

Module 02 Student Reference Guide: Problem Statement

Release 2023 R1

Please note:

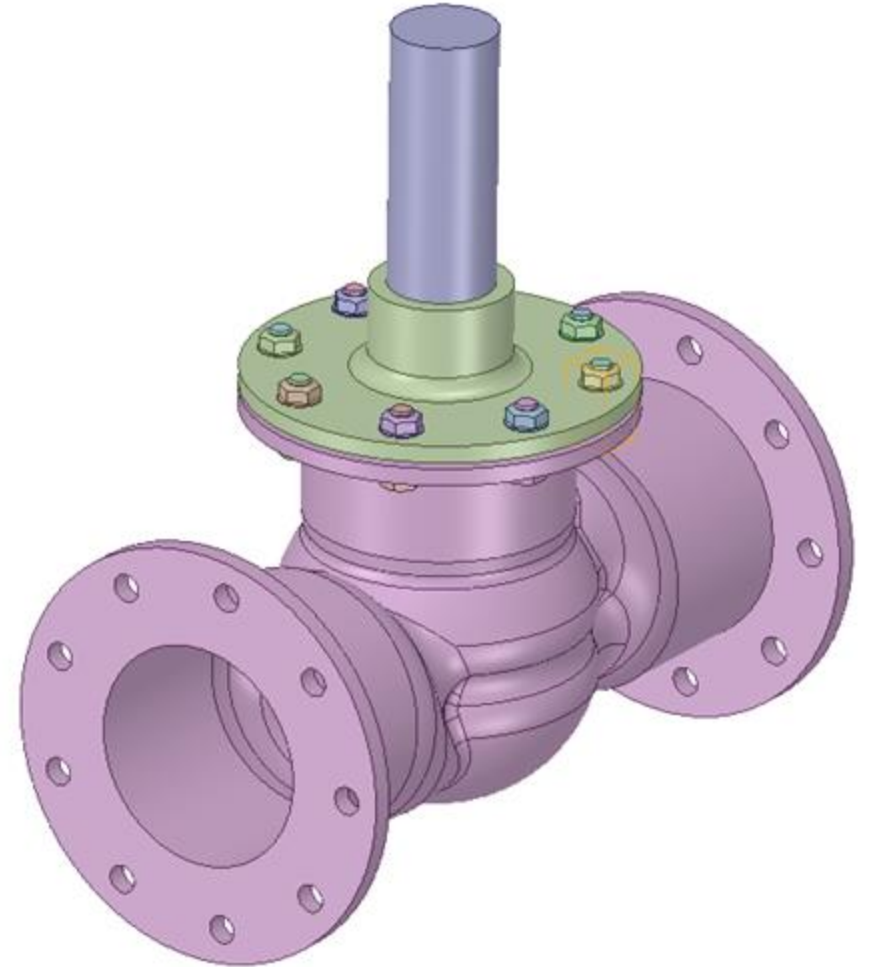
- These training materials were developed and tested in Ansys Release 2023 R1. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software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

