AUTODESK UNIVERSITY 2014

Finite Element Analysis for the Casual User in Inventor

JD Mather – Pennsylvania College of Technology

MD6583 This is an introductory class into the advanced topic of finite element analysis (FEA). This class will be application-oriented rather than theory oriented, and it is intended for those of us who can gain design value from some basic FEA techniques without having to have a PhD in mathematics. The goal is to bring the accessibility of advanced Digital Prototyping tools to the casual user. Each year, the "expert systems" software development brings practical accessibility to more and more users. You should jump in here and start learning these tools - now is the time. You can leave your calculator at home.

Learning Objectives

At the end of this class, you will be able to:

- Learning objective 1 – Consider the meaning of life: How do you know you have the answer?
- Learning objective 2 – Learn how to identify the scope and limitations of FEA in Inventor software.
- Learning objective 3 - Setting up FEA of simple assemblies in Autodesk Inventor.
- Learning objective 4 – Learn how to do a parametric dimension FEA test of part iterations in Inventor software.

About the Speaker

Dr. J. D. Mather is an assistant professor of engineering design technology at Pennsylvania College of Technology, Williamsport, PA. He previously worked in industry positions for 15 years, including as a journeyman machinist, a research and development technician, and an industrial engineering technician. J.D. is an Autodesk Inventor Certified Professional and an Autodesk AutoCAD Certified Professional, as well as a Certified SolidWorks Professional. He also serves as an Autodesk Expert Elite on the Autodesk Community Forums. J.D. received the 2010 Autodesk Manufacturing Community Achievement Award. Contact him at jmather@pct.edu.

Learning Objective 1 – Consider the meaning of life: How do you know you have the answer?

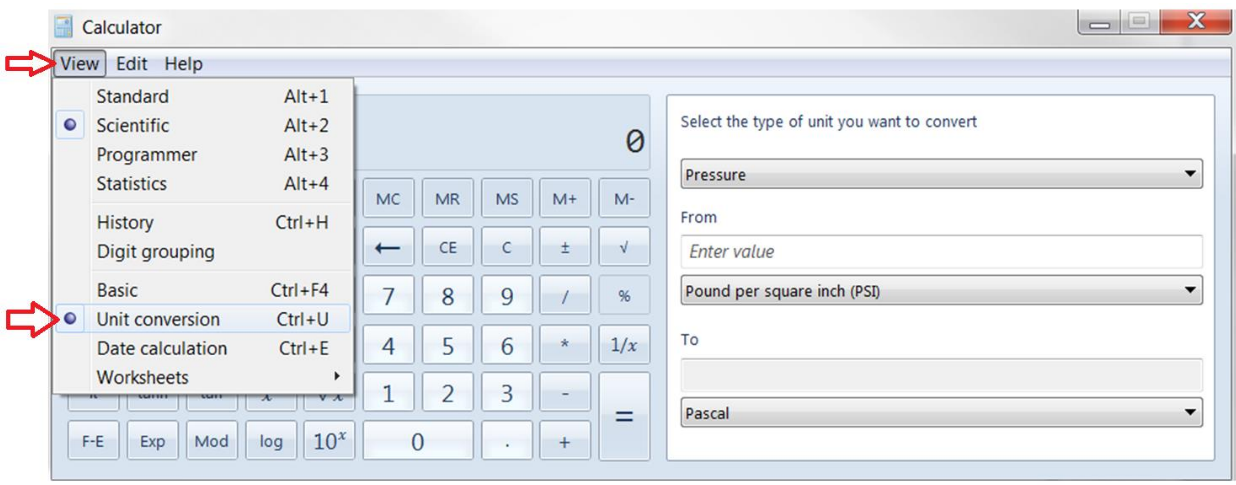
Why do FEA?

Predictive
Validation
Weight Reduction
Material Optimization
Marketing?

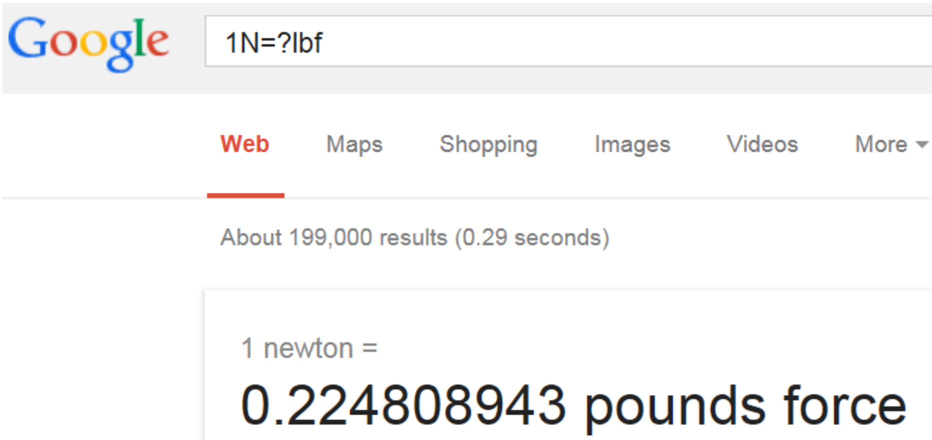
Units – numbers without units lack meaning.

Part of the problem in identifying whether or not you have the “answer” is that the numbers might be very large or very small or have unfamiliar units.

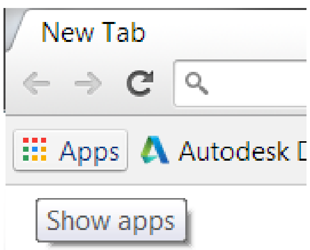
Windows Calculator
Google Unit Conversion
Autodesk ForceEffect (what is Force and what is Pressure?)

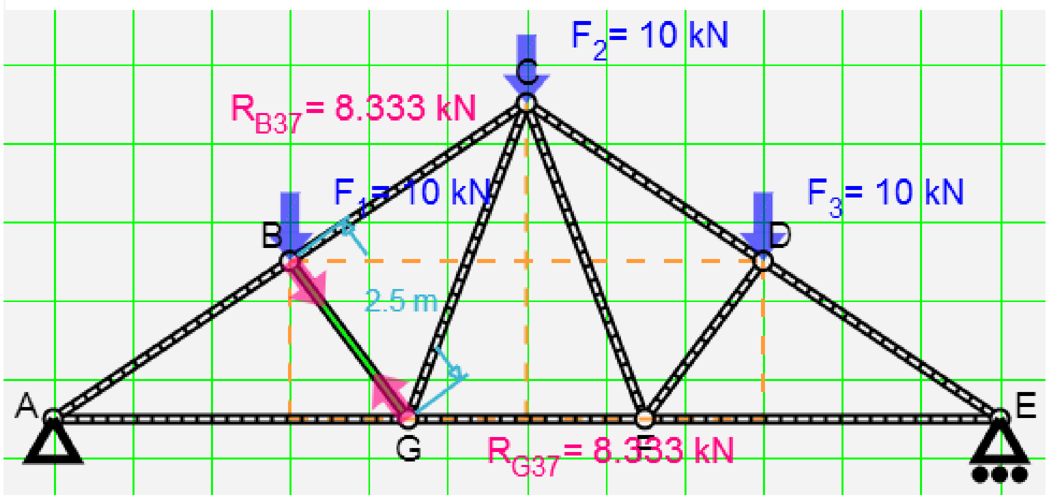


Note that you can get access to additional functionality of Scientific and Unit conversion within the Windows Calculator.



