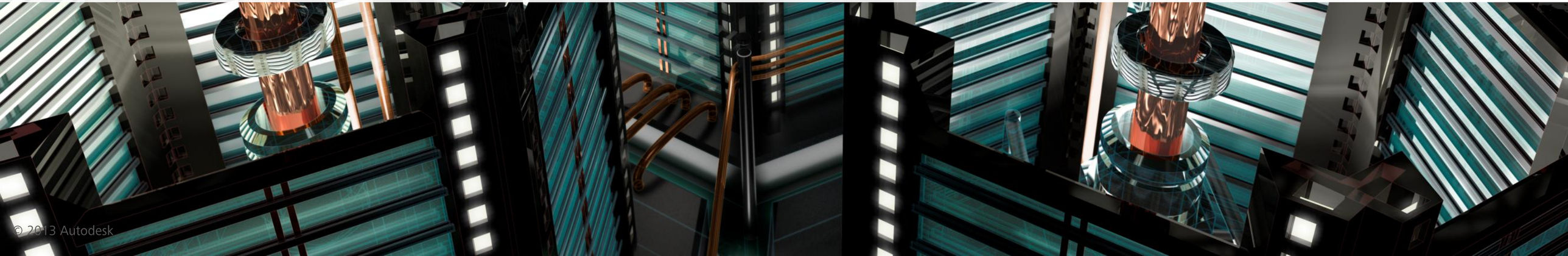
Stress Analysis with Autodesk Inventor 2014

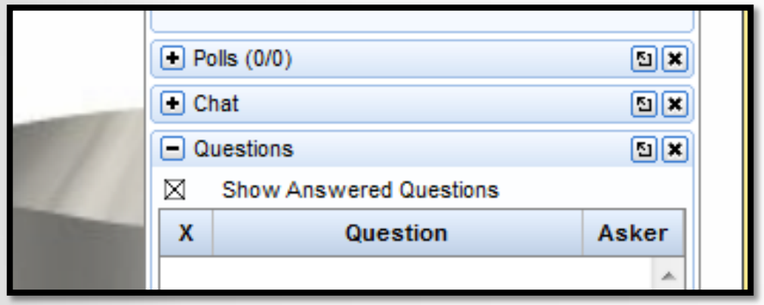
Andrew Sears – Fusion360 and Inventor QA
Dan Banach – Program Manager, Autodesk Education

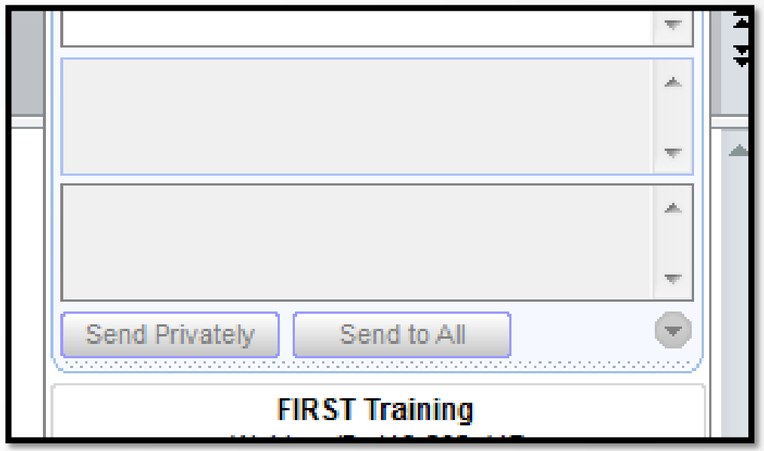
AUTODESK.



Webinar Q&A

- Type questions in GoTo Webinar





- We are recording and will post when possible

AUTODESK.

Student Resources

- http://students.autodesk.com
- http://autodesk.com/first
- http://www.youtube.com/user/AutodeskEd

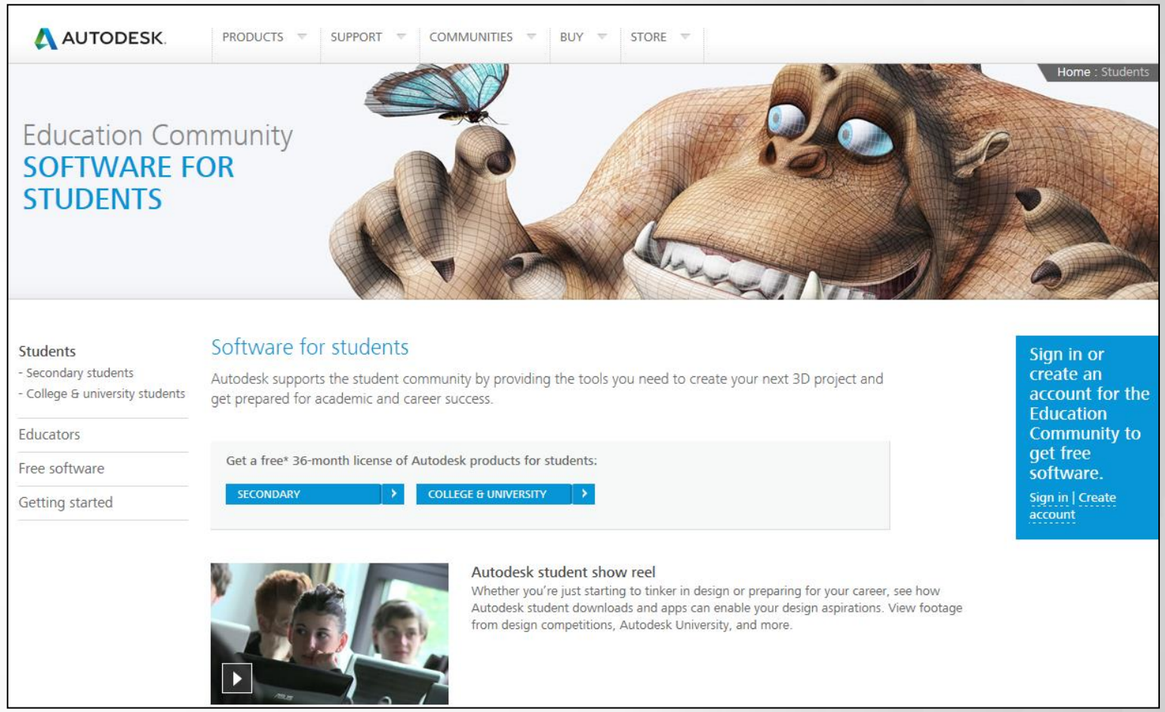
- http://wikihelp.autodesk.com/enu
- http://engineeringexploration.autodesk.com

- FIRST@autodesk.com
- http://bxd.autodesk.com/

AUTODESK.

