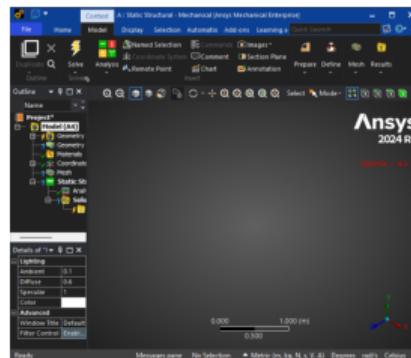
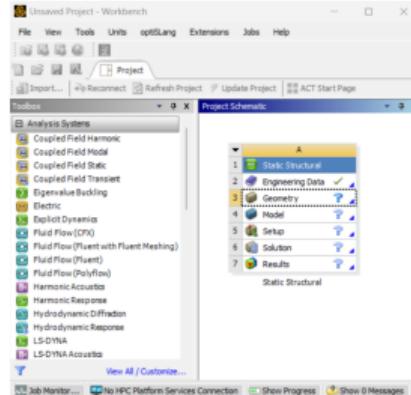
# ANSYS Mechanical: Setting Up FEA Studies

**Outline:** Creating a new FEA study follows these steps:

- 1. Open Workbench:** Select "Static Structural" and drag to the workflow space.
- 2. Engineering Data:** Specify material properties and other settings.
- 3. Geometry:** Build the geometry and designate properties.
- 4. Model:** Define boundary conditions and specify analysis to be performed.
- 5. Results:** Run calculations and visualize the outcomes.

More details will be given in our live demonstrations.

Many tutorials can be found on ANSYS website and at the links we provided.



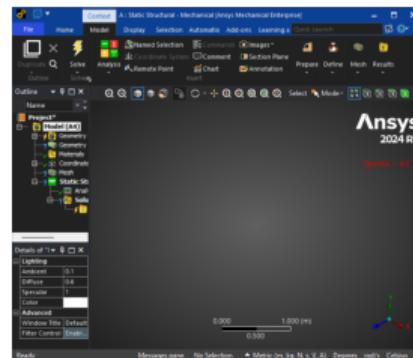
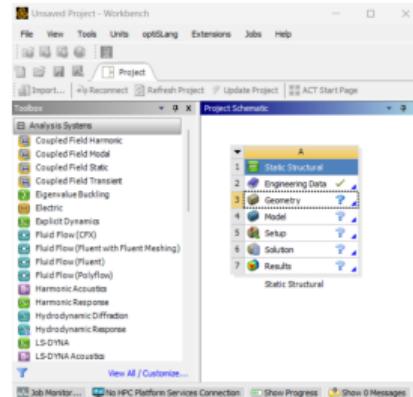
# ANSYS Mechanical: Setting Up FEA Studies

**Outline:** Creating a new FEA study follows these steps:

- 1. Open Workbench:** Select "Static Structural" and drag to the workflow space.
- 2. Engineering Data:** Specify material properties and other settings.
- 3. Geometry:** Build the geometry and designate properties.
- 4. Model:** Define boundary conditions and specify analysis to be performed.
- 5. Results:** Run calculations and visualize the outcomes.

More details will be given in our live demonstrations.

Many tutorials can be found on ANSYS website and at the links we provided.

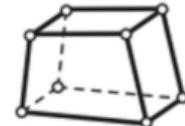


# ANSYS: Finite Element Types Utilized

FEM: The most common element types are



**Tet4**



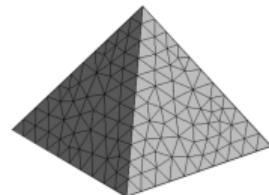
**Hex8**



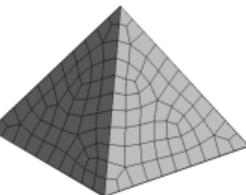
**Tet10**



**Hex20**



**Tet4**



**Hex8**

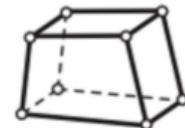
# ANSYS: Finite Element Types Utilized

FEM: The most common element types are

- **Tet4**: tetrahedral linear element with 4 nodes.
- **Hex8**: hexahedral linear element with 8 nodes.