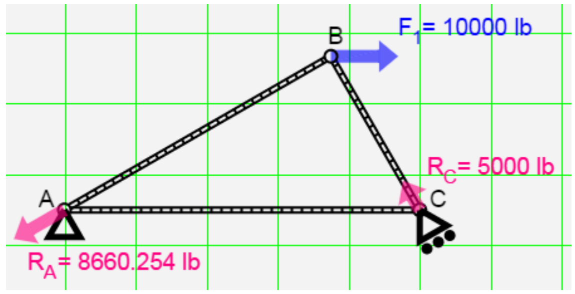
Note that Google will return direct unit conversions.



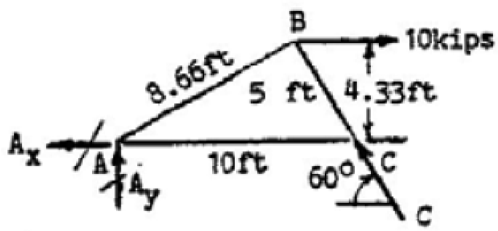


Autodesk ForceEffect can be used for some preliminary calculations.

9.

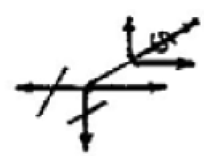


Reference:
Statics and Strength of Materials, 7th Ed. H.W. Morrow.



$$\sum M_A = 8.66C - 10(5 \sin 60°) = 0$$
$$C = 5 \text{ kips @ } 120° \qquad <$$
$$\sum F_y = A_y + C \sin 60° = 0$$
$$A_y = -4.33 \text{ kips} \qquad <$$
$$\sum F_x = 10 - A_x - C \cos 60° = 0$$
$$A_x = -7.5 \text{ kips} \qquad <$$

Joint A



$$\sum F_x = 0.5F_{AB} - 4.33 = 0$$
$$F_{AB} = 8.66 \text{ kips } T \qquad <$$
$$\sum F_y = F_{AC} + 0.866\, F_{AB} - 7.5 = 0$$
$$F_{AC} = 0 \qquad <$$

Joint C by inspection

$$F_{BC} = -5 \text{ kips } C \qquad <$$

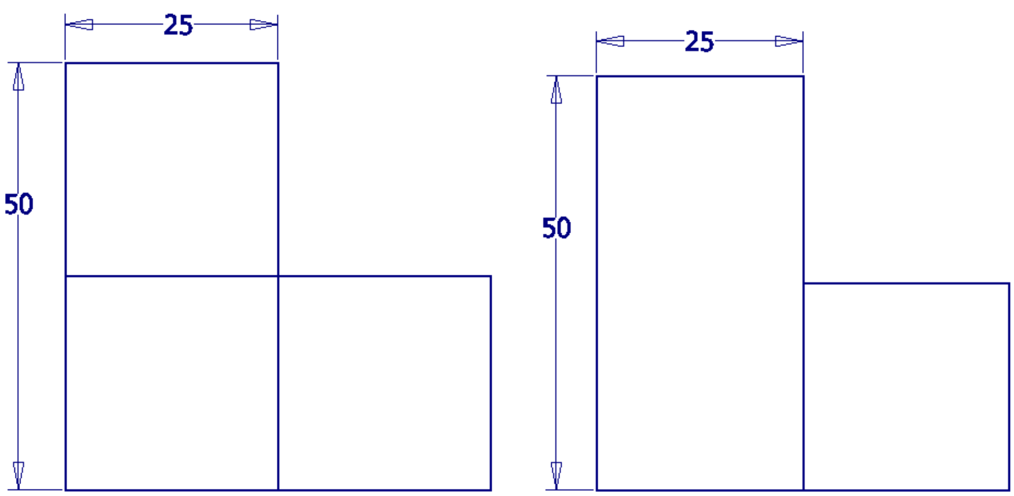
Force and Pressure - Demonstration

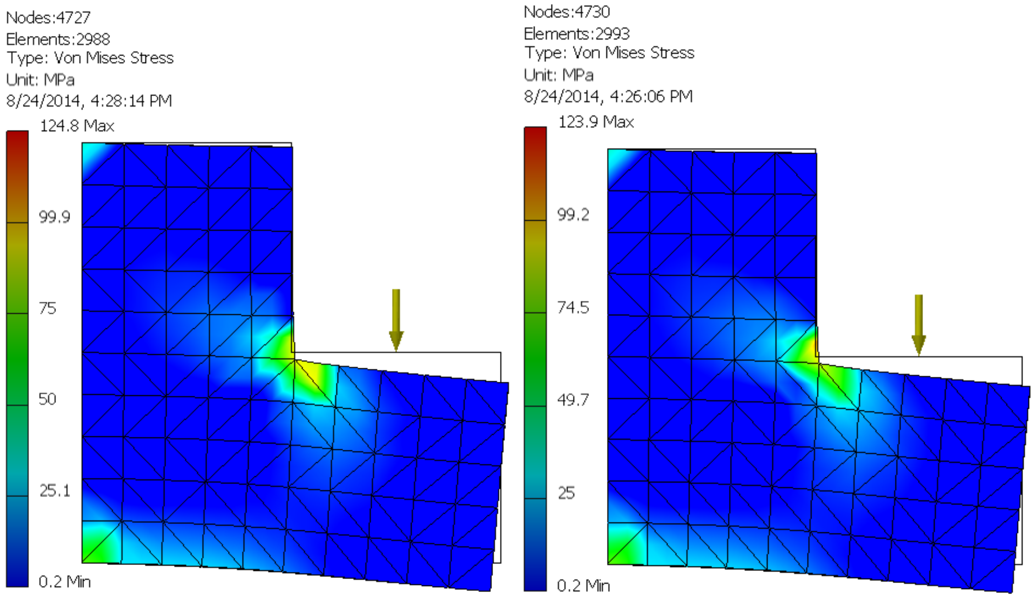
($20 bill - I need a volunteer from the audience to demonstrate the difference between force and pressure.
First I will stand on the volunteer's chest with one foot.
Then I will balance my weight on a nail on the volunteer's chest.)

$$P=F/A$$

Where: P=Pressure; F=Force, A=Area

Demonstration – Why do we get different results with the "same" input conditions?