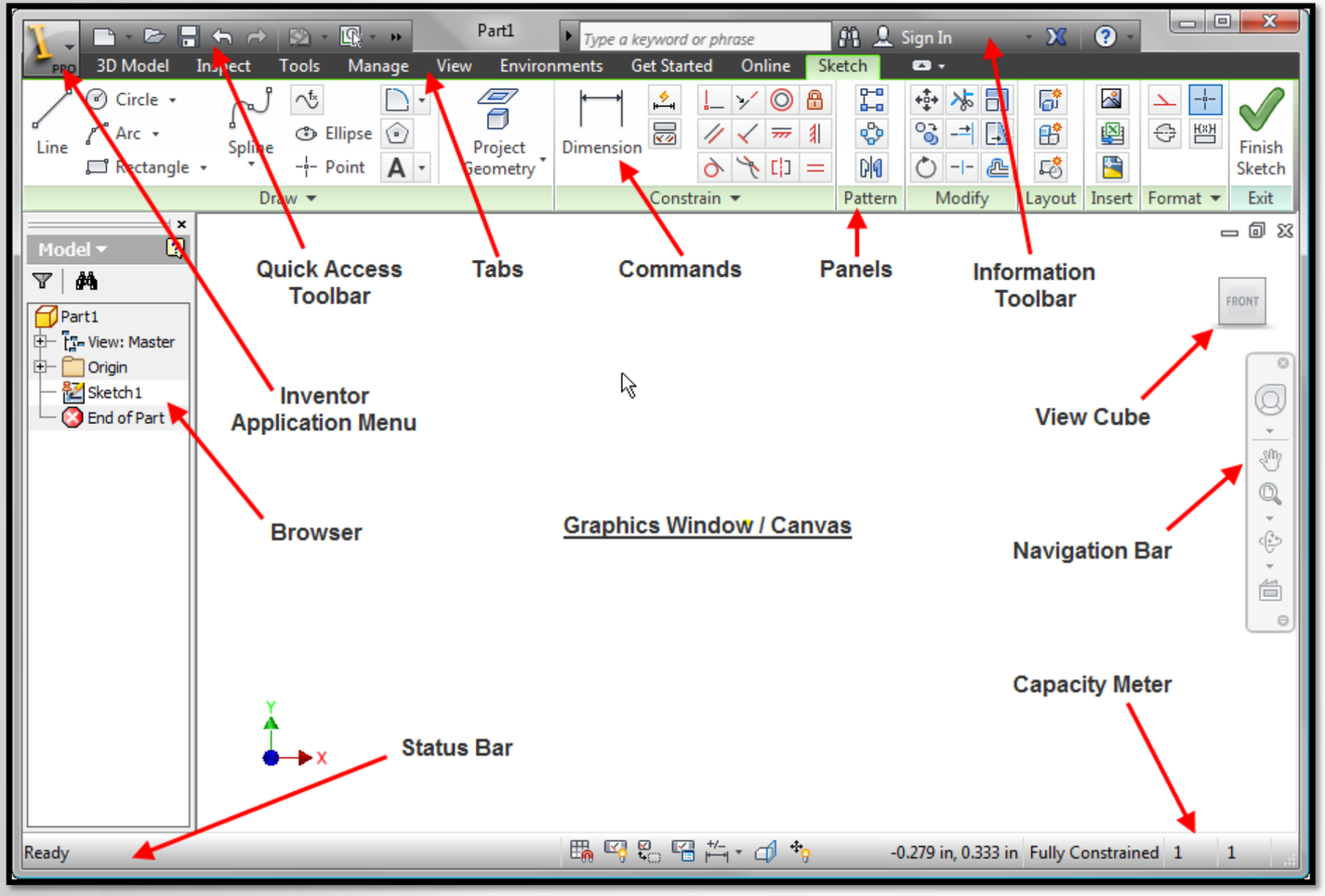
Free software for students, teachers and mentors – personal computer

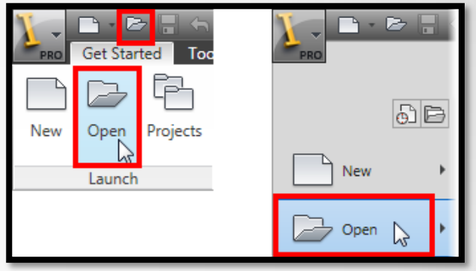
- Download from Autodesk Education Community
  - http://students.autodesk.com
- 40+ titles available for FREE
  - Autodesk® Inventor®
  - AutoCAD®
  - Autodesk® 3ds Max®
  - Autodesk® Maya®
  - And more…
- 3 year license
- 2014 software – no watermark
- 7.5 + million members
- Gallery showcase
- Learning materials

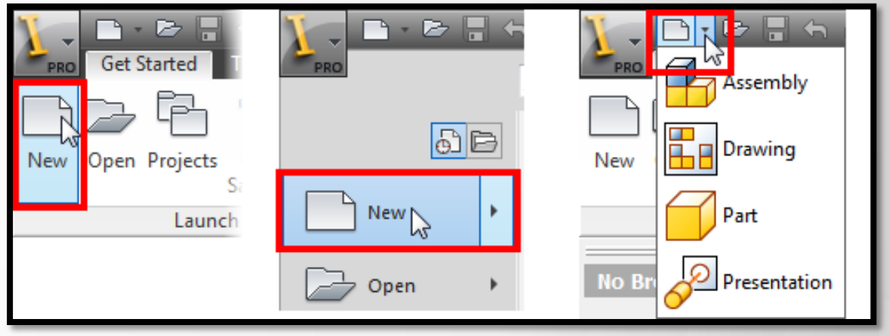


AUTODESK.

User Interface – Refresher



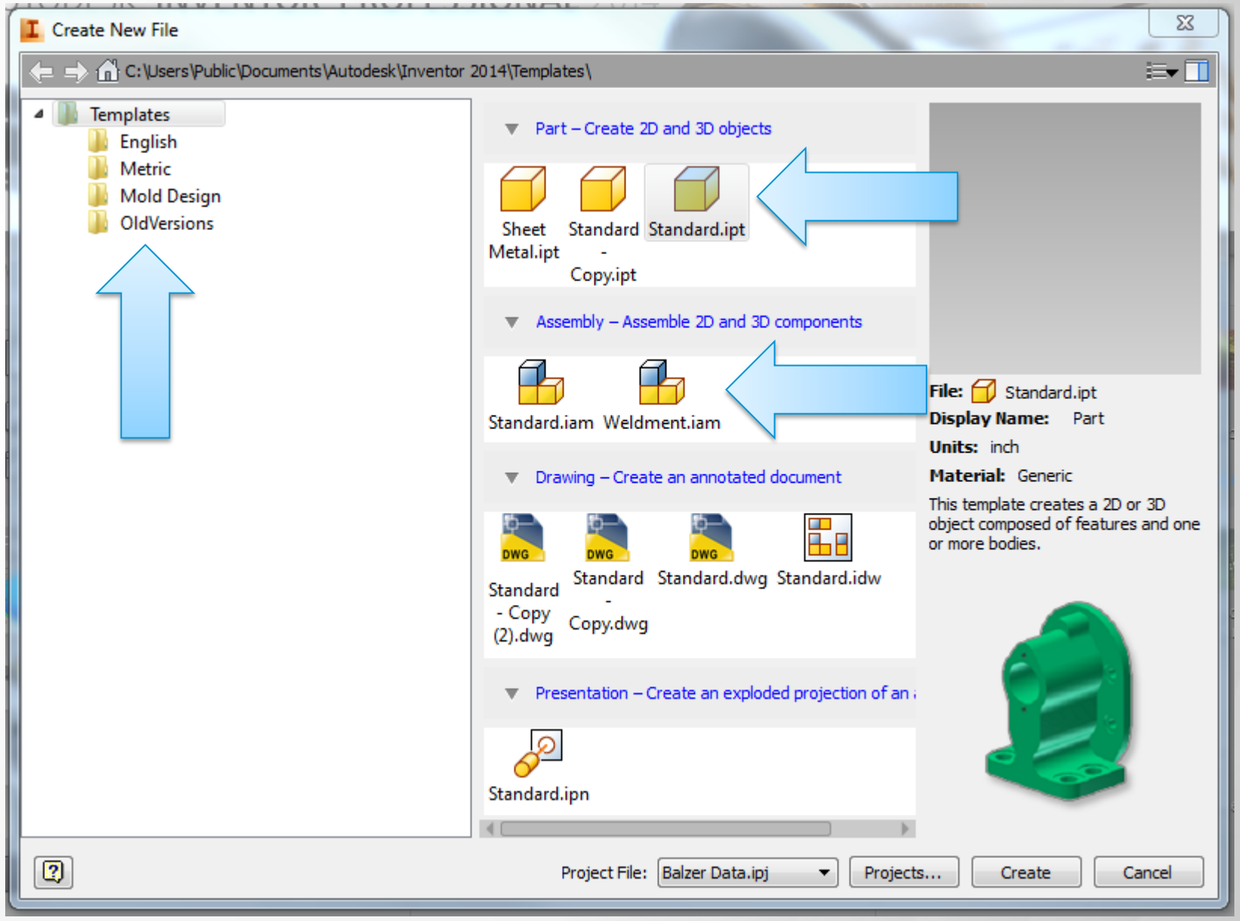




AUTODESK.