**Tet4**



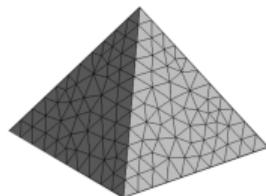
**Hex8**



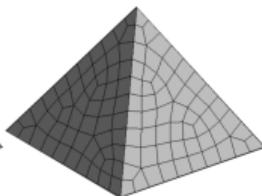
**Tet10**



**Hex20**



**Tet4**



**Hex8**

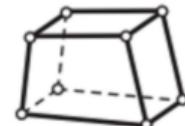
# ANSYS: Finite Element Types Utilized

FEM: The most common element types are

- **Tet4**: tetrahedral linear element with 4 nodes.
- **Hex8**: hexahedral linear element with 8 nodes.
- **Tet10**: tetrahedral quadratic element with 10 nodes.
- **Hex20**: hexahedral quadratic element with 20 nodes.



**Tet4**



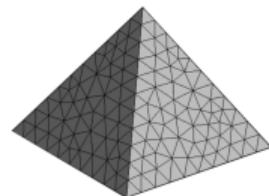
**Hex8**



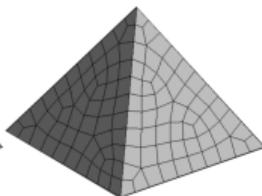
**Tet10**



**Hex20**



**Tet4**



**Hex8**

# ANSYS: Finite Element Types Utilized

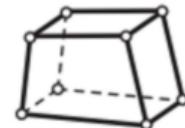
FEM: The most common element types are

- **Tet4**: tetrahedral linear element with 4 nodes.
- **Hex8**: hexahedral linear element with 8 nodes.
- **Tet10**: tetrahedral quadratic element with 10 nodes.
- **Hex20**: hexahedral quadratic element with 20 nodes.

Lagrange elements using polynomial function space  $\mathcal{P}_k$ .



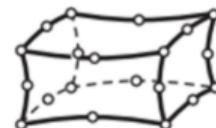
**Tet4**



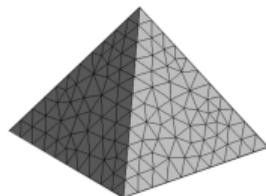
**Hex8**



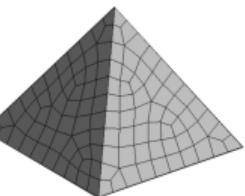
**Tet10**



**Hex20**



**Tet4**



**Hex8**

# ANSYS: Finite Element Types Utilized

**FEM:** The most common element types are

- **Tet4:** tetrahedral linear element with 4 nodes.
- **Hex8:** hexahedral linear element with 8 nodes.
- **Tet10:** tetrahedral quadratic element with 10 nodes.
- **Hex20:** hexahedral quadratic element with 20 nodes.

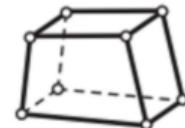
**Lagrange elements** using polynomial function space  $\mathcal{P}_k$ .

**Nodal variables** are evaluated at the locations shown on the right,

$$u(x) = \sum_{i=1}^N u_i \phi_i(x_i).$$



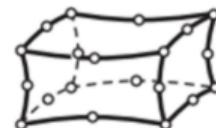
**Tet4**



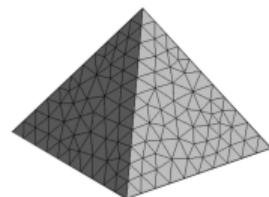
**Hex8**



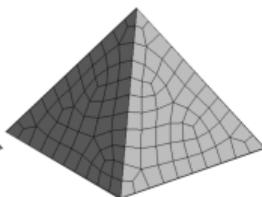
**Tet10**



**Hex20**



**Tet4**



**Hex8**

# ANSYS: Finite Element Types Utilized

**FEM:** The most common element types are

- **Tet4:** tetrahedral linear element with 4 nodes.
- **Hex8:** hexahedral linear element with 8 nodes.
- **Tet10:** tetrahedral quadratic element with 10 nodes.
- **Hex20:** hexahedral quadratic element with 20 nodes.