Percentage Difference Formula
(|123.9-124.8|)/((123.9+124.8)/2))*100)=0.72% difference
http://www.mathsisfun.com/percentage-difference.html
http://www.calculatorsoup.com/calculators/algebra/percent-difference-calculator.php

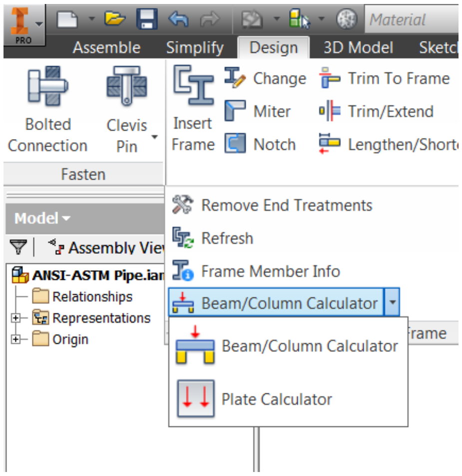
In general we are looking for "ballpark" answers when doing FEA Analysis.

Learning Objective 2 – Learn how to identifying the scope and limitations of FEA in Inventor software.

Theory-based Calculations

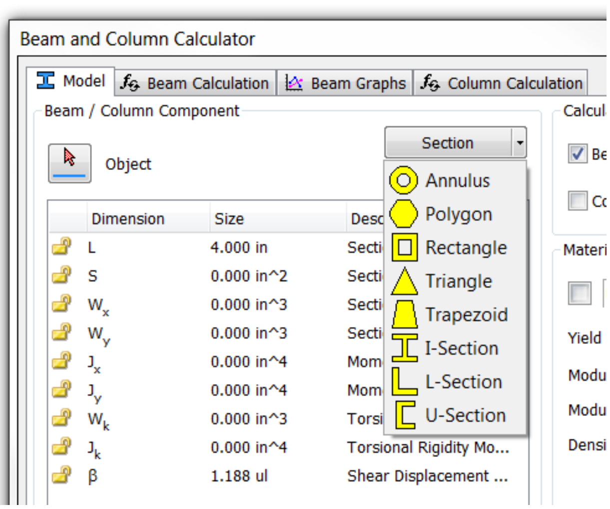
FEA stands for Finite Element Analysis. (consider the implied limitations)

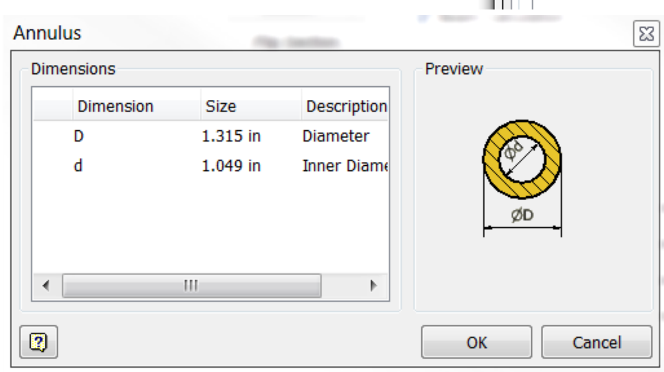
"Hand" Calculations Example ANSI/ASTM Pipe (see Appendix)
Demonstration – Beam and Column Calculator
Demonstration – Inventor FEA Analysis
Demonstration – Inventor Frame Analysis
Stress/Strain Graph (Reference Younis, 2014, pg 10)
Definition: of Safety Factor



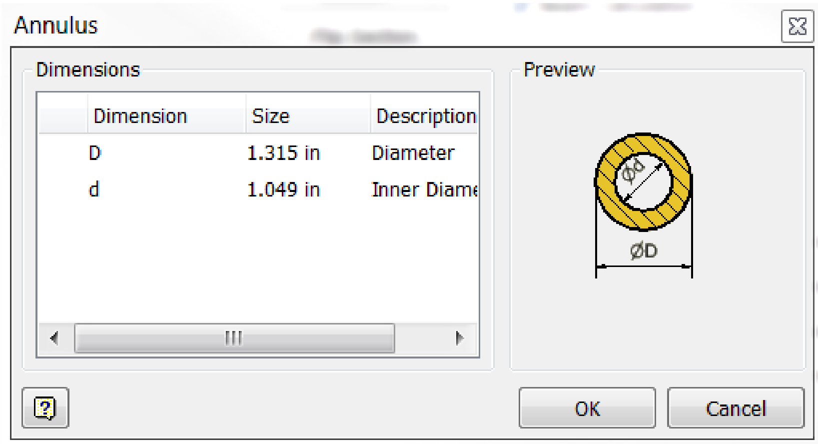
There is a Beam/Column and Plate Calculator in the Inventor Design Accelerators. Simply start a new assembly file (*.aim) and go to the Design tab.

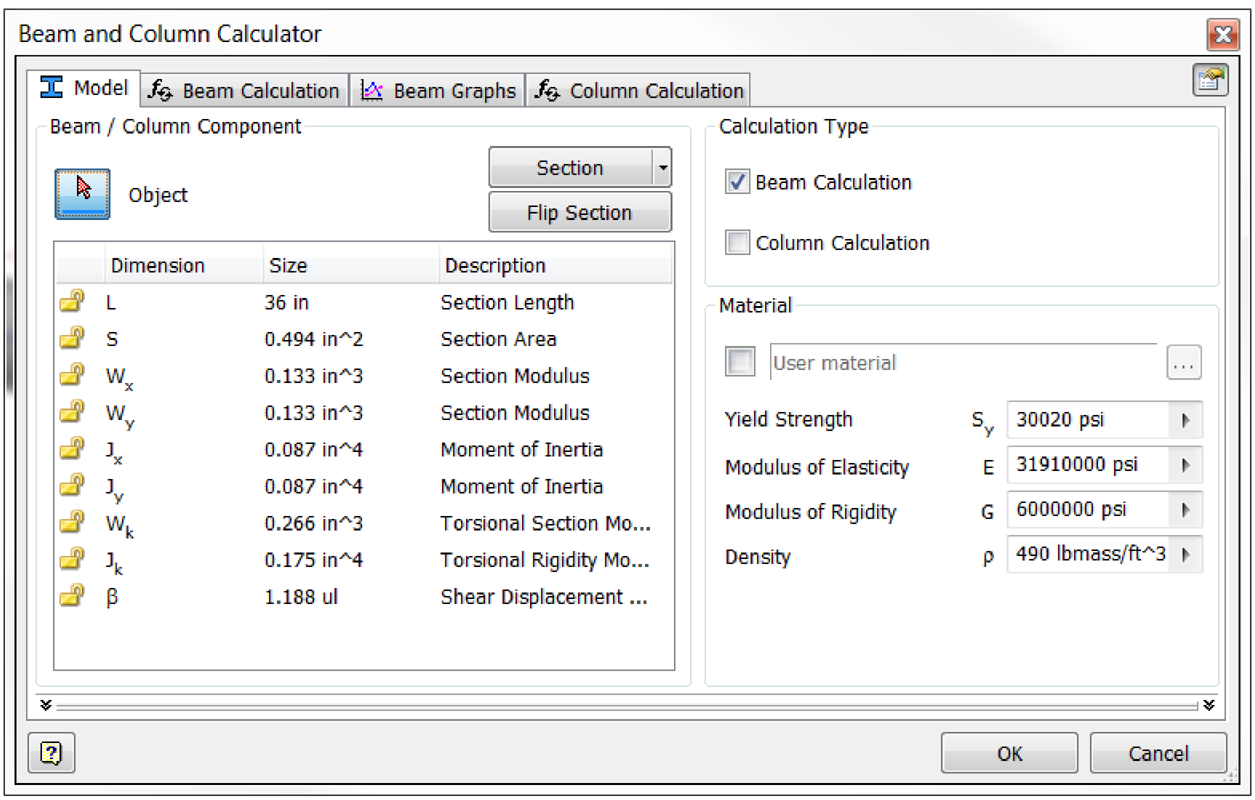
An existing Object (*.ipt) can be selected from the graphics window, or select from the Section profiles.





