Module 03 Step-by-Step Guide: Modeling Approach

Release 2023 R1

Please note:
- These training materials were developed and tested in Ansys Release 2023 R1. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.

Ansys

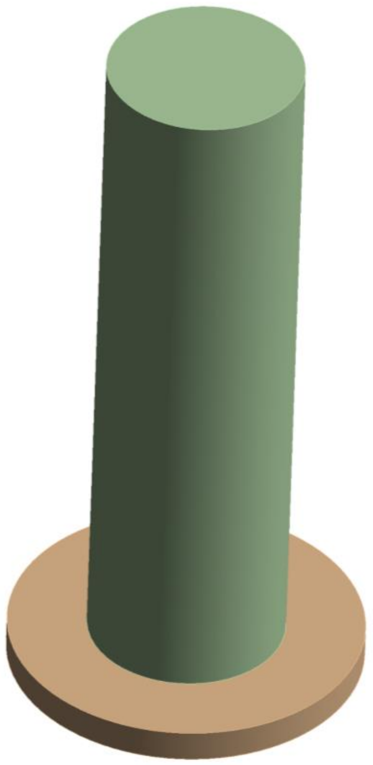
Step-by-Step Guide 03: Modeling Approach

Use this guide as a summary of the modeling approach discussed in this module.

- The Preliminary Decisions that will lead us to a reasonable analysis approach are the following:
  - What components to include in the model and why?
  - How will I characterize those components from a geometry and material standpoint?
  - How will I model the interfaces among the components?
  - How will I characterize the operating environment?
  - How will I validate both my model and my design?

Step-by-Step Guide 03: Modeling Approach

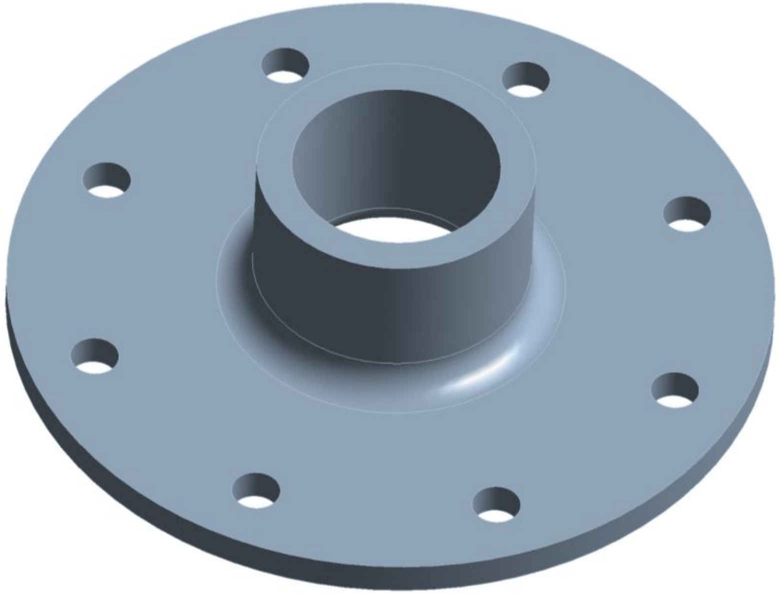
- Discussion point #1: What components do I model and why?
  - Valve body, required per the stated analysis objective
  - **Valve rod**
  - **Valve seal**
  - Flange
  - Nuts/Bolts



- *These components will be modeled as solid bodies*
  - *We're not concerned with their mass (no inertial loading)*
  - *Their stiffness is important as they distribute the internal pressure to the Flange part and are important to the load path*
  - *Later in the course we'll refine the interface condition between these parts along with the interface between the Valve Rod and Flange*

Step-by-Step Guide 03: Modeling Approach

- Discussion point #1: What components do I model and why?
  - Valve body
  - Valve rod
  - Valve seal
  - **Flange**
  - Nuts/Bolts



- *This component will be modeled as a solid body*
  - *We're not concerned with its mass (no inertial loading)*
  - *Flange and Valve Seal interface is critical for proper load path for internal pressure*
  - *Gap between Flange and Valve body could be important; there's likely a gasket between these parts which we have no information about; that could influence the load path and also the stress results local to that interface*

Step-by-Step Guide 03: Modeling Approach

- Discussion point #1: What components do I model and why?
  - Valve body
  - Valve rod
  - Valve seal
  - Flange
  - **Nuts/Bolts**

- *These components will be modeled as solid bodies*
  - *We're not concerned with their mass (no inertial loading)*
  - *They contribute local stiffness between Valve body and Flange*
  - *Later in the course we'll consider the effect of their preload on the assembly as well as abstract them by replacing them with beams*

Step-by-Step Guide 03: Modeling Approach

- Discussion point #2: What material properties do we need?
  - Valve body (Type 40 Gray Cast Iron)
  - Valve rod (AISI 6150 Steel)
  - Valve seal (Type 302 Stainless Steel)
  - Flange (Type 40 Gray Cast Iron)
  - Nuts/Bolts (AISI 6150 Steel)

- *Required properties are Young's Modulus, Poisson's ratio, Strength*
  - *Strength is used for Safety Factor calculations*
  - *Density is only needed for inertial loading*
  - *CTE and Conductivity are only needed for thermal stress and thermal analysis*
  - *Fatigue strengths are only needed for fatigue analysis*
  - *Stress/strain data is only needed for plasticity or hyperelasticity*

Step-by-Step Guide 03: Modeling Approach

- Discussion point #3: How do I treat the interfaces among the components, along with interfaces to the rest of the world?
  - In the real world, what happens at the interface between components?
  - What are some ways that can be handled in FEA?
  - If we choose not to model a component, how and/or when do we account for its presence?

    - *Supports are needed at interfaces to rest of "world" to prevent rigid body motion (Sum of forces = 0)*
    - *Bonded contact or shared topology (mesh) can be used to enforce load transfer through part interfaces*
    - *Later in the course we'll investigate frictionless contact at some interfaces*
    - *Stress at supports may be approximate and/or inaccurate, much like the supports themselves*

Step-by-Step Guide 03: Modeling Approach

- Discussion point #4: What do I know about the operating environment and what simplifying assumptions will/must I make?
  - How will the pressure be modeled ?
  - What effect will the preload in the bolts have?
  - Based on what we know about the mating flanges at the inlet/outlet, how will we model that condition?

- *Pressure is likely transient and non-uniform, but we'll assume a worst-case static uniform pressure as it conforms to the scope of this approach*
- *Later in the course we'll implement preloads in the bolts using common engineering handbook guidelines*
- *Stress at supports may be approximate and/or inaccurate, much like the supports themselves*

Step-by-Step Guide 03: Modeling Approach

- Discussion point #5: How will I know if/when I’ve met the design objectives?
  - What stress are we going to use to validate the design criteria?
  - How will I know whether the stress value is accurate?
  - How else can we validate the model in order to determine whether we should believe the answers?

- *Under fault pressure of 7 MPa, Valve body stress must be < 0.5*Ultimate Strength*
- *Equivalent, Principal and Directional (Normal) stresses to validate the design*
- *Failure criteria for brittle material*
- *Ensure static equilibrium via reaction force balance*
- *Influence of mesh on the stress results*

Ansys

End of presentation