**Lagrange elements** using polynomial function space  $\mathcal{P}_k$ .

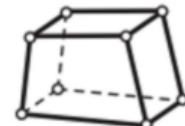
**Nodal variables** are evaluated at the locations shown on the right,

$$u(x) = \sum_{i=1}^N u_i \phi_i(x_i).$$

**Isoparametric** elements with same shape space for geometry.



**Tet4**



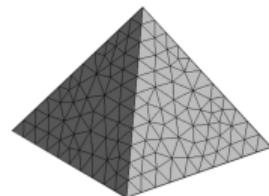
**Hex8**



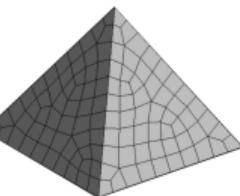
**Tet10**



**Hex20**



**Tet4**



**Hex8**

# ANSYS: Finite Element Types Utilized

**FEM:** The most common element types are

- **Tet4:** tetrahedral linear element with 4 nodes.
- **Hex8:** hexahedral linear element with 8 nodes.
- **Tet10:** tetrahedral quadratic element with 10 nodes.
- **Hex20:** hexahedral quadratic element with 20 nodes.

**Lagrange elements** using polynomial function space  $\mathcal{P}_k$ .

**Nodal variables** are evaluated at the locations shown on the right,

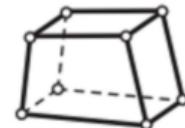
$$u(x) = \sum_{i=1}^N u_i \phi_i(x_i).$$

**Isoparametric** elements with same shape space for geometry.

**Element choice** involves trade-offs



**Tet4**



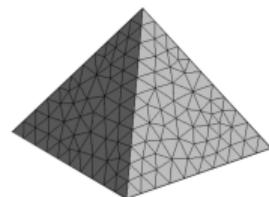
**Hex8**



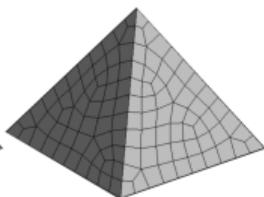
**Tet10**



**Hex20**



**Tet4**



**Hex8**

# ANSYS: Finite Element Types Utilized

**FEM:** The most common element types are

- **Tet4:** tetrahedral linear element with 4 nodes.
- **Hex8:** hexahedral linear element with 8 nodes.
- **Tet10:** tetrahedral quadratic element with 10 nodes.
- **Hex20:** hexahedral quadratic element with 20 nodes.

**Lagrange elements** using polynomial function space  $\mathcal{P}_k$ .

**Nodal variables** are evaluated at the locations shown on the right,

$$u(x) = \sum_{i=1}^N u_i \phi_i(x_i).$$

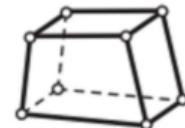
**Isoparametric** elements with same shape space for geometry.

**Element choice** involves trade-offs

- (i) lower-order elements less expensive but less accurate.



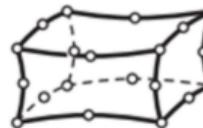
**Tet4**



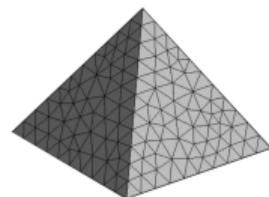
**Hex8**



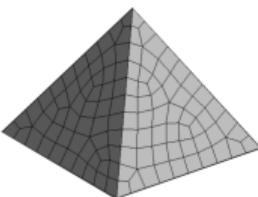
**Tet10**



**Hex20**



**Tet4**



**Hex8**

# ANSYS: Finite Element Types Utilized

**FEM:** The most common element types are

- **Tet4:** tetrahedral linear element with 4 nodes.
- **Hex8:** hexahedral linear element with 8 nodes.
- **Tet10:** tetrahedral quadratic element with 10 nodes.
- **Hex20:** hexahedral quadratic element with 20 nodes.

**Lagrange elements** using polynomial function space  $\mathcal{P}_k$ .

**Nodal variables** are evaluated at the locations shown on the right,

$$u(x) = \sum_{i=1}^N u_i \phi_i(x_i).$$

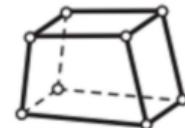
**Isoparametric** elements with same shape space for geometry.

**Element choice** involves trade-offs

- lower-order elements less expensive but less accurate.
- higher-order elements are more expensive, but can better accommodate local variations.



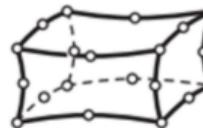
**Tet4**



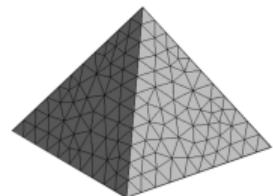
**Hex8**



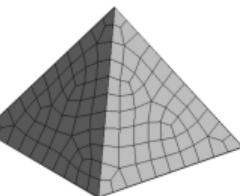
**Tet10**



**Hex20**



**Tet4**



**Hex8**

# ANSYS: Finite Element Types Utilized

**FEM:** The most common element types are

- **Tet4:** tetrahedral linear element with 4 nodes.
- **Hex8:** hexahedral linear element with 8 nodes.
- **Tet10:** tetrahedral quadratic element with 10 nodes.
- **Hex20:** hexahedral quadratic element with 20 nodes.