/ Module 02: Learning Environment

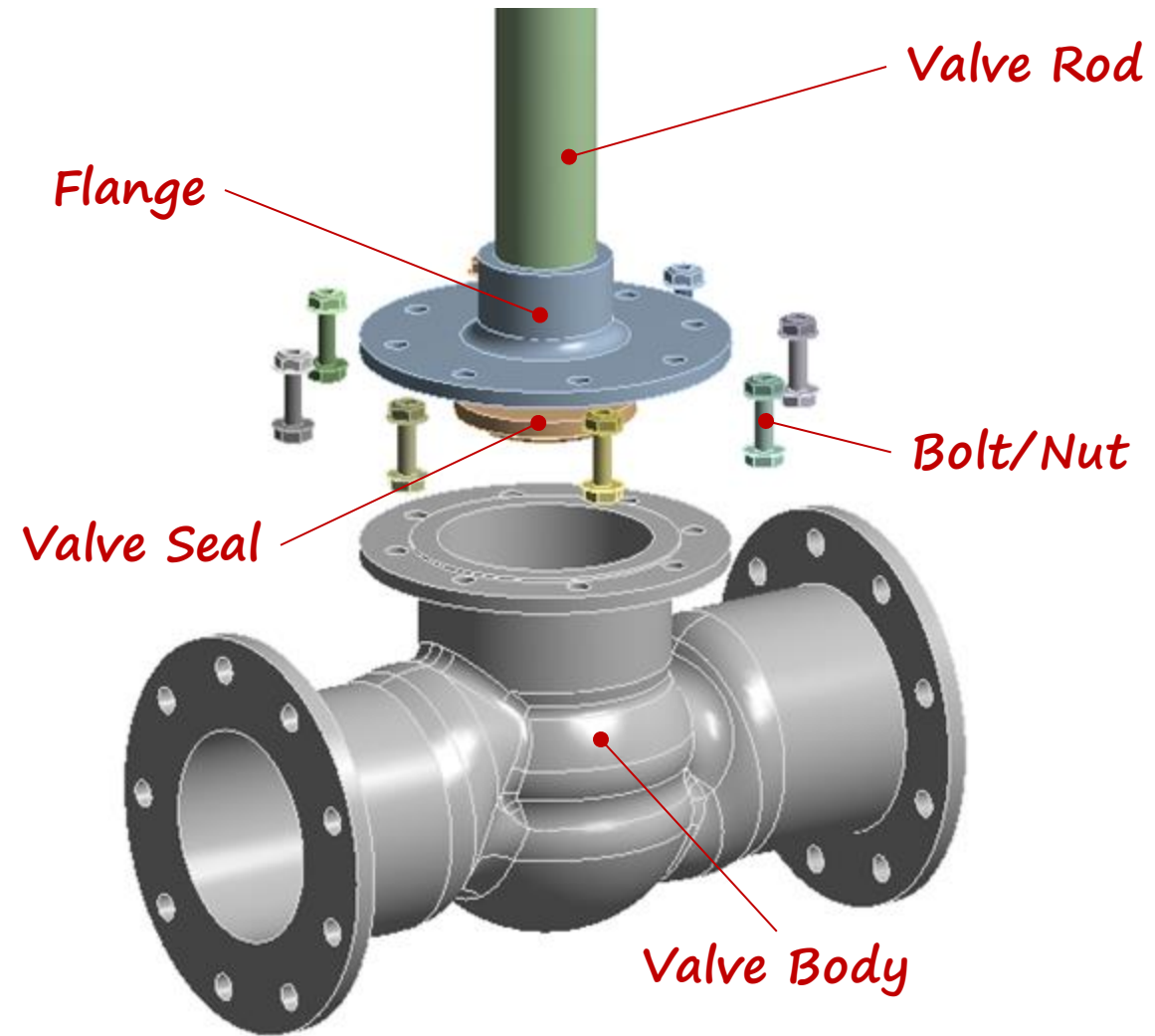
- Student materials include:
 - Student Step-by-Step Guides (PR): allow the student to optionally repeat the same steps shown in the instructor demonstration
 - Student Reference Guides (RF): provide the student with additional background, reference information, or details on any of several topics mentioned during the live demonstrations
 - Workshops (WS): provide the student with the opportunity to apply the concepts and procedures presented in the instructor demonstration to a specified analysis problem