Open and Create New Files – Refresher

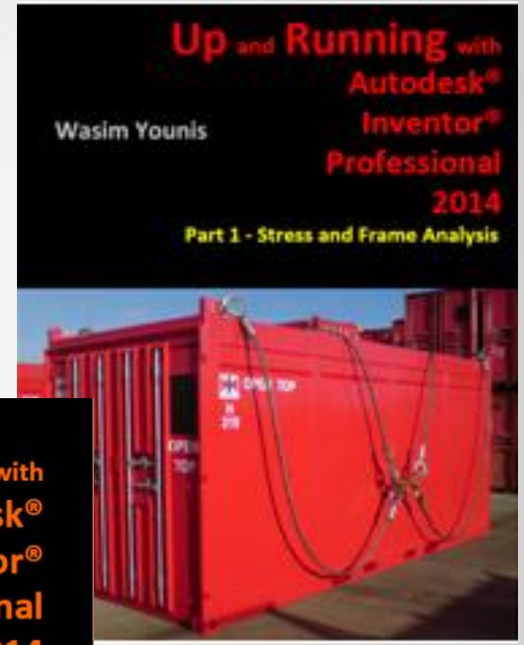
- Open
  - Open an existing Autodesk Inventor file
- New
  - Template for a new part, sheet metal part, assembly, presentation, or drawing



AUTODESK.

Up and Running with Autodesk Inventor Professional 2014 - Simulation

Up and Running with Autodesk® Inventor® Professional 2014 is dedicated to the requirements of Inventor users who need to quickly learn or refresh their skills and apply the capabilities of Inventor Professional 2014. Providing clear guidance and All important real world tutorials, the step-by-step, heavily illustrated approach of this book will help designers, engineers, and manufactures of all skill levels become Simulation experts



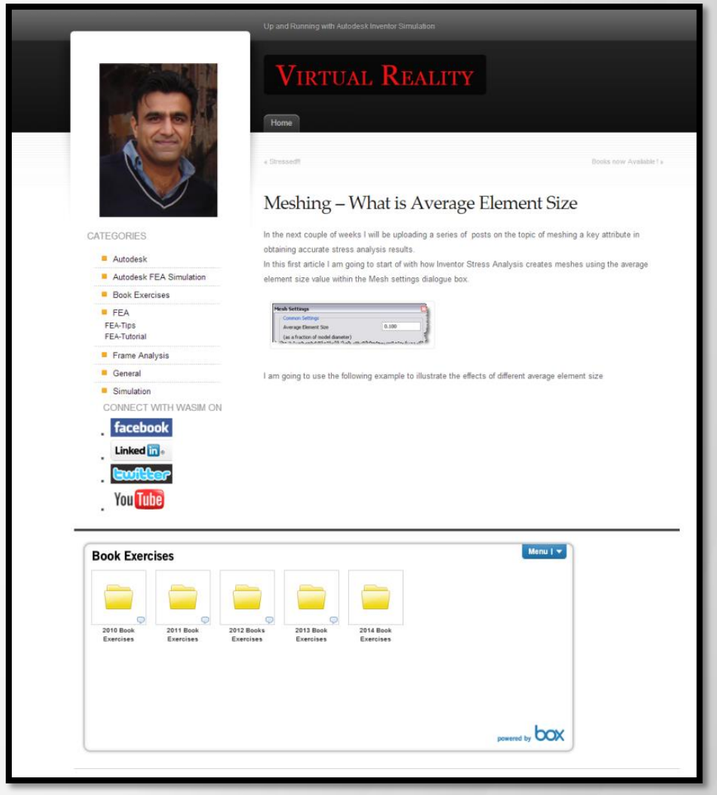
AUTODESK.

Dynamic Simulation Book – Useful Information

- Books are available from amazon.com

Up and Running with Inventor Simulation

- Access book exercise files at vrblog.info



AUTODESK.

Stress Analysis (Finite Element Analysis – FEA)

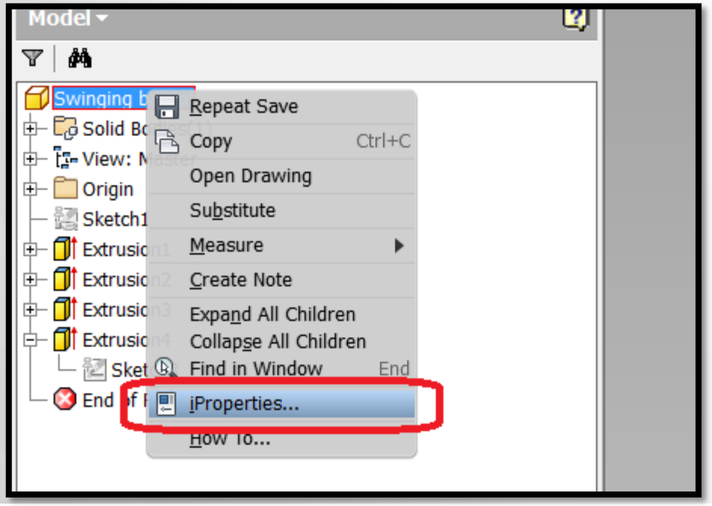
- What I am showing you is a small amount of information around Inventor Stress Analysis (Finite Element Analysis – FEA).
- The software solutions are the results from model setup and material definition.
- FEA is a mathematical representation of the real part.
- Appropriate safety factors should be defined prior to manufacturing.
- All model setup and results need to be reviewed by a mentor familiar with Inventor and FEA techniques.
- Always use your best judgment.
- Think of it this way, FEA is spell checker for designers

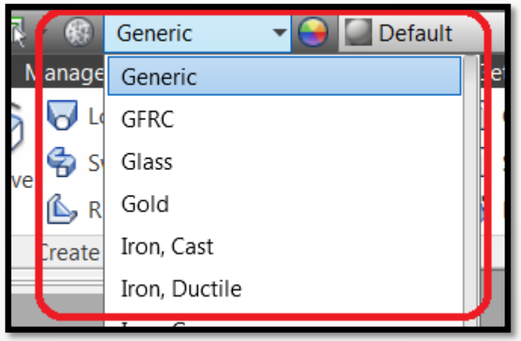
Stress Analysis

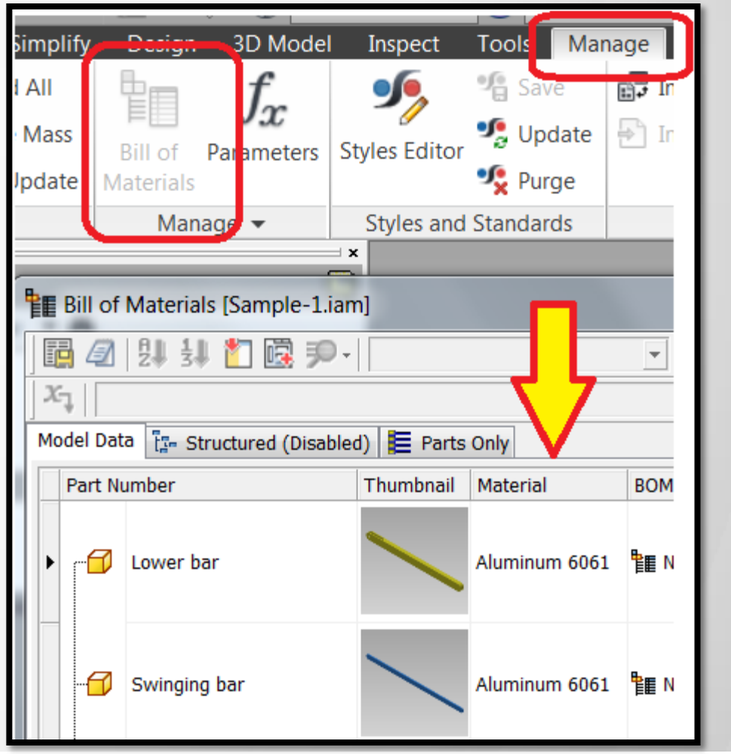
- After this webinar, you will be able to:
  - Setup a Static Stress analysis
    - Assign material to parts
    - Define constraints (boundary conditions) to the model
    - Define loads (force & torque) to the model
    - Define contacts
  - Make decisions about
    - Material selection (plastic vs. aluminum vs. steel)
  - Export forces from Dynamic Simulation
  - Review the results with a mentor

Setup - Assigning Physical Properties to Parts

- Assign a physical property (material) to parts
  - Aluminum, ABS plastic, clear polycarbonate, etc.
    - iProperties (RMB on upper node)
    - Material Browser
    - Bill of Materails (BOM) editor