**Lagrange elements** using polynomial function space  $\mathcal{P}_k$ .

**Nodal variables** are evaluated at the locations shown on the right,

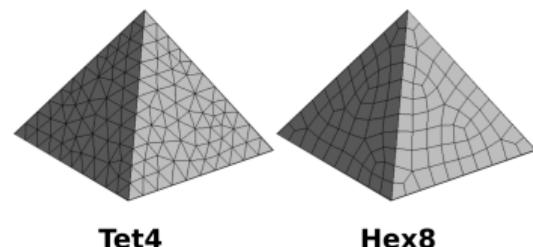
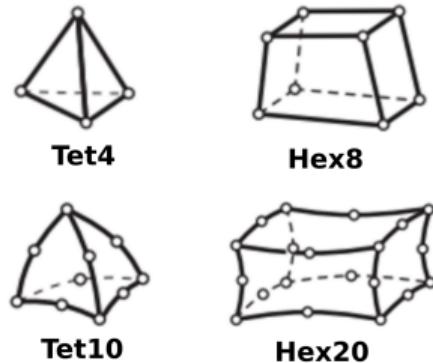
$$u(x) = \sum_{i=1}^N u_i \phi_i(x_i).$$

**Isoparametric** elements with same shape space for geometry.

**Element choice** involves trade-offs

- lower-order elements less expensive but less accurate.
- higher-order elements are more expensive, but can better accommodate local variations.

**Meshing** quality is also important to ensure reliable results.



# ANSYS: Finite Element Types Utilized

**FEM:** The most common element types are

- **Tet4:** tetrahedral linear element with 4 nodes.
- **Hex8:** hexahedral linear element with 8 nodes.
- **Tet10:** tetrahedral quadratic element with 10 nodes.
- **Hex20:** hexahedral quadratic element with 20 nodes.

**Lagrange elements** using polynomial function space  $\mathcal{P}_k$ .

**Nodal variables** are evaluated at the locations shown on the right,

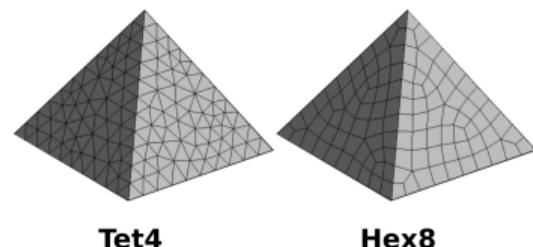
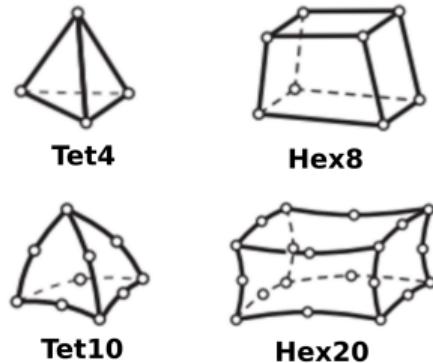
$$u(x) = \sum_{i=1}^N u_i \phi_i(x_i).$$

**Isoparametric** elements with same shape space for geometry.

**Element choice** involves trade-offs

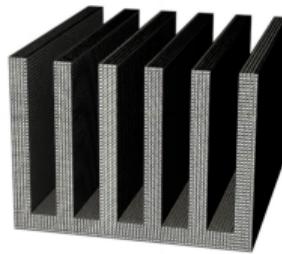
- lower-order elements less expensive but less accurate.
- higher-order elements are more expensive, but can better accommodate local variations.

**Meshing** quality is also important to ensure reliable results.

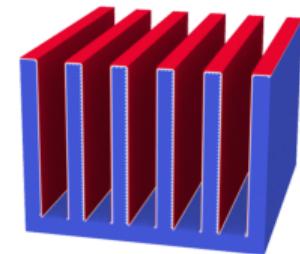


# Example: Thermal Analysis of Heat Exchanger

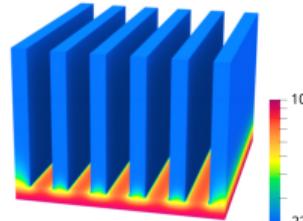
**Problem:** Compute the thermal transfer for a given heat-exchanger geometry.



