

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.

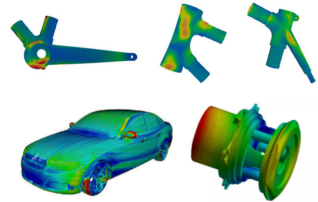
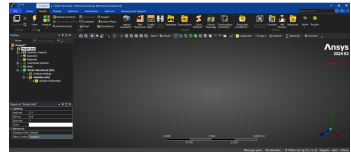
Current Practice: FEA is a common part of the engineering design process allowing for rapid prototyping and insights to help reduce costs. While physical builds are still needed for validation, may reduce number of trials.

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

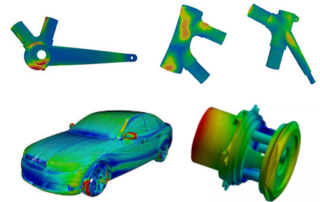
Provides software for design, modeling, and analysis to aid in engineering systems.



ANSYS: Software

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

Comprised of several components and modules

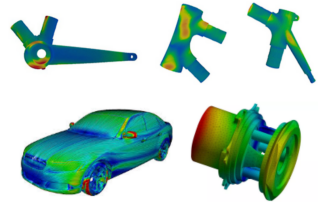
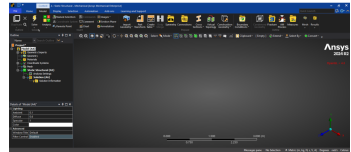


ANSYS: Software

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.

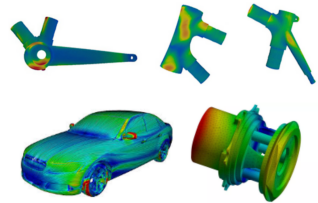
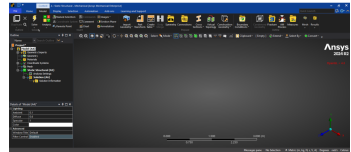


ANSYS: Software

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.

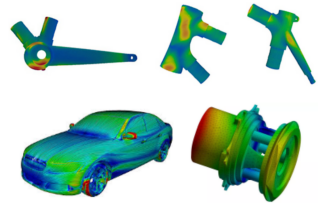
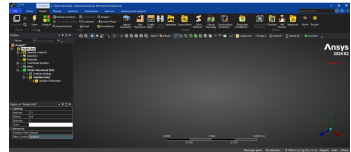


ANSYS: Software

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).

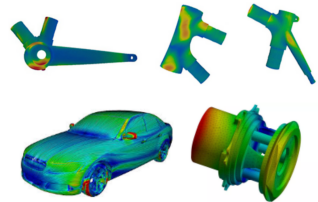
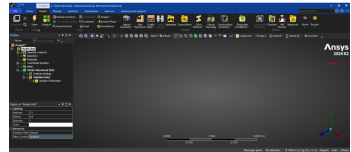


ANSYS: Software

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.

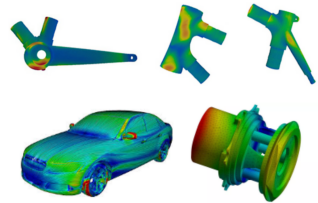
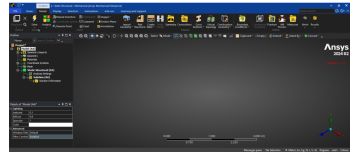


ANSYS: Software

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.



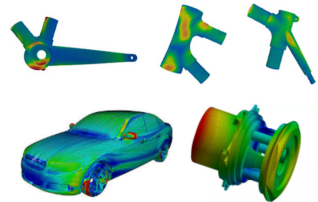
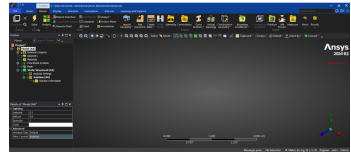
ANSYS: Software

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

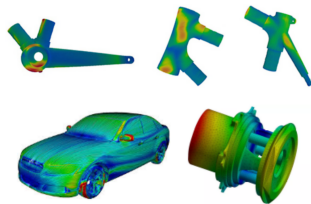
Provides software for design, modeling, and analysis to aid in engineering systems.