/ Module 02: Reference Material

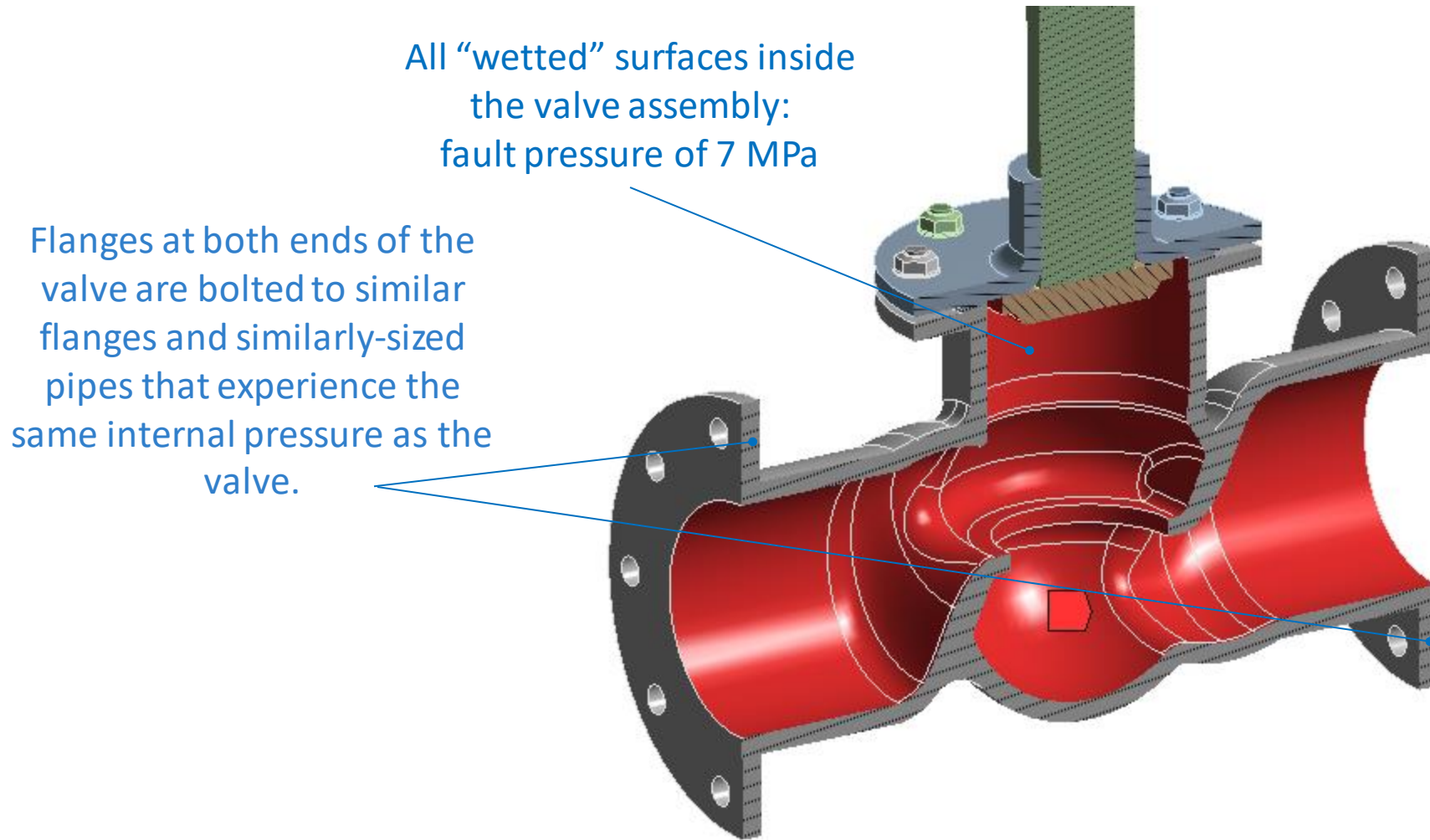
You're analyzing a large globe valve that is part of an industrial cooling system. It is already known that this valve satisfies design conditions, but it is also required that the valve be able to withstand "fault" loading conditions of five times the design pressure without exceeding one-half of its material ultimate strength. The flange, valve seal, valve rod, bolt, and nut components are all commercial items and have been evaluated separately, so you will focus on the valve body component in the context of the overall assembly model. Your assignment is to determine whether the current design of the valve body satisfies the specified fault limits for the governing boundary conditions.