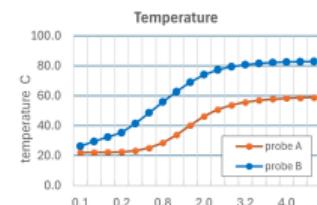
Mesh



Boundary Conditions



Solution



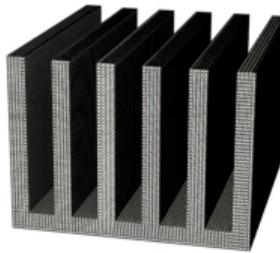
Probes

# Example: Thermal Analysis of Heat Exchanger

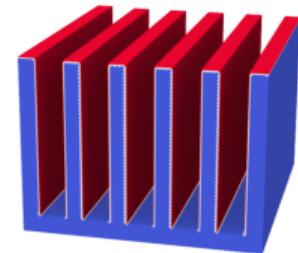
**Problem:** Compute the thermal transfer for a given heat-exchanger geometry.

**Model:**

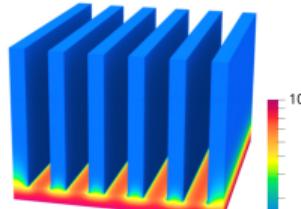
$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$



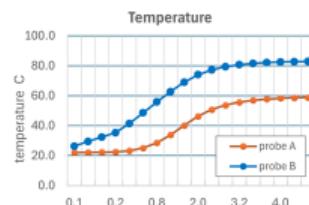
Mesh



Boundary Conditions



Solution



Probes

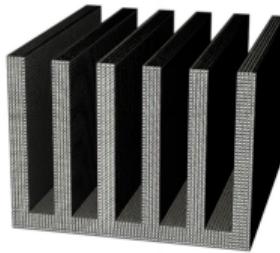
# Example: Thermal Analysis of Heat Exchanger

**Problem:** Compute the thermal transfer for a given heat-exchanger geometry.

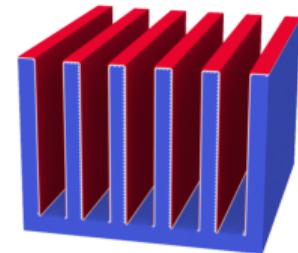
**Model:**

$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$

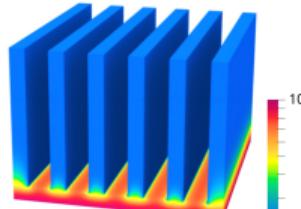
**Geometry** is obtained using CAD tools for sketching and extrusions using the ANSYS design module.



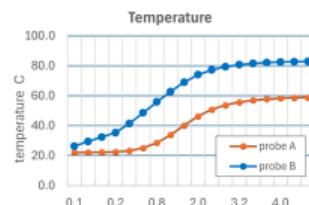
Mesh



Boundary Conditions



Solution



Probes

# Example: Thermal Analysis of Heat Exchanger

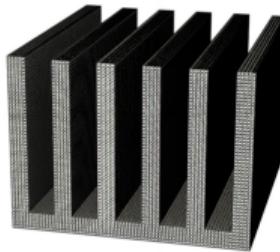
**Problem:** Compute the thermal transfer for a given heat-exchanger geometry.

**Model:**

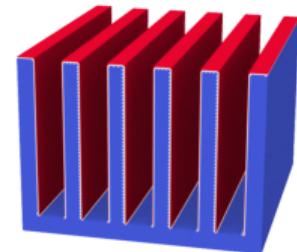
$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$

**Geometry** is obtained using CAD tools for sketching and extrusions using the ANSYS design module.

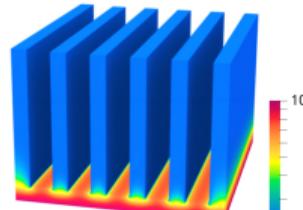
**Meshing** is set up using the modeling component and specified to use Hex8 linear elements.