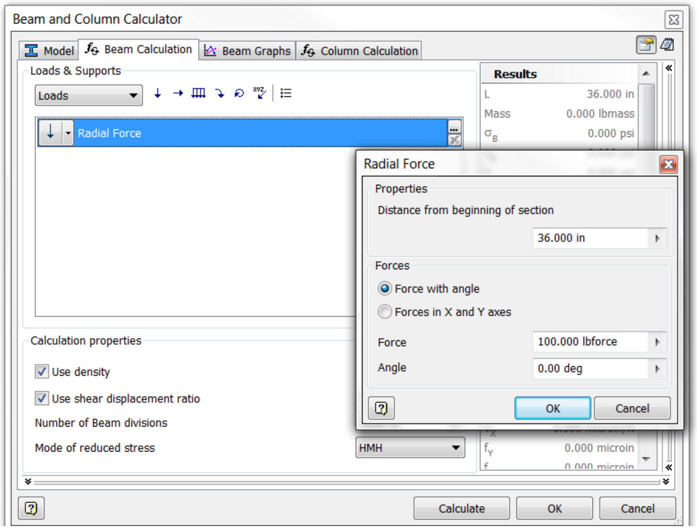
For a nominal size 1" pipe.

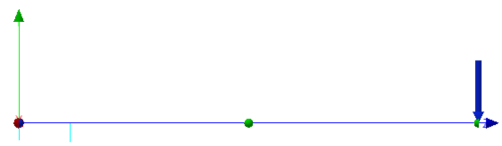




If you click on User Material - there are thousands of Materials in the Design Accelerator.

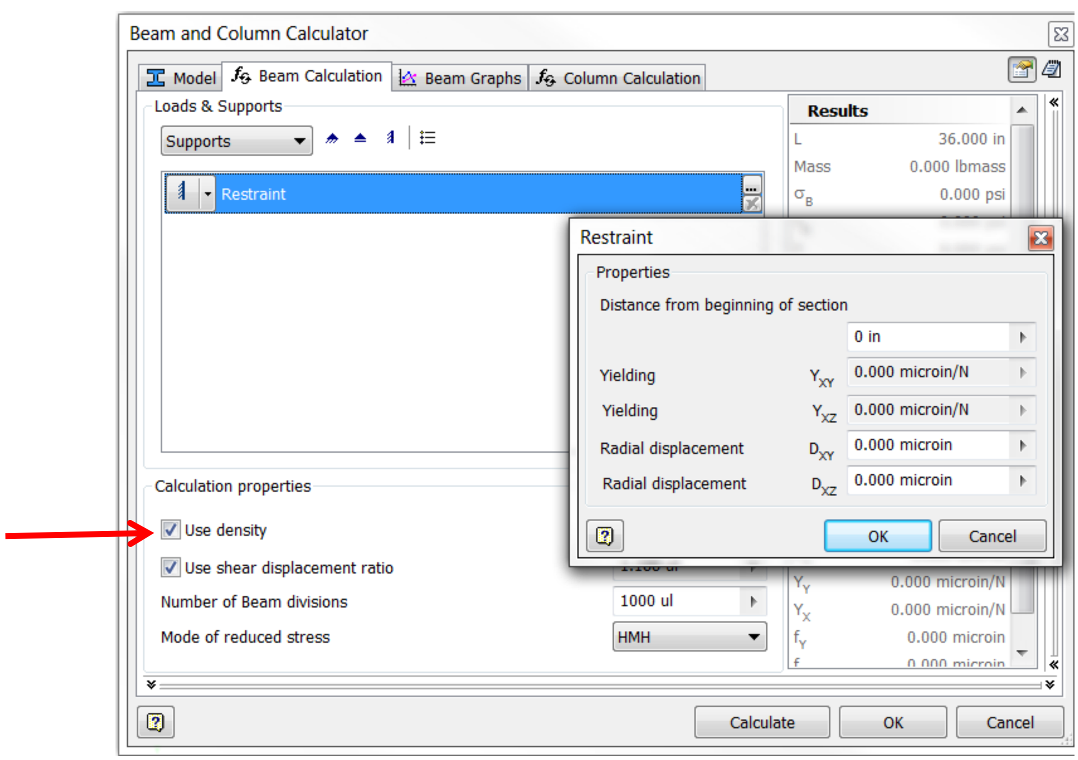
In this case – I entered the Material properties manually.

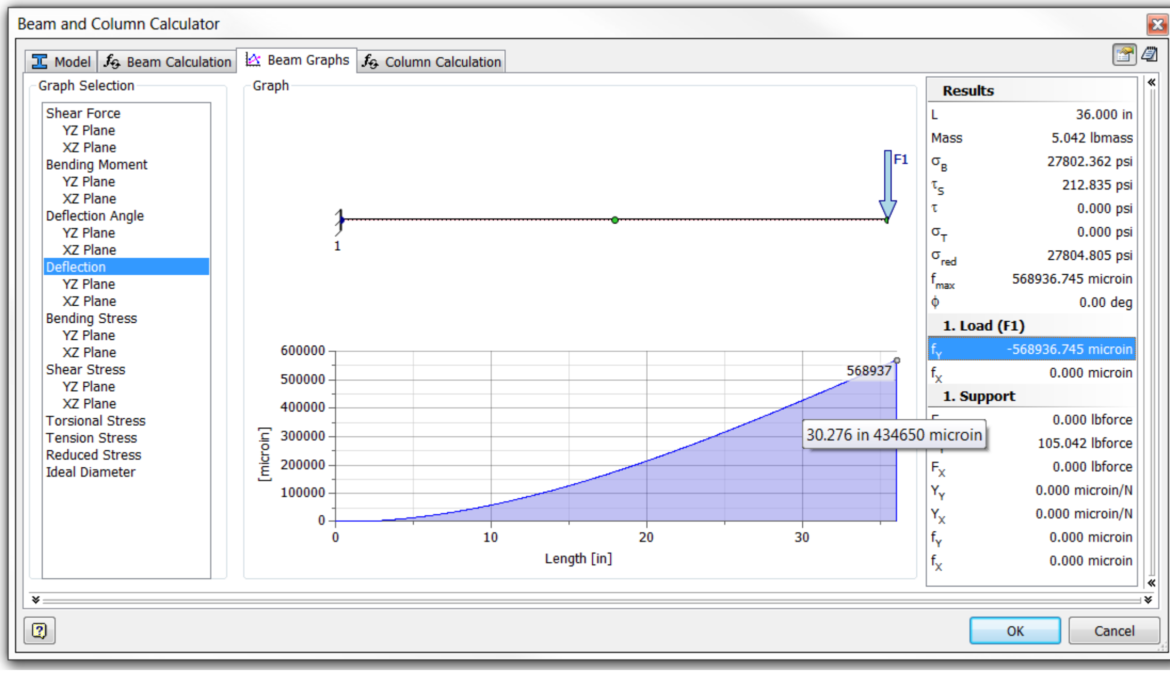




(Do not use Gravity/Mass/Density of part unless you consider that in the hand calculations).