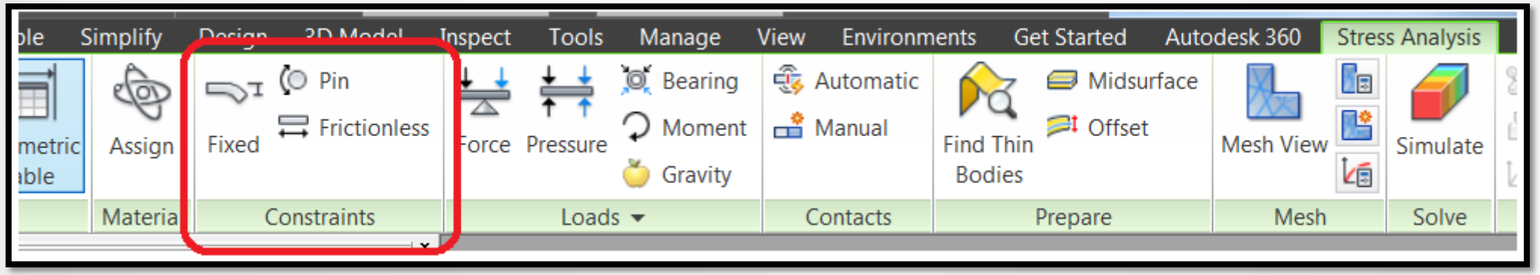


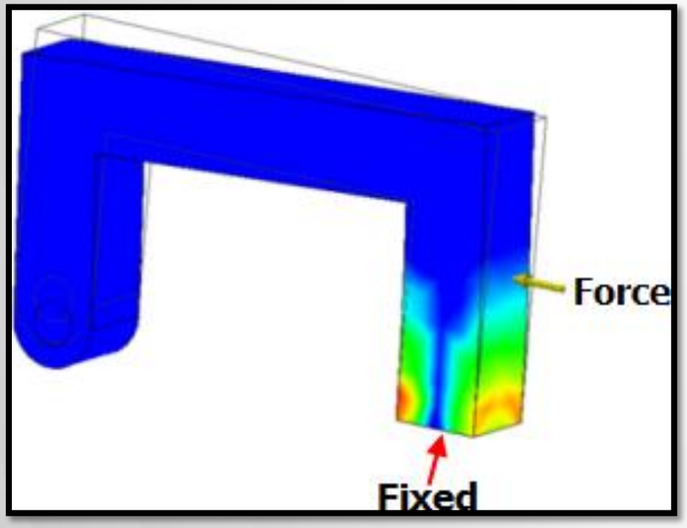


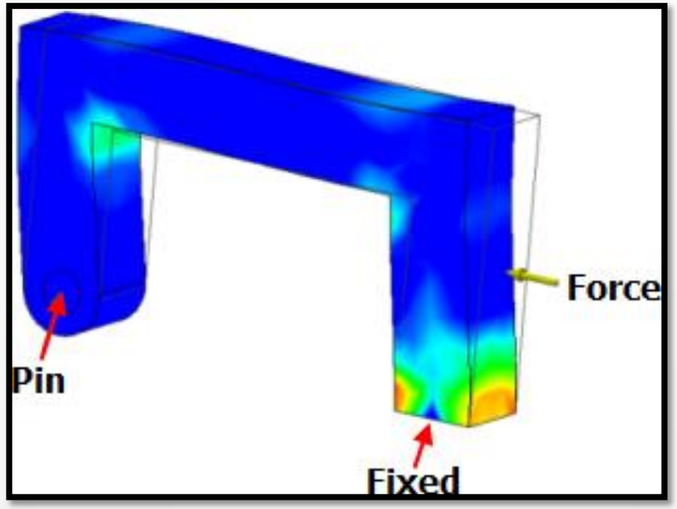
AUTODESK.

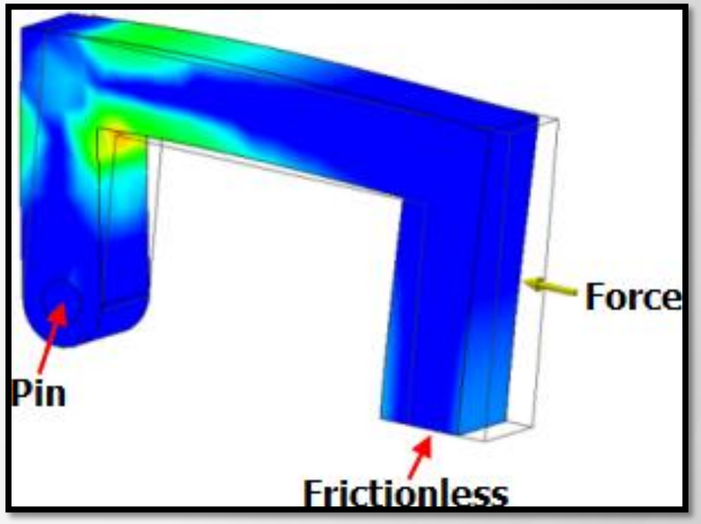
Setup – Define Constraints (Boundary Conditions)

- Constraints define how the parts can (or can't) move





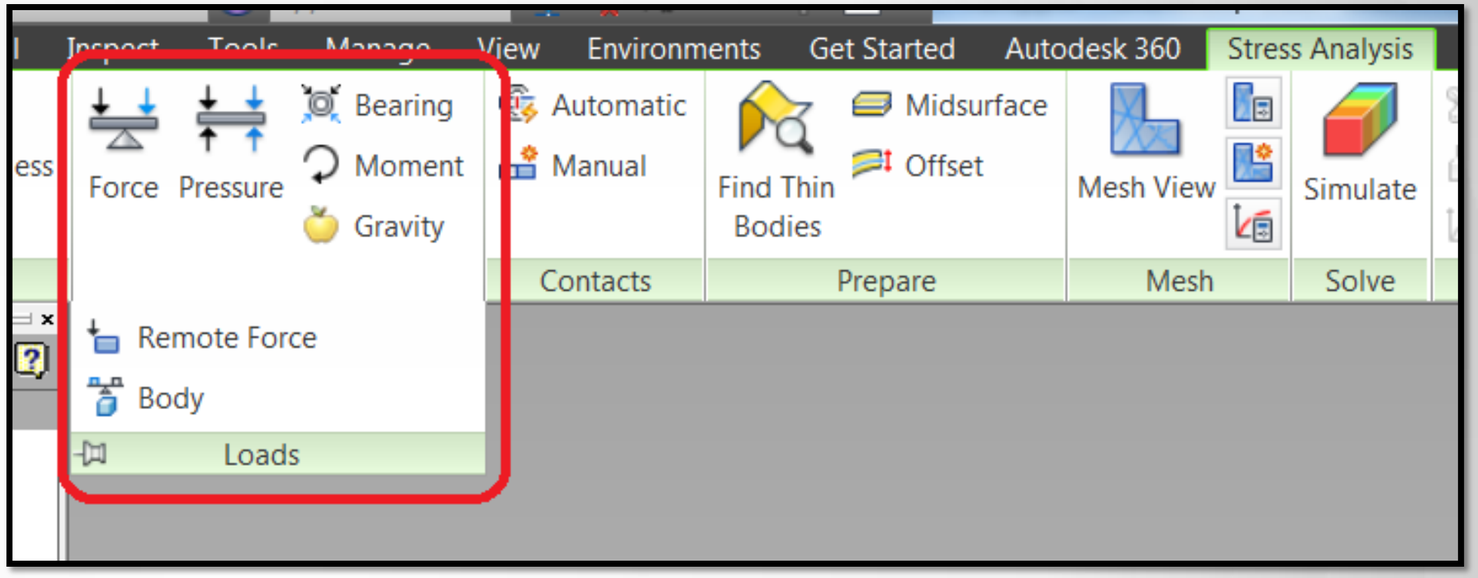


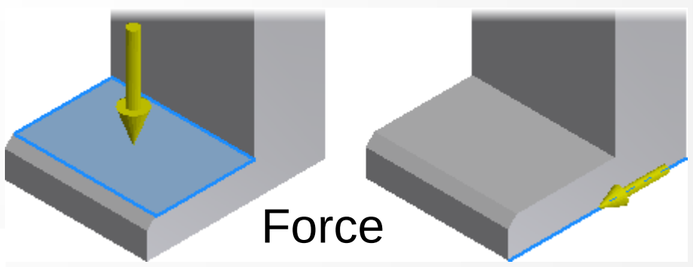


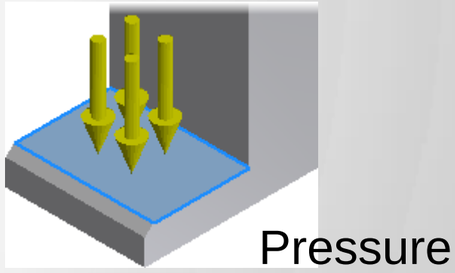
AUTODESK.