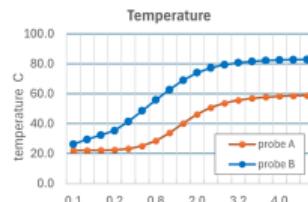
Mesh



Boundary Conditions



Solution



Probes

# Example: Thermal Analysis of Heat Exchanger

**Problem:** Compute the thermal transfer for a given heat-exchanger geometry.

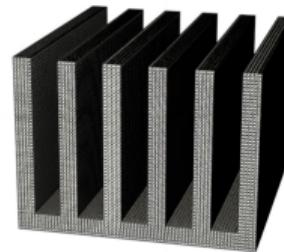
**Model:**

$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$

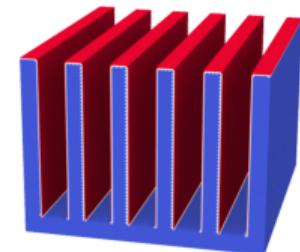
**Geometry** is obtained using CAD tools for sketching and extrusions using the ANSYS design module.

**Meshing** is set up using the modeling component and specified to use Hex8 linear elements.

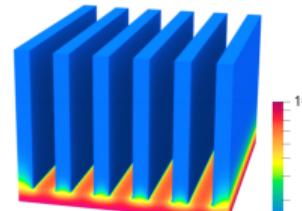
**Boundary Conditions** are set up using the modeling component and are shown with  $h = 0$  in blue and  $g$  in red (right).



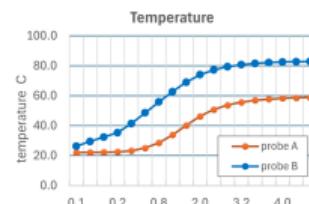
Mesh



Boundary Conditions



Solution



Probes

# Example: Thermal Analysis of Heat Exchanger

**Problem:** Compute the thermal transfer for a given heat-exchanger geometry.

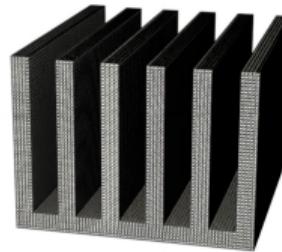
**Model:**

$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$

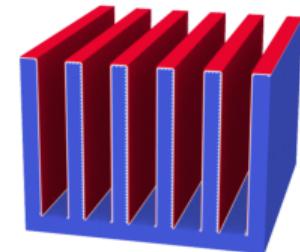
**Geometry** is obtained using CAD tools for sketching and extrusions using the ANSYS design module.

**Meshing** is set up using the modeling component and specified to use Hex8 linear elements.

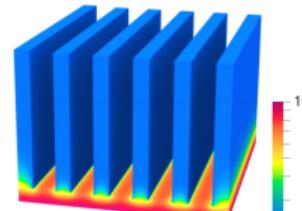
**Boundary Conditions** are set up using the modeling component and are shown with  $h = 0$  in blue and  $g$  in red (right). In particular,



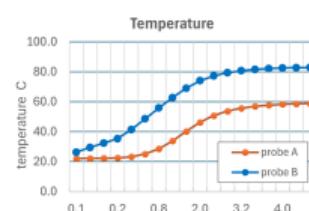
Mesh



Boundary Conditions



Solution



Probes

# Example: Thermal Analysis of Heat Exchanger

**Problem:** Compute the thermal transfer for a given heat-exchanger geometry.

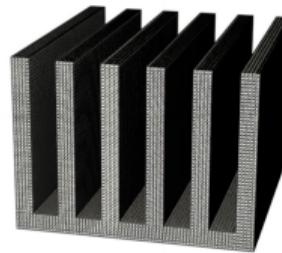
**Model:**

$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$

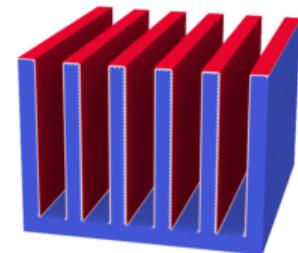
**Geometry** is obtained using CAD tools for sketching and extrusions using the ANSYS design module.

**Meshing** is set up using the modeling component and specified to use Hex8 linear elements.

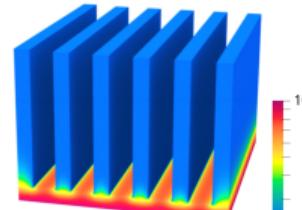
**Boundary Conditions** are set up using the modeling component and are shown with  $h = 0$  in blue and  $g$  in red (right). In particular,  
(i) the bottom surface is set at  $g = 100^\circ \text{ C}$ ,  
(ii) the fin surfaces are set at  $g = 22^\circ \text{ C}$ .