Note in the dialog box, the Use density (for the hand calculation – this would have to be added to the load).
For a pipe this short – it has little effect.



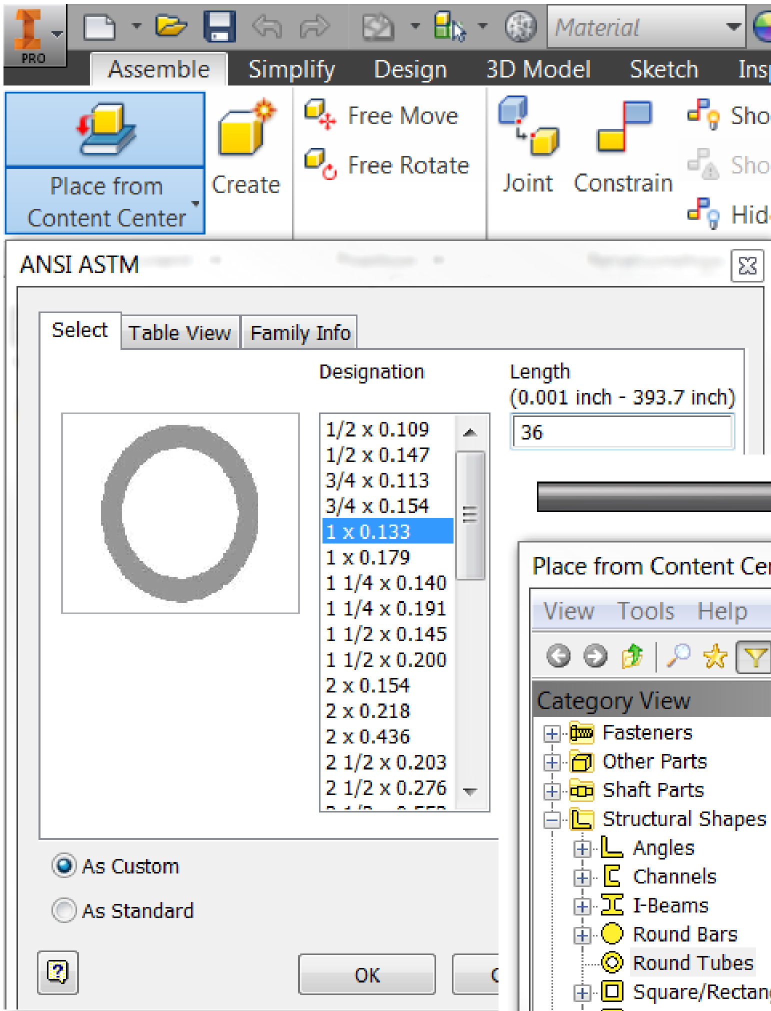


568937 micro inch = .569 inch

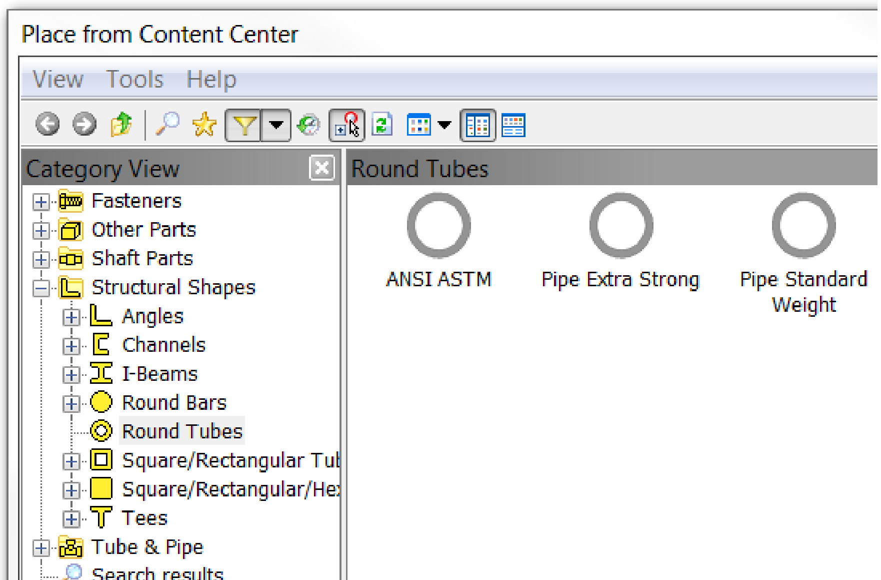
2% difference than hand calculation (***if*** I did the math right).

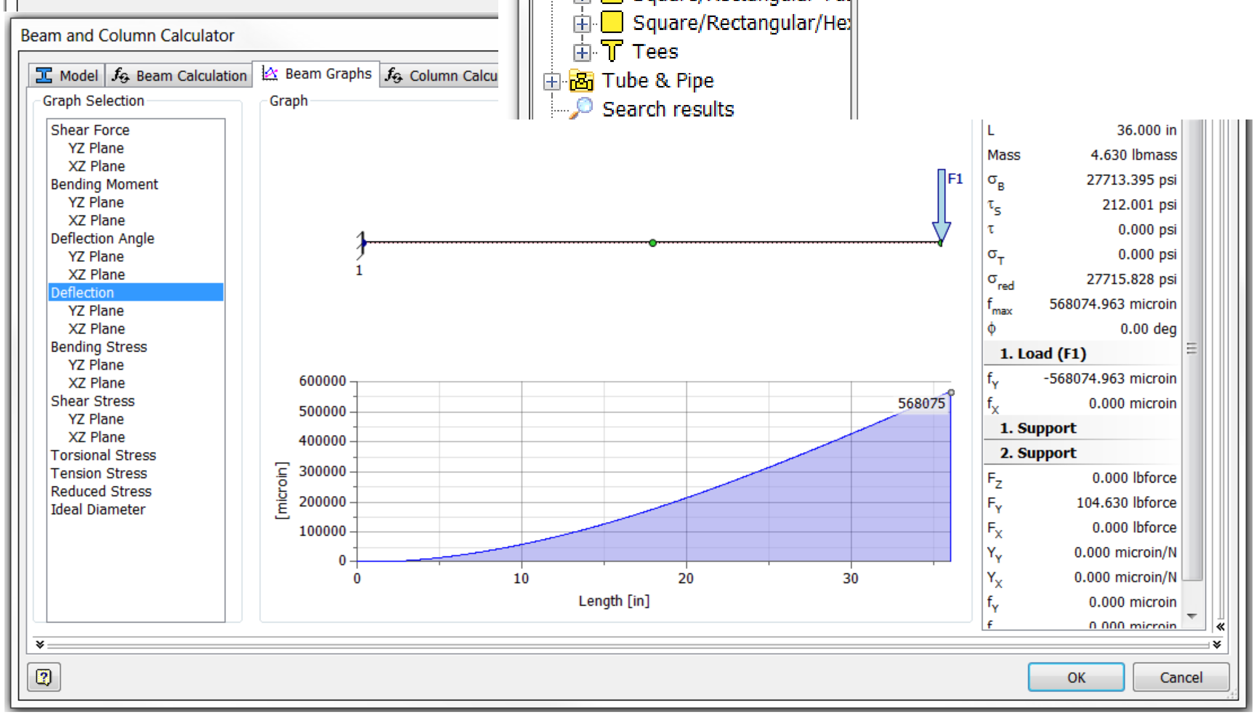
Q. Did I use gravity? If so, then I have added the weight of the beam in addition to F1 into the analysis. Approximately 105lbf rather than the 100lbf specified in the hand calcs problem. (see Appendix)