Module 02: Reference Material



Module 02: Reference Material



Goal:

- *Introduce the engineering analysis problem that will form the basis of the lectures throughout the rest of the course*



Fault Loading Condition:

- *Pressure on wetted surfaces*

$$P_w = 7.00 \text{ MPa}$$

Design Requirement:

- *Maximum equivalent stress*

$$\sigma_{\max} = \frac{1}{2}\sigma_{\text{ult}}$$



Assumption:

- *Both end flanges are bolted to pipes having the same diameter, stiffness, and flange geometry.*



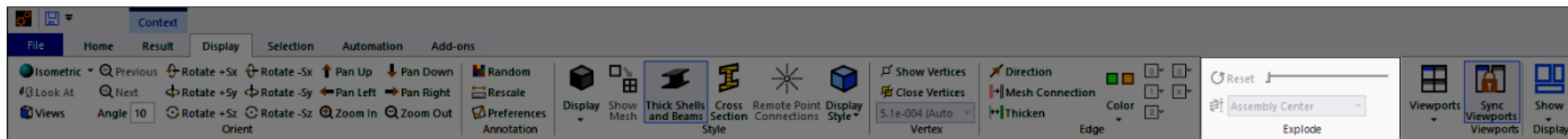
Module 02: Reference Material

- Explode

https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Ribbon_Display_tab.html

Explode

Highlighted below, the **Explode** group is a graphical display feature used to create imaginary distance between geometry bodies (only) of your model for viewing purposes.



Once the mesh is generated, this feature is not supported when you have the **Mesh** object selected or when the **Show Mesh** feature is turned on. In addition, when viewing the mesh, exploded geometry bodies, although not visible in the Geometry window, are still in an exploded state and passing the cursor over an exploded body will highlight the (otherwise invisible) body and it is also selectable at this time.


Reset Button

This button reassembles the parts of your model to their original position.

Explode View Factor Slider

This slider tool enables you to change the exploded distance between the parts from their original position.

Move Springs/Beams with Parts

The button  for this option enables you to see an accurate representation of connections on your model, such as Springs and Beams, by showing the connections stretched from the assigned locations on the moving parts. Because the display is graphically accurate, the processing requirements are intensive. Use the default position (not active/depressed) when moving the slider for large models and when connection representations are not critical.

Assembly Center Drop-Down List

This drop-down list provides the available coordinate system options as well as the **Assembly Center** option (default setting) that defines the position in space from which the exploded view originates and the **Assembly Center (Visible)** option that accounts for the visible parts only. The **Global Coordinate System** is always an available option as well as any [user-defined coordinate systems](#).

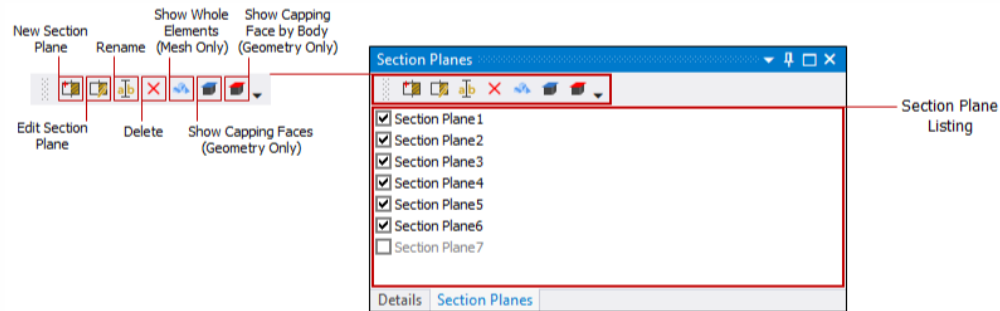
Module 02: Reference Material

- Creating Section Planes

https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_New_Sec_Plane.html

Application

Select the **Section Plane** option from the **Insert** group of the **Home** tab to open the **Section Planes** window illustrated below. The window displays a list of existing section planes (once created) and also provides the tools used to add, modify, or delete you section planes.



Note: The Section Planes tool supports up to six (6) defined planes. Once you exceed six, unchecking an existing plane enables you to activate any defined planes greater than six.

Section Plane Tools

The **Section Planes** window provides the following tools. You toggle these tools on and off by selecting the button.

New Section Plane



End of presentation