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

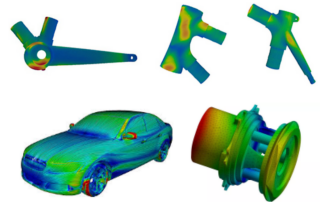
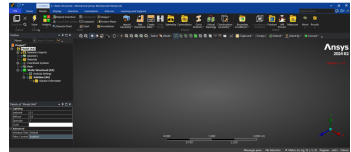
Provides software for design, modeling, and analysis to aid in engineering systems.

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.

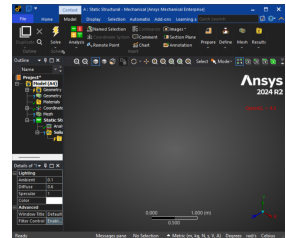
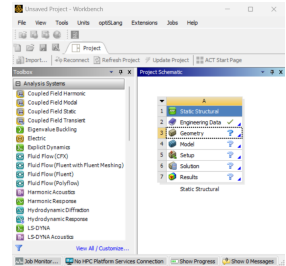
ANSYS Mechanical finite element methods will be our primary focus. Other components are used similarly.

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



ANSYS Mechanical: Setting Up FEA Studies

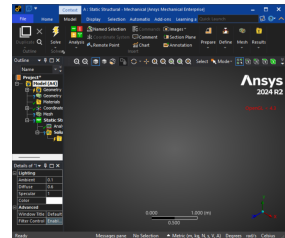
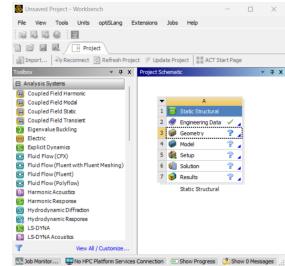
Outline: Creating a new FEA study follows these steps:



ANSYS Mechanical: Setting Up FEA Studies

Outline: Creating a new FEA study follows these steps:

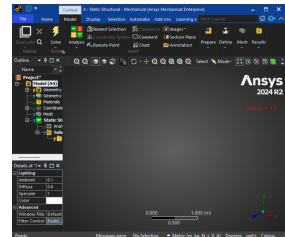
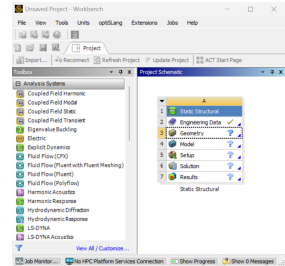
1. **Open Workbench:** Select "Static Structural" and drag to the workflow space.



ANSYS Mechanical: Setting Up FEA Studies

Outline: Creating a new FEA study follows these steps:

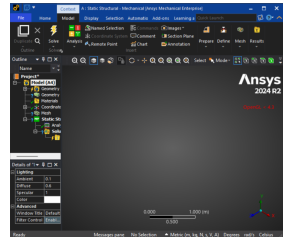
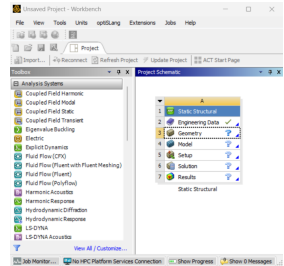
1. **Open Workbench:** Select "Static Structural" and drag to the workflow space.
2. **Engineering Data:** Specify material properties and other settings.