Next I place the same component from Content Center.

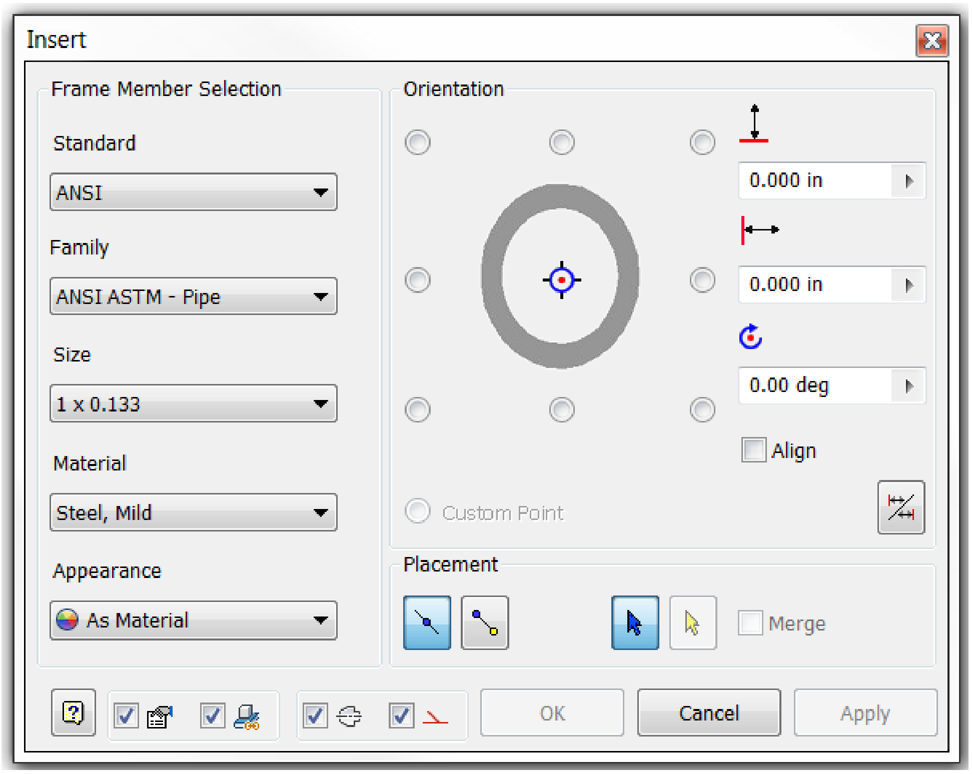


Equivalent results from Content Center part.

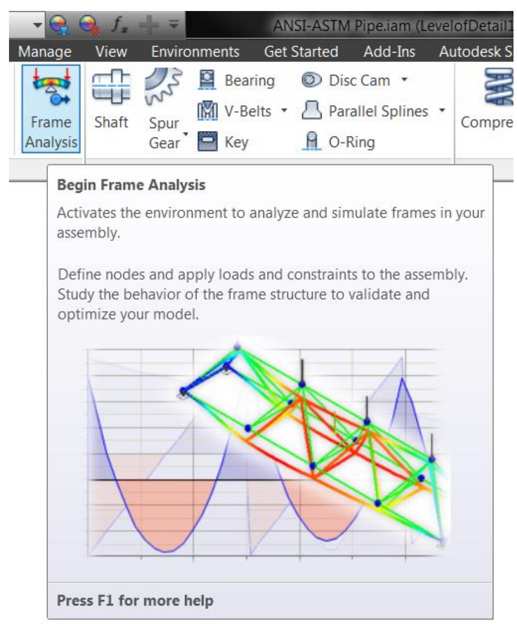




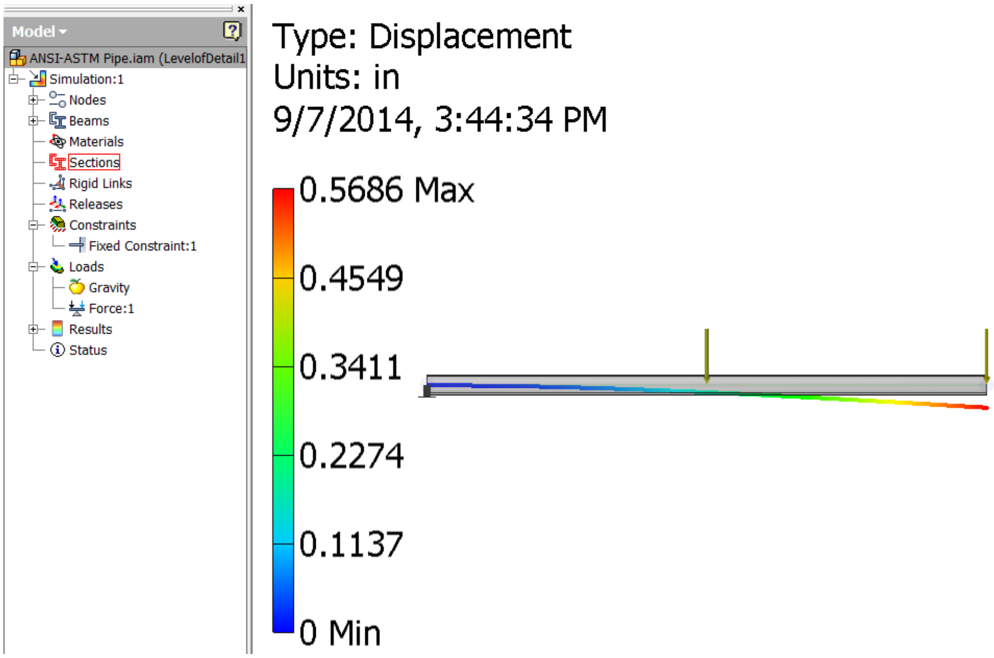
Placing from the Frame Generator.