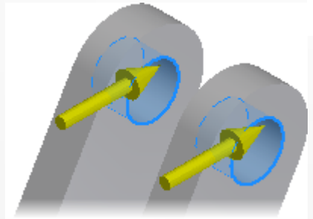
Setup – Define Loads

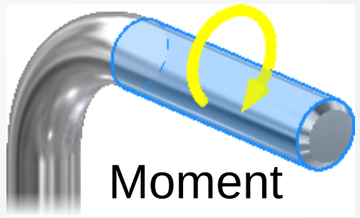
- Loads are defined by
  - Force
  - Pressure
  - Bearing
  - Moment
  - Gravity
  - Remote Force
  - Body







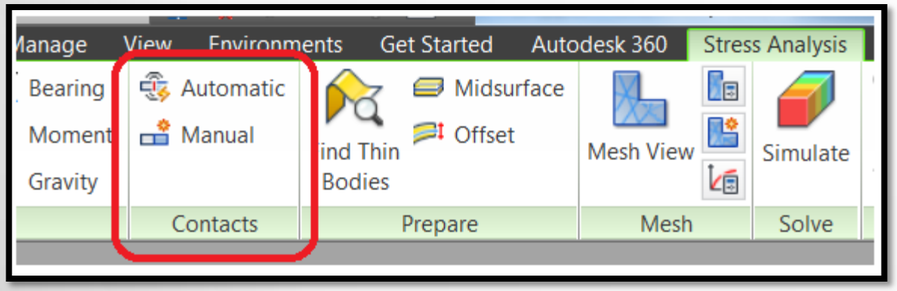


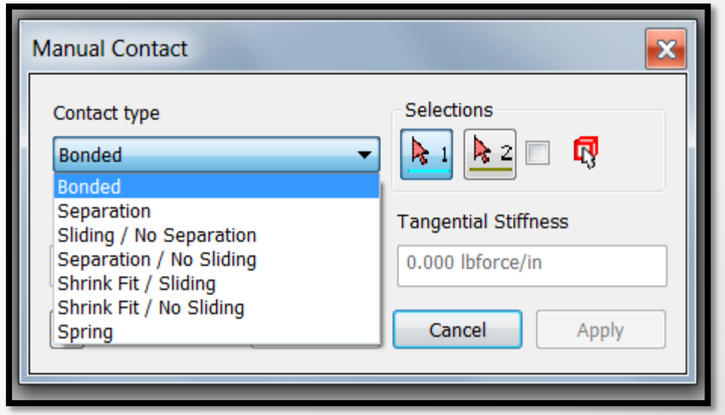


AUTODESK.

Setup – Define Contacts

- Contacts keep surfaces from passing through each other
  - Contacts can be part to part (assembly) or self contact (lock washer)
  - Automatic – Converts constraints and assembly joints into contact type
  - Manual – Gives more control over defining the type of contact

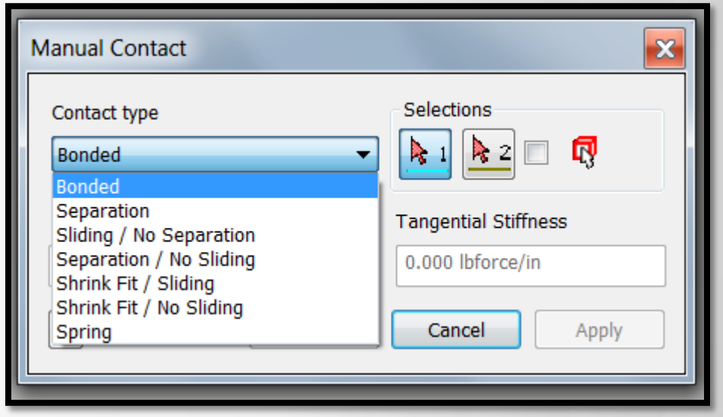




AUTODESK.

Contacts Definitions

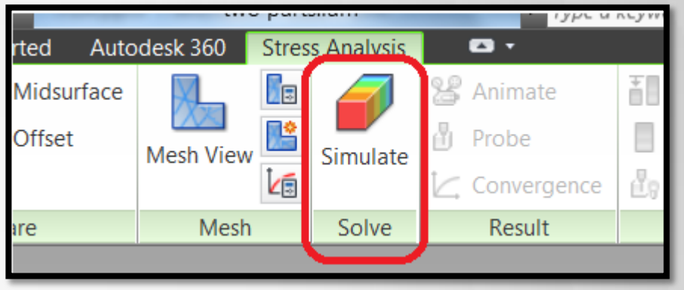
Bonded	Bonds contact faces to each other rigidly.
Separation	Separates contact faces partially or fully while sliding
Sliding / No Separation	Bonds contact faces in normal to face direction while sliding under deformation.
Separation / No Sliding	Separates contact faces partially or fully without their sliding against each other.
Shrink Fit / Sliding	Provides conditions like 'Separation' with initial parts overlapping. The initial distance between the contact faces is negative
Shrink Fit / No Sliding	Provides conditions of Separation/no sliding with initial parts overlapping, meaning negative initial distance
Spring	Creates equivalent springs between the two faces. You define total Normal and/or Tangential stiffness.

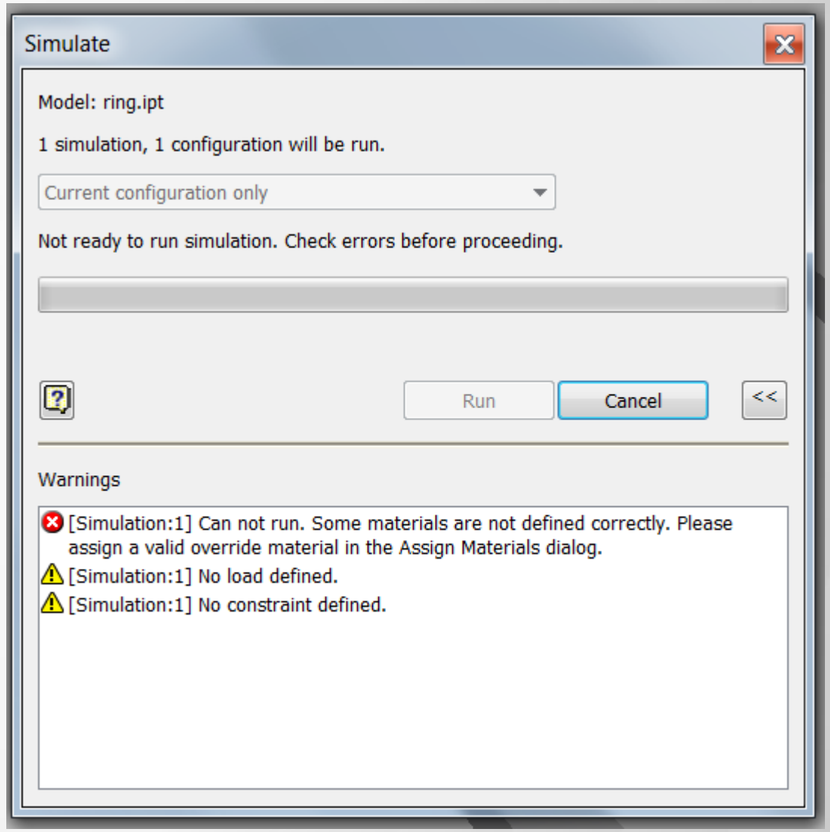


AUTODESK.