ANSYS Mechanical: Setting Up FEA Studies

Outline: Creating a new FEA study follows these steps:

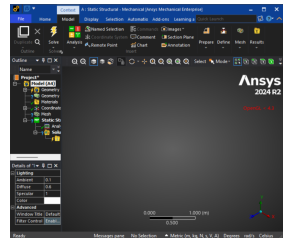
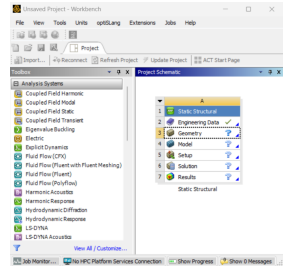
1. **Open Workbench:** Select "Static Structural" and drag to the workflow space.
2. **Engineering Data:** Specify material properties and other settings.
3. **Geometry:** Build the geometry and designate properties.



ANSYS Mechanical: Setting Up FEA Studies

Outline: Creating a new FEA study follows these steps:

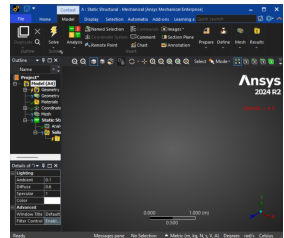
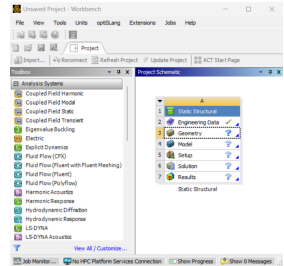
1. **Open Workbench:** Select "Static Structural" and drag to the workflow space.
2. **Engineering Data:** Specify material properties and other settings.
3. **Geometry:** Build the geometry and designate properties.
4. **Model:** Define boundary conditions and specify analysis to be performed.



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.

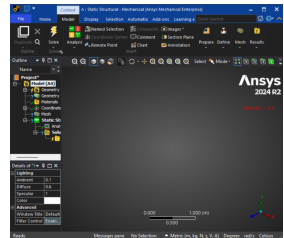
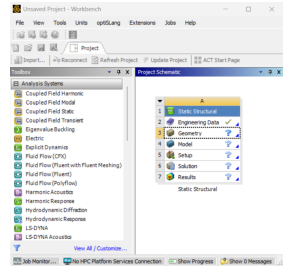


ANSYS Mechanical: Setting Up FEA Studies

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.



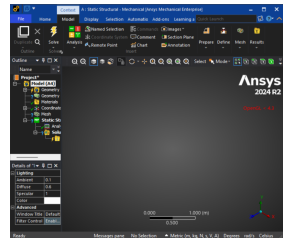
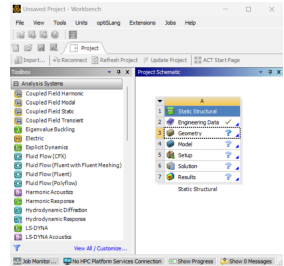
ANSYS Mechanical: Setting Up FEA Studies

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.



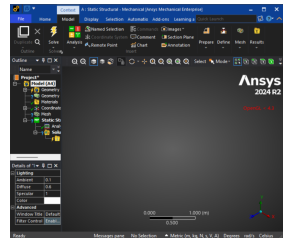
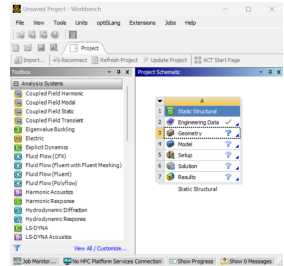
ANSYS Mechanical: Setting Up FEA Studies

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.