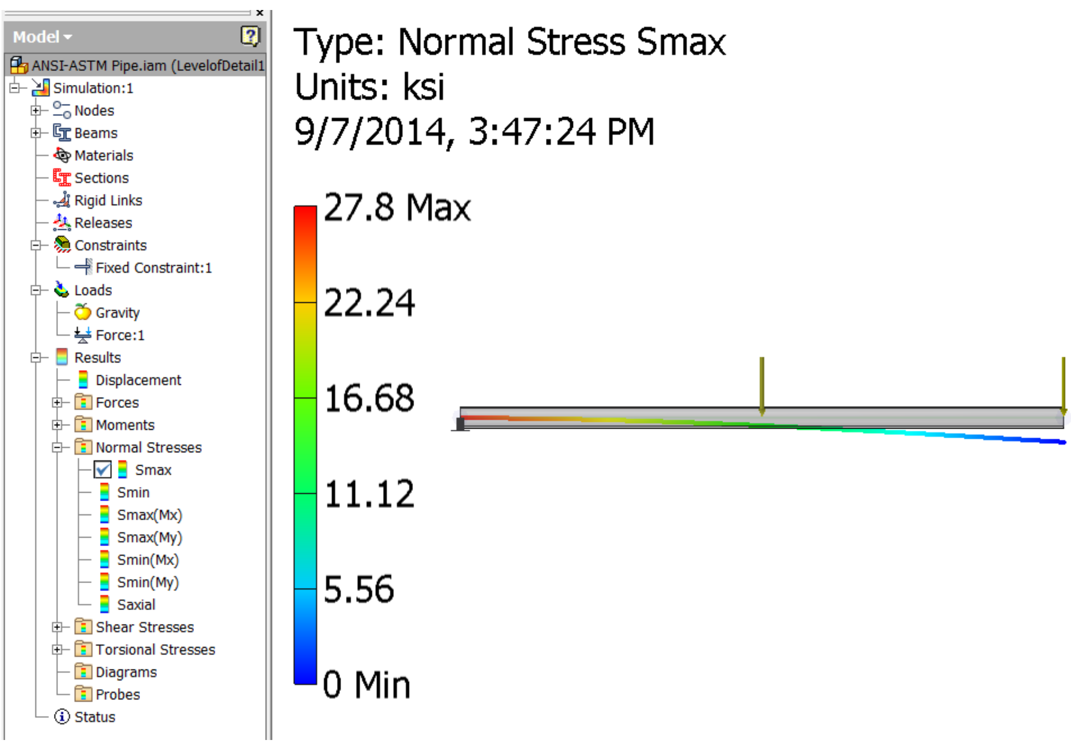


Running Analysis from **Frame Analysis**.

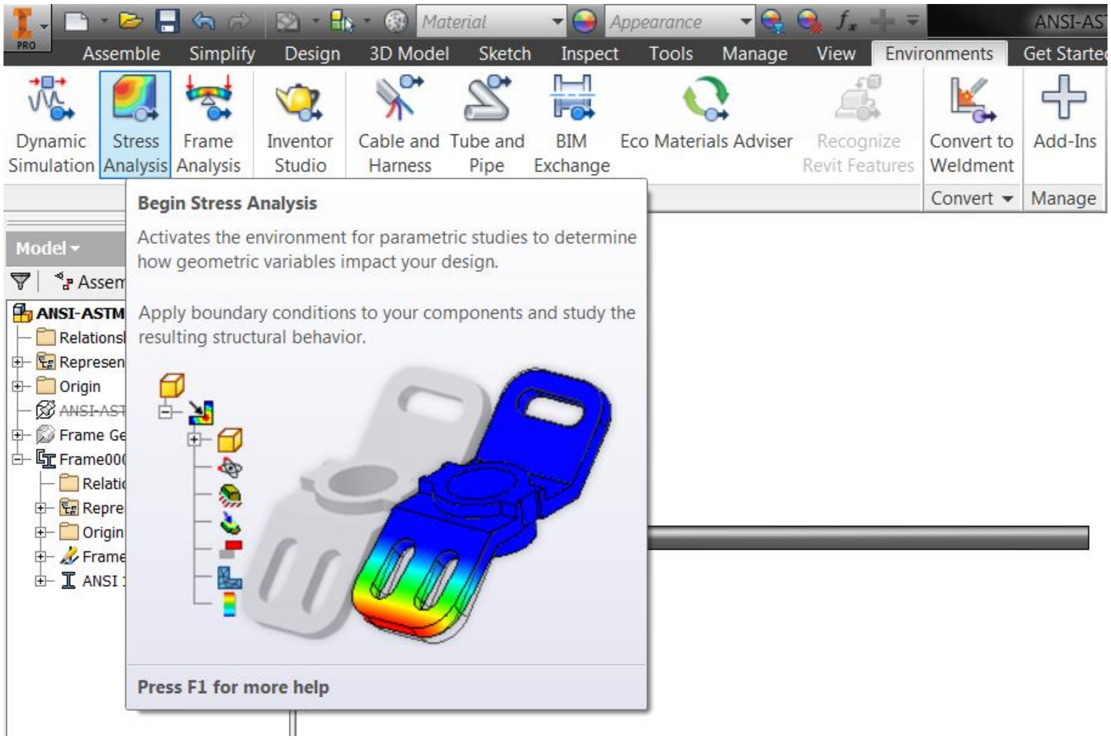


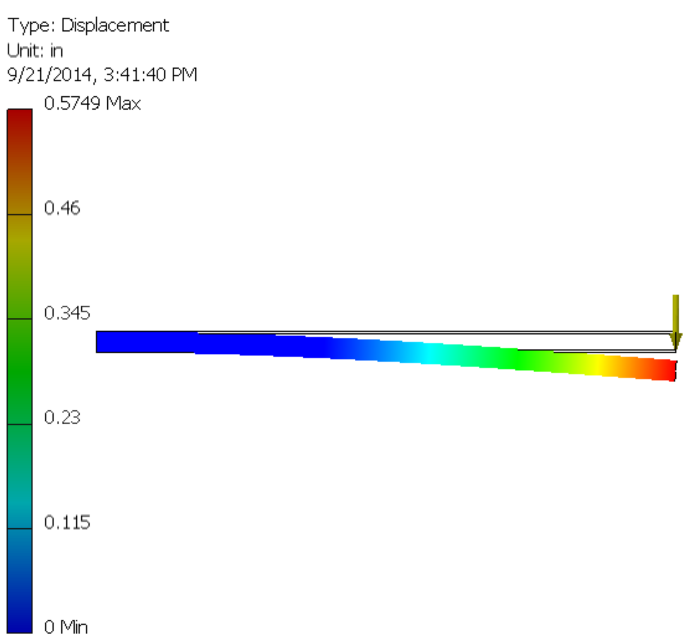
Inventor Frame Analysis can only be used with components created using the Frame Generator – all other components will be ignored.