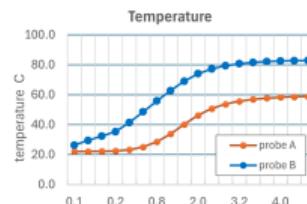
Mesh



Boundary Conditions



Solution



Probes

# Example: Thermal Analysis of Heat Exchanger

**Problem:** Compute the thermal transfer for a given heat-exchanger geometry.

**Model:**

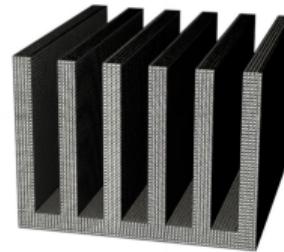
$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$

**Geometry** is obtained using CAD tools for sketching and extrusions using the ANSYS design module.

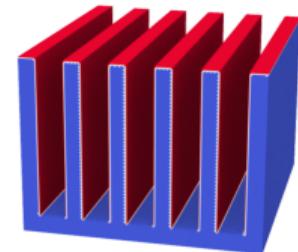
**Meshing** is set up using the modeling component and specified to use Hex8 linear elements.

**Boundary Conditions** are set up using the modeling component and are shown with  $h = 0$  in blue and  $g$  in red (right). In particular,  
(i) the bottom surface is set at  $g = 100^\circ \text{ C}$ ,  
(ii) the fin surfaces are set at  $g = 22^\circ \text{ C}$ .

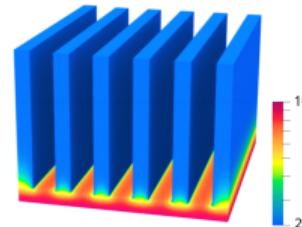
**Results:** The temperature rapidly drops off in this geometry. However, surrounding air can not heat up.



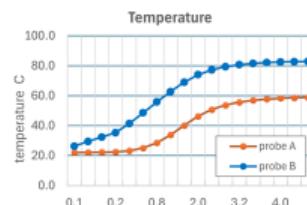
Mesh



Boundary Conditions



Solution



Probes

# Example: Thermal Analysis of Heat Exchanger

**Problem:** Compute the thermal transfer for a given heat-exchanger geometry.

**Model:**

$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$

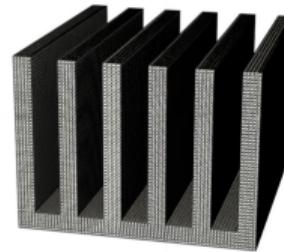
**Geometry** is obtained using CAD tools for sketching and extrusions using the ANSYS design module.

**Meshing** is set up using the modeling component and specified to use Hex8 linear elements.

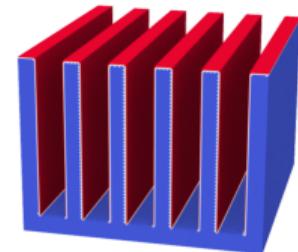
**Boundary Conditions** are set up using the modeling component and are shown with  $h = 0$  in blue and  $g$  in red (right). In particular,  
(i) the bottom surface is set at  $g = 100^\circ \text{ C}$ ,  
(ii) the fin surfaces are set at  $g = 22^\circ \text{ C}$ .

**Results:** The temperature rapidly drops off in this geometry. However, surrounding air can not heat up.

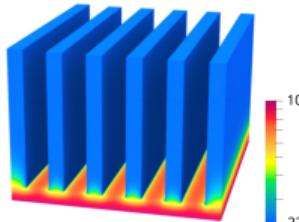
More sophisticated models possible incorporating air-flow & convection effects.



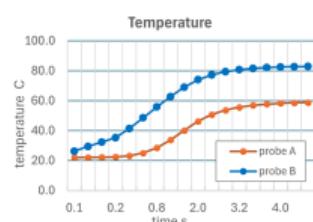
Mesh



Boundary Conditions



Solution



Probes

# Example: Thermal Analysis of Heat Exchanger

**Problem:** Compute the thermal transfer for a given heat-exchanger geometry.

**Model:**

$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$

**Geometry** is obtained using CAD tools for sketching and extrusions using the ANSYS design module.

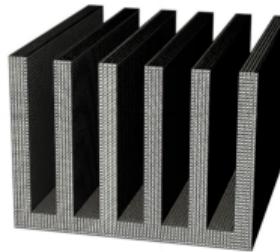
**Meshing** is set up using the modeling component and specified to use Hex8 linear elements.