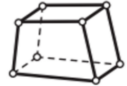


ANSYS: Finite Element Types Utilized

FEM: The most common element types are



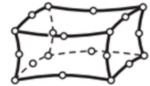
Tet4



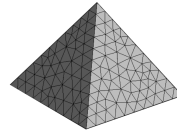
Hex8



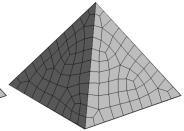
Tet10



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



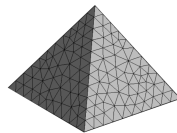
Hex8



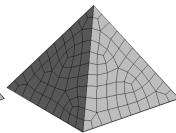
Tet10



Hex20



Tet4



Hex8

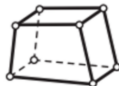
ANSYS: Finite Element Types Utilized

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



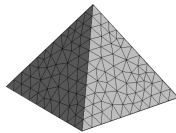
Hex8



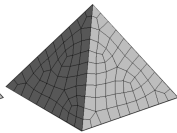
Tet10



Hex20



Tet4



Hex8

ANSYS: Finite Element Types Utilized

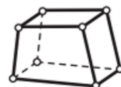
FEM: The most common element types are

- **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



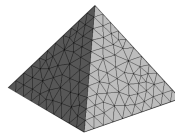
Hex8



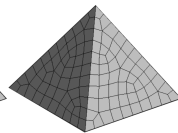
Tet10



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 evaluation at the locations shown on the right,

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



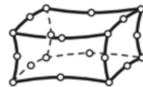
Tet4



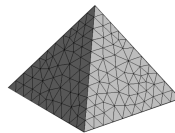
Hex8



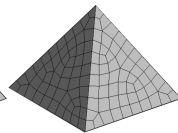
Tet10



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 evaluation 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



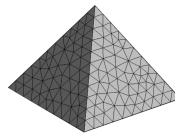
Hex8



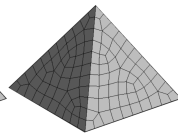
Tet10



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 evaluation 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



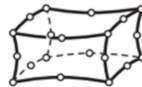
Tet4



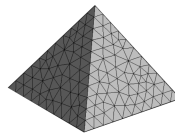
Hex8



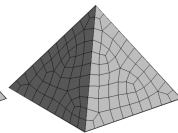
Tet10



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 evaluation at the locations shown on the right,

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

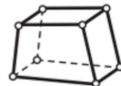
Isoparametric elements with same shape space for geometry.

Element choice involves trade-offs

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



Tet4



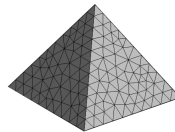
Hex8



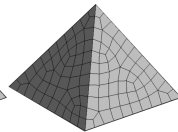
Tet10



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 evaluation at the locations shown on the right,

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

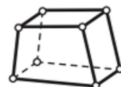
Isoparametric elements with same shape space for geometry.

Element choice involves trade-offs

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



Tet4



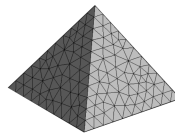
Hex8



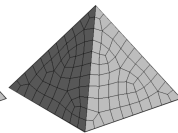
Tet10



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 evaluation 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.

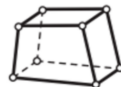
Element choice involves trade-offs

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

Meshing quality is also important to ensure reliable results.



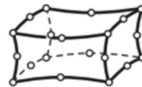
Tet4



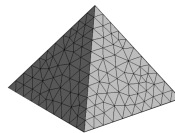
Hex8



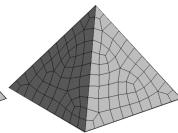
Tet10



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 evaluation 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.

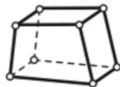
Element choice involves trade-offs

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

Meshing quality is also important to ensure reliable results.



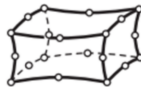
Tet4



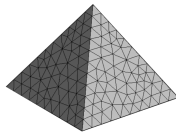
Hex8



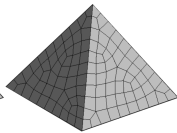
Tet10



Hex20



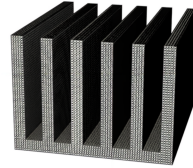
Tet4



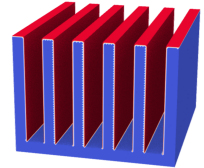
Hex8

Example: Thermal Analysis of Heat Exchanger

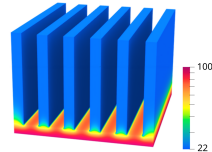
Problem: Compute the thermal transfer for a given heat-exchanger geometry.