Setup – Starting the Simulation

- Click “Simulate”
  - This will verify the material properties of the parts
  - The dialog will give you information about the simulation and let you know if something is wrong

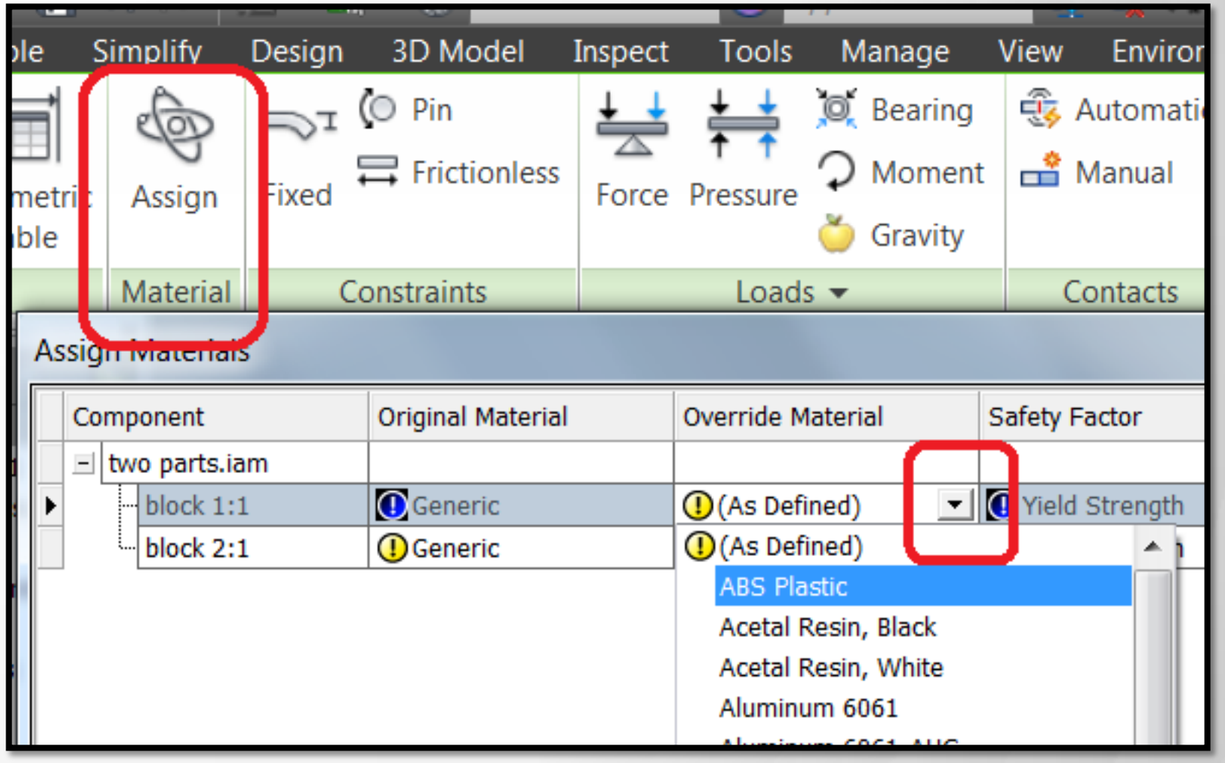




AUTODESK.

Changing Material in Stress Analysis Environment

- Sometimes you will not realize a component does not have a material assigned to it until you are in the Stress Analysis environment



AUTODESK.

DEMO

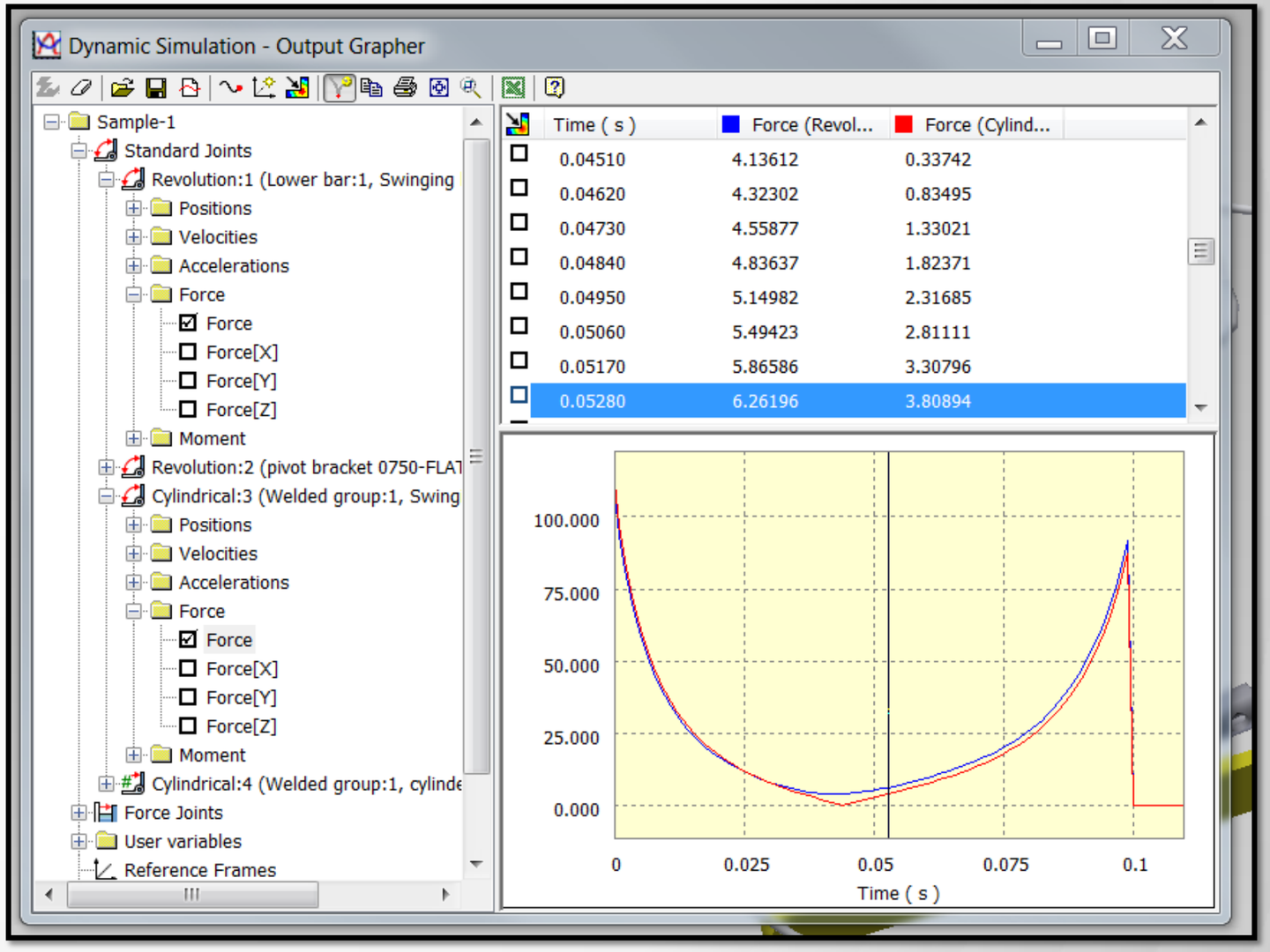
- Single Part Stress Analysis
- Assembly Stress Analysis
- Excluding Features
- Animating Results

Dynamic Simulation - Refresher

- Assembly constraints and joints are converted to DS joints
- Verify moving parts are defined correctly
- Apply loads and/or define impose motion of a DS joint
- Define simulation duration and time steps
- Review results in the Output Grapher