**Boundary Conditions** are set up using the modeling component and are shown with  $h = 0$  in blue and  $g$  in red (right). In particular,  
(i) the bottom surface is set at  $g = 100^\circ \text{ C}$ ,  
(ii) the fin surfaces are set at  $g = 22^\circ \text{ C}$ .

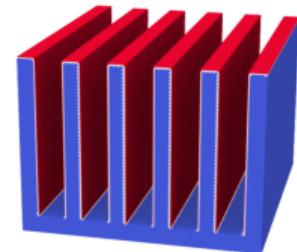
**Results:** The temperature rapidly drops off in this geometry.

However, surrounding air can not heat up.

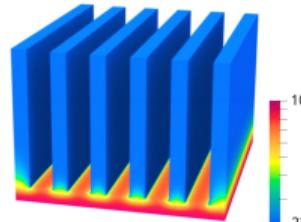
More sophisticated models possible incorporating air-flow & convection effects.



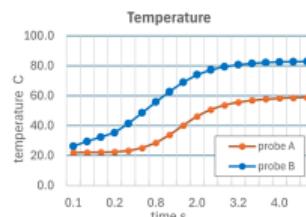
Mesh



Boundary Conditions



Solution



Probes

# Summary

**Software:** There are many packages and interfaces for encapsulating Finite Element Methods and related workflows for use in engineering and scientific investigations.



# Summary

**Software:** There are many packages and interfaces for encapsulating Finite Element Methods and related workflows for use in engineering and scientific investigations.

**Industry** often uses packages like ANSYS, Abaqus, SolidWorks, COMSOL to provide interfaces for rapid prototyping, geometric modeling, obtaining material properties, parametric studies, and managing analysis workflows.



# Summary

**Software:** There are many packages and interfaces for encapsulating Finite Element Methods and related workflows for use in engineering and scientific investigations.



**Industry** often uses packages like ANSYS, Abaqus, SolidWorks, COMSOL to provide interfaces for rapid prototyping, geometric modeling, obtaining material properties, parametric studies, and managing analysis workflows.

**Academia** often uses open source packages like Deal.II, FEniCS, SfePy, and others allowing for more control over the numerical methods, researching techniques, and new applications.

# Summary

**Software:** There are many packages and interfaces for encapsulating Finite Element Methods and related workflows for use in engineering and scientific investigations.



**Industry** often uses packages like ANSYS, Abaqus, SolidWorks, COMSOL to provide interfaces for rapid prototyping, geometric modeling, obtaining material properties, parametric studies, and managing analysis workflows.

**Academia** often uses open source packages like Deal.II, FEniCS, SfePy, and others allowing for more control over the numerical methods, researching techniques, and new applications.

Here, we gave a brief introduction to existing software packages for FAE and how to use ANSYS. In upcoming lectures and exercises we discuss how to develop your own FEMs, extensions, and additional examples.

# Summary

**Software:** There are many packages and interfaces for encapsulating Finite Element Methods and related workflows for use in engineering and scientific investigations.



**Industry** often uses packages like ANSYS, Abaqus, SolidWorks, COMSOL to provide interfaces for rapid prototyping, geometric modeling, obtaining material properties, parametric studies, and managing analysis workflows.

**Academia** often uses open source packages like Deal.II, FEniCS, SfePy, and others allowing for more control over the numerical methods, researching techniques, and new applications.

Here, we gave a brief introduction to existing software packages for FAE and how to use ANSYS. In upcoming lectures and exercises we discuss how to develop your own FEMs, extensions, and additional examples.

**Additional Information:** Many video tutorials and technical details also are available on-line and from the software package websites.

# Summary

**Software:** There are many packages and interfaces for encapsulating Finite Element Methods and related workflows for use in engineering and scientific investigations.



**Industry** often uses packages like ANSYS, Abaqus, SolidWorks, COMSOL to provide interfaces for rapid prototyping, geometric modeling, obtaining material properties, parametric studies, and managing analysis workflows.

**Academia** often uses open source packages like Deal.II, FEniCS, SfePy, and others allowing for more control over the numerical methods, researching techniques, and new applications.

Here, we gave a brief introduction to existing software packages for FAE and how to use ANSYS. In upcoming lectures and exercises we discuss how to develop your own FEMs, extensions, and additional examples.

**Additional Information:** Many video tutorials and technical details also are available on-line and from the software package websites.