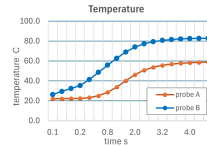
Mesh



Boundary Conditions



Solution



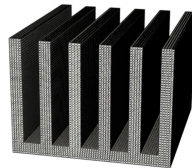
Probes

Example: Thermal Analysis of Heat Exchanger

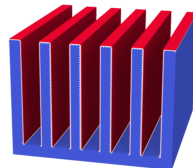
Problem: Compute the thermal transfer for a given heat-exchanger geometry.

Model:

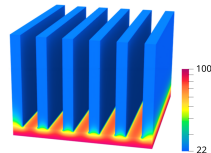
$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$



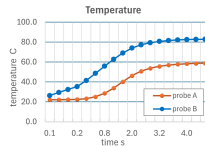
Mesh



Boundary Conditions



Solution



Probes

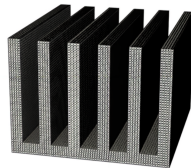
Example: Thermal Analysis of Heat Exchanger

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

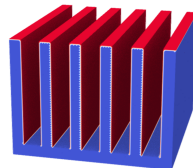
Model:

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

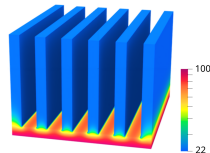
Geometry is obtained using CAD tools for sketching and extrusions using the ANSYS design module.



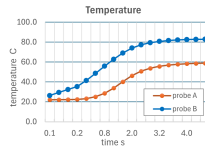
Mesh



Boundary Conditions



Solution



Probes

Example: Thermal Analysis of Heat Exchanger

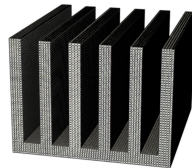
Problem: Compute the thermal transfer for a given heat-exchanger geometry.

Model:

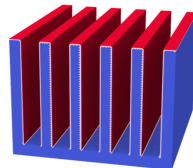
$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$

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

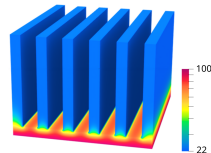
Meshing is set up using the modeling component and specified to use Hex8 linear elements.



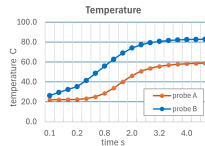
Mesh



Boundary Conditions



Solution



Probes

Example: Thermal Analysis of Heat Exchanger

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

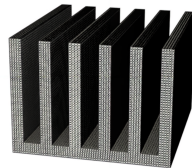
Model:

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

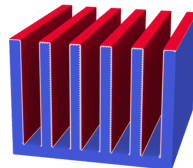
Geometry is obtained using CAD tools for sketching and extrusions using the ANSYS design module.

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

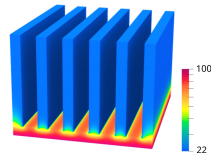
Boundary Conditions are set up using the modeling component and are shown with $h = 0$ in blue and g in red (right).



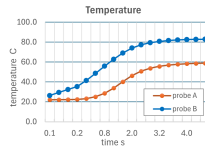
Mesh



Boundary Conditions



Solution



Probes

Example: Thermal Analysis of Heat Exchanger

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

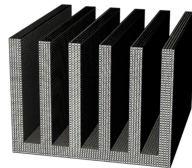
Model:

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

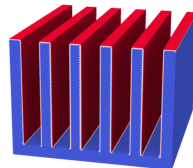
Geometry is obtained using CAD tools for sketching and extrusions using the ANSYS design module.