Dynamic Simulation – Output Grapher Refresher

- Displays graphs and numerical values of all the input and output variables

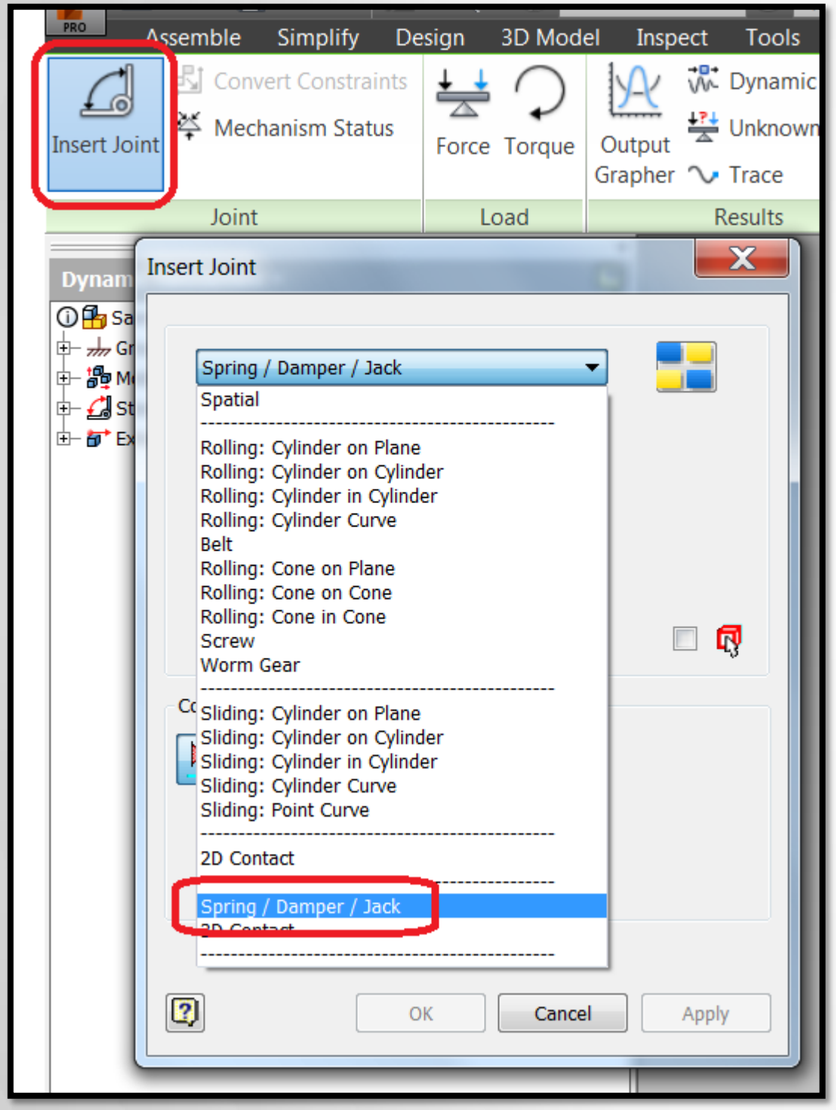


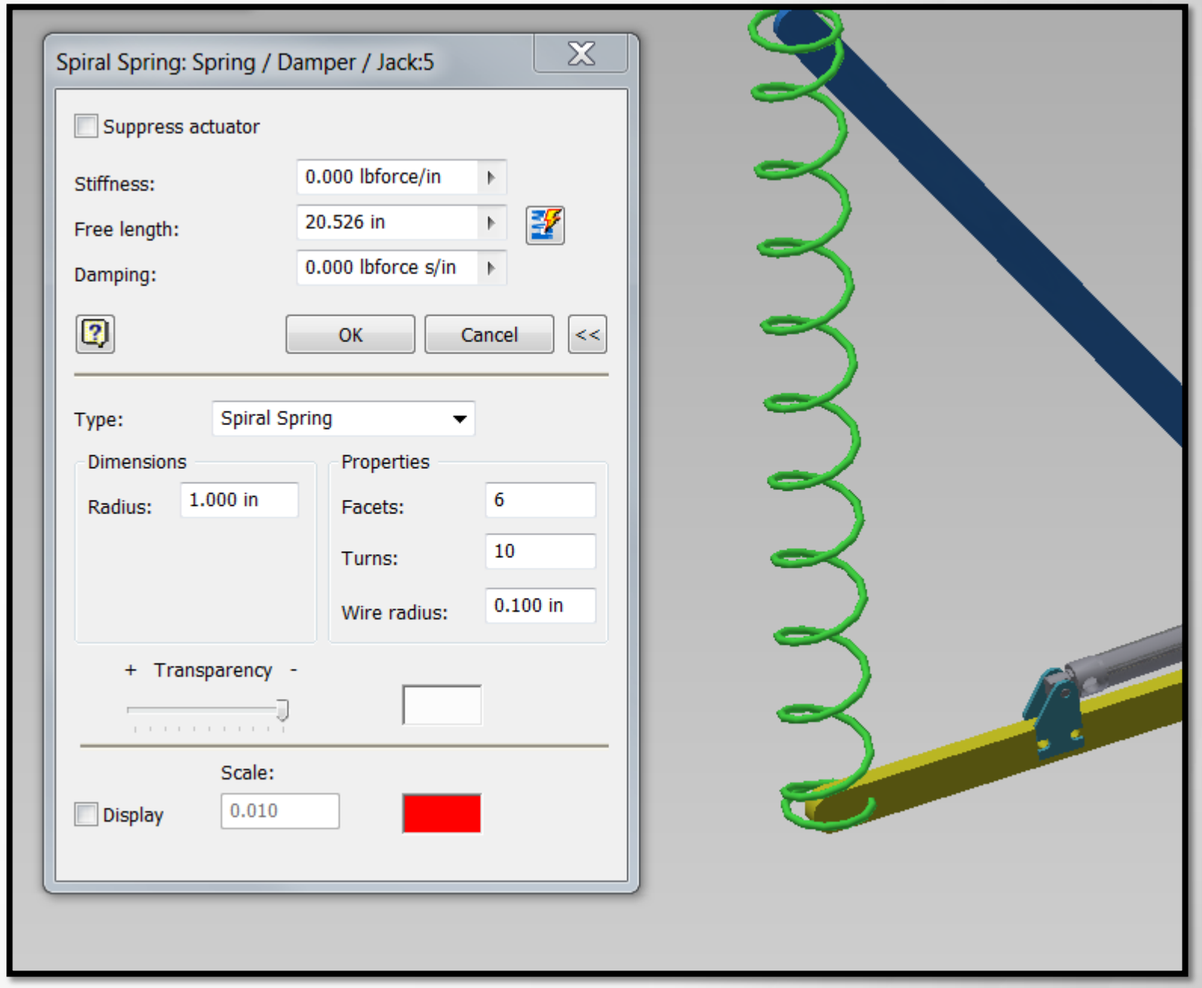
AUTODESK.

Dynamic Simulation – Model Setup

- We are going to look at a simple mechanism using a pneumatic cylinder with a spring.
  - Time to extend the cylinder 3 inches is 0.1 second
  - Apply a “Spring” force joint
  - Review resulting loads in joints

Adding a Dynamic Simulation Spring





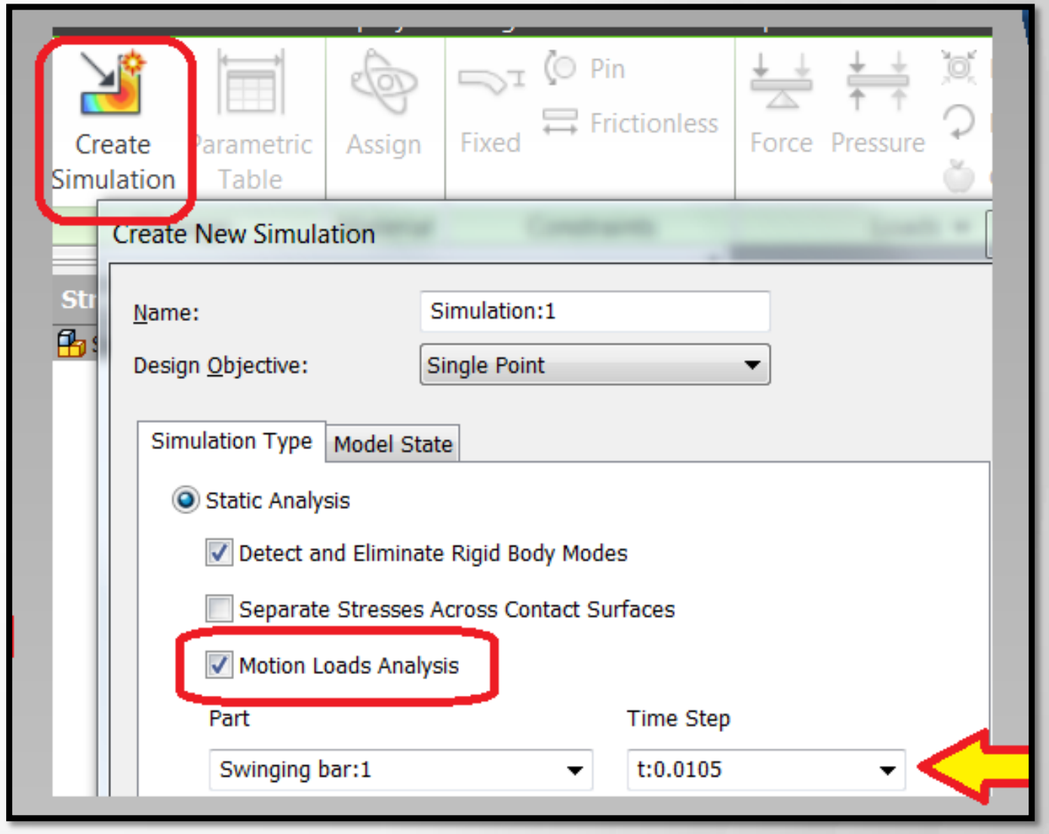
AUTODESK.