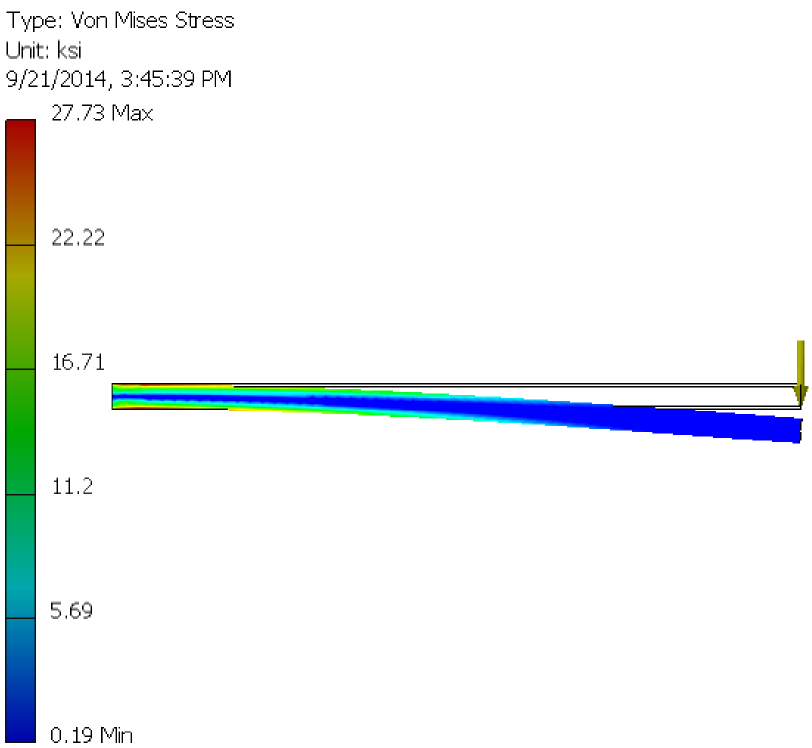

Model
ANSI-ASTM Pipe.iam (LevelofDetail1
Simulation:1
Nodes
Beams
Materials
Sections
Rigid Links
Releases
Constraints
Fixed Constraint:1
Loads
Gravity
Force:1
Results
Status
Type: Displacement
Units: in
9/7/2014, 3:44:34 PM
0.5686 Max
0.4549
0.3411
0.2274
0.1137
0 Min

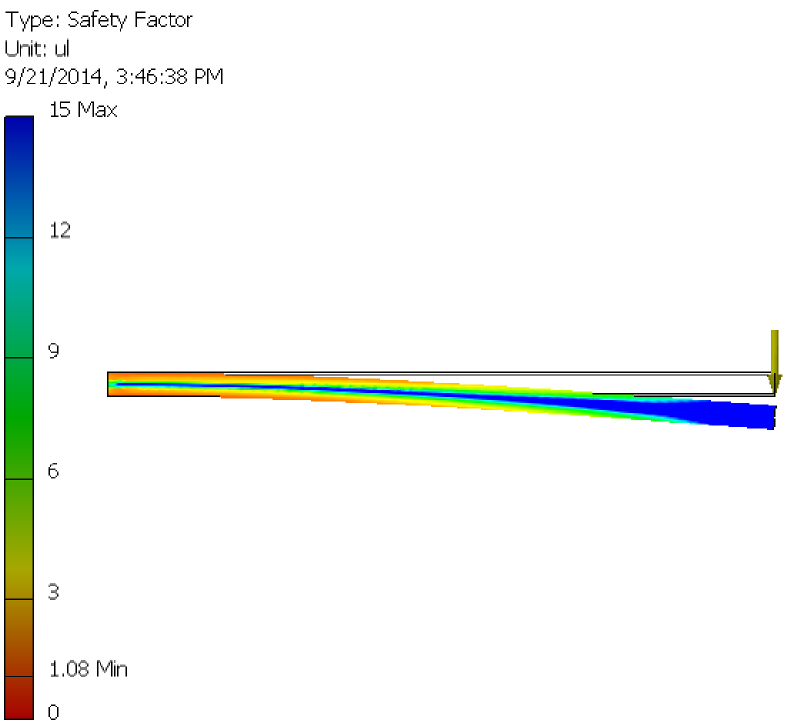

Model
ANSI-ASTM Pipe.iam (LevelofDetail1
Simulation:1
Nodes
Beams
Materials
Sections
Rigid Links
Releases
Constraints
Fixed Constraint:1
Loads
Gravity
Force:1
Results
Displacement
Forces
Moments
Normal Stresses
Smax
Smin
Smax(Mx)
Smax(My)
Smin(Mx)
Smin(My)
Saxial
Shear Stresses
Torsional Stresses
Diagrams
Probes
Status
Type: Normal Stress Smax
Units: ksi
9/7/2014, 3:47:24 PM
27.8 Max
22.24
16.68
11.12
5.56
0 Min

Running Analysis from **Stress Analysis**.







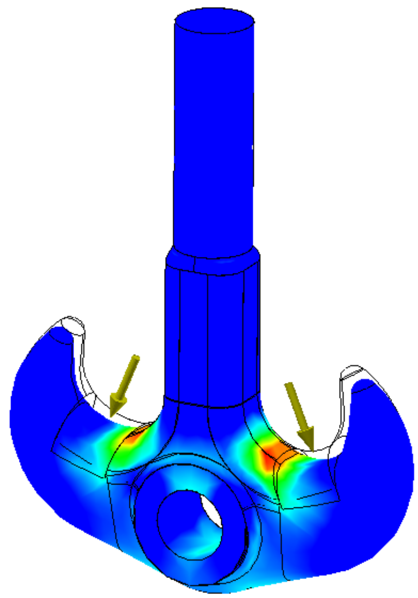


Equivalent results with 4 different techniques.

Experience-based Validation

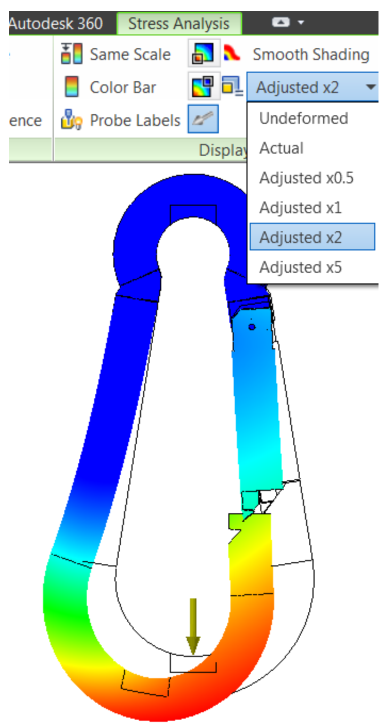
Validate your digital analysis model with empirical data for predictive purposes.

Demonstration – Hook Example (split faces)



We often need to define faces on which to apply our loads and/or constraints.

Demonstration – Carabiner