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

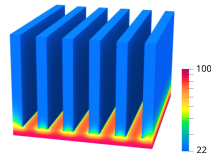
Boundary Conditions are set up using the modeling component and are shown with $h = 0$ in blue and g in red (right). In particular,



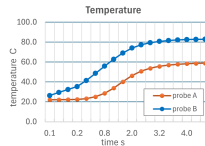
Mesh



Boundary Conditions



Solution



Probes

Example: Thermal Analysis of Heat Exchanger

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

Model:

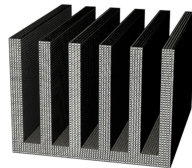
$$\begin{cases} -\Delta u &= -f, & x \in \Omega \\ u &= g, & x \in \partial\Omega_D \\ \nabla u \cdot \mathbf{n} &= h, & x \in \partial\Omega_N. \end{cases}$$

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

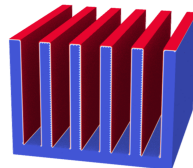
Meshing is set up using the modeling component and specified to use Hex8 linear elements.

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

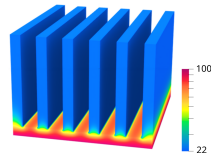
- (i) the bottom surface is set at $g = 100^\circ \text{C}$,
- (ii) the fin surfaces are set at $g = 22^\circ \text{C}$.



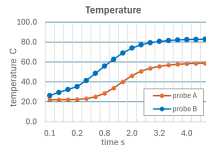
Mesh



Boundary Conditions



Solution



Probes

Example: Thermal Analysis of Heat Exchanger

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

Model:

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

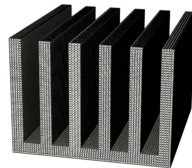
Geometry is obtained using CAD tools for sketching and extrusions using the ANSYS design module.

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

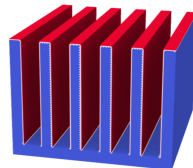
Boundary Conditions are set up using the modeling component and are shown with $h = 0$ in blue and g in red (right). In particular,

- (i) the bottom surface is set at $g = 100^\circ \text{C}$,
- (ii) the fin surfaces are set at $g = 22^\circ \text{C}$.

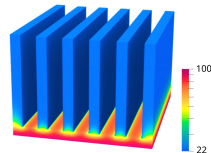
Results: The temperature rapidly drops off in this geometry. However, surrounding air can not heat up.



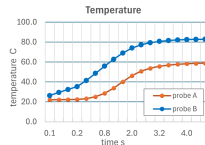
Mesh



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.

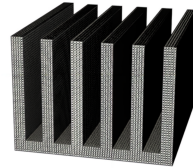
Meshing is set up using the modeling component and specified to use Hex8 linear elements.

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

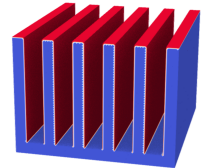
- (i) the bottom surface is set at $g = 100^\circ \text{C}$,
- (ii) the fin surfaces are set at $g = 22^\circ \text{C}$.

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

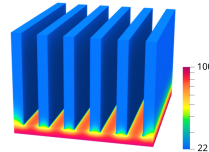
More sophisticated models possible incorporating air-flow & convection effects.



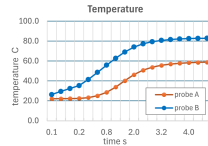
Mesh



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.

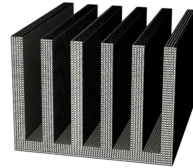
Meshing is set up using the modeling component and specified to use Hex8 linear elements.

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

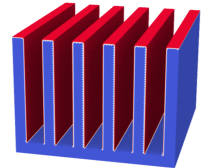
- (i) the bottom surface is set at $g = 100^\circ \text{C}$,
- (ii) the fin surfaces are set at $g = 22^\circ \text{C}$.

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

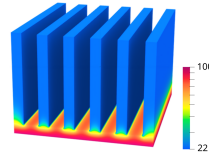
More sophisticated models possible incorporating air-flow & convection effects.



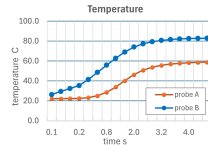
Mesh



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.