Exporting Forces from Dynamic Simulation

- Select maximum forces for export
  - Run the simulation
  - Choose the time step where max force occurs
  - Select part to analyze
  - Select faces the joints act on

Importing Forces into FEA

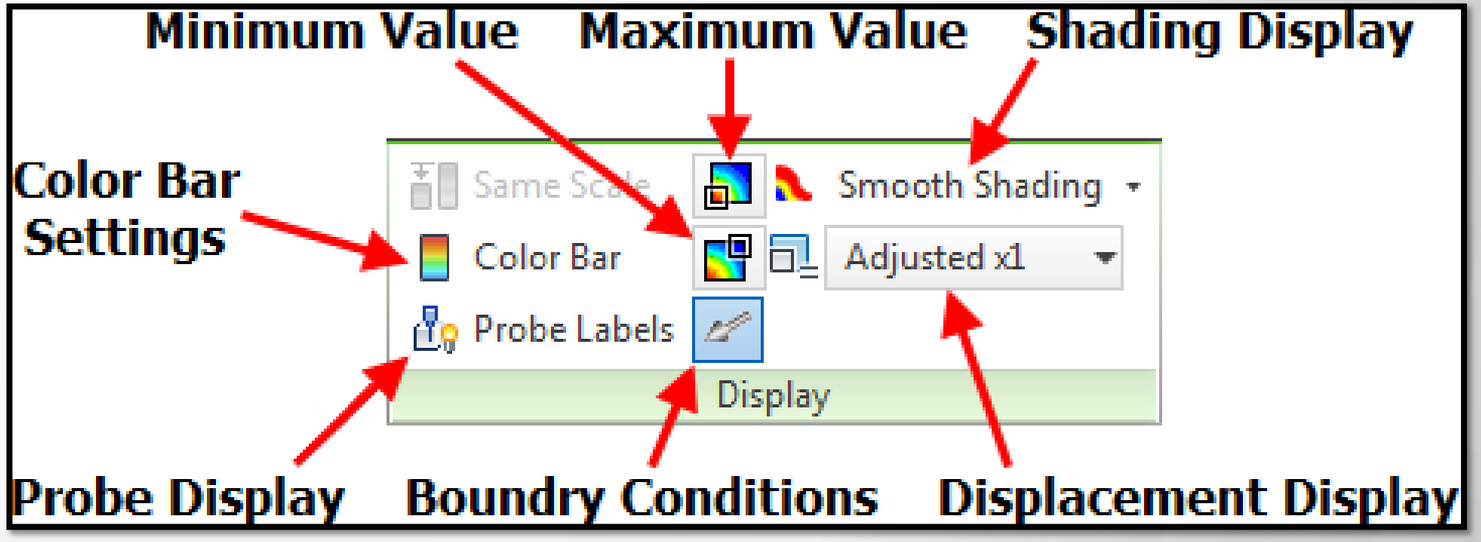
- With the mechanism open, switch to Stress Analysis
  - Create a new simulation
  - Choose “Motion Loads Analysis
  - Select the time step
  - Hit “ok”
- This will bring in the DS loads
- Review the imported loads
- Hit “Simulate” to solve
- Review the results with a mentor



AUTODESK.

Reviewing the results

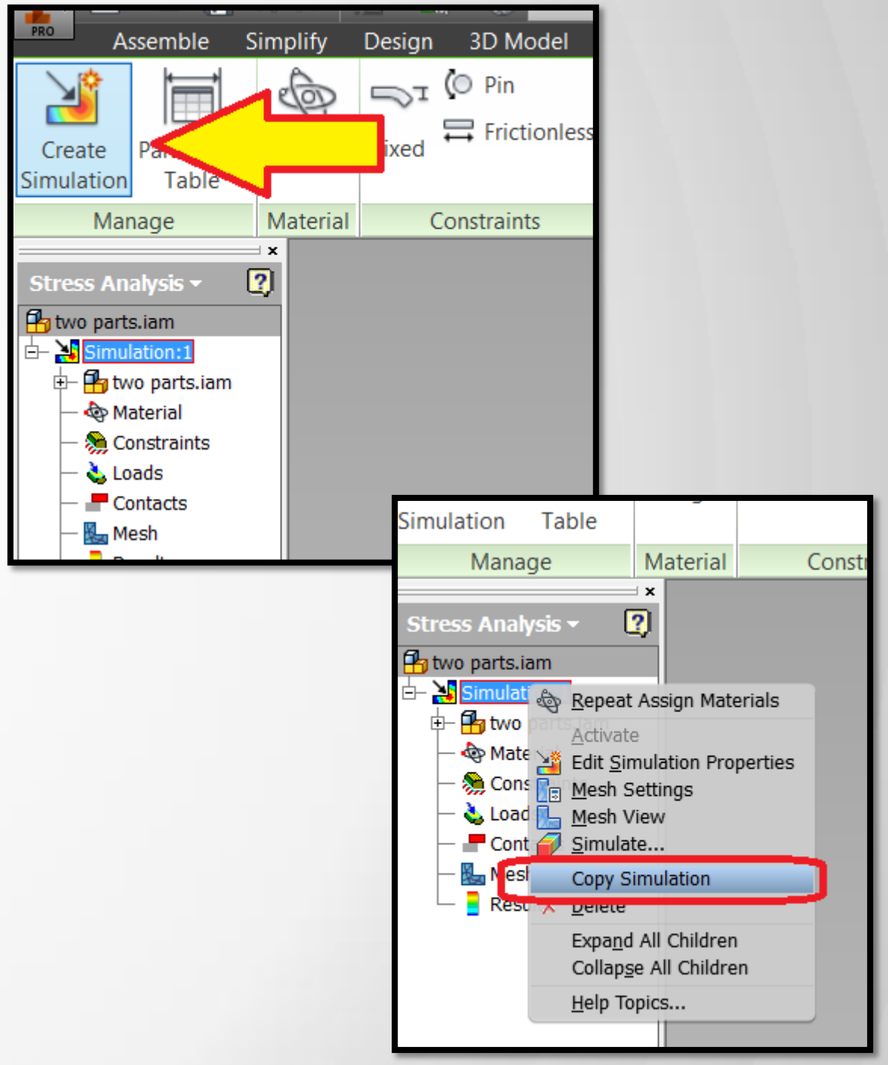
- Options to help reviewing the results
  - Minimum values
  - Maximum values
  - Shading Display
  - Displacement Display
  - Boundary Conditions
  - Probe Display
  - Color Bar Setting



AUTODESK.

Multiple Simulations

- Most of the time you are trying to find an answer to “will UHMW be strong enough?” or “Is the displacement to much?”
- Verify multiple materials in one analysis by copying the simulation or creating a new one



AUTODESK.

Factor of Safety

- How confident are you that something won't break?
- Safety factors for common components
  - Aircraft components = 1.5-2.5
  - Bolts = 8.5
  - Lifting equipment = 8.5
  - Structural steel in buildings = 4-6
  - Structural Steel in bridges = 5-7

DEMO

- Exporting loads from DS and reviewing results in Stress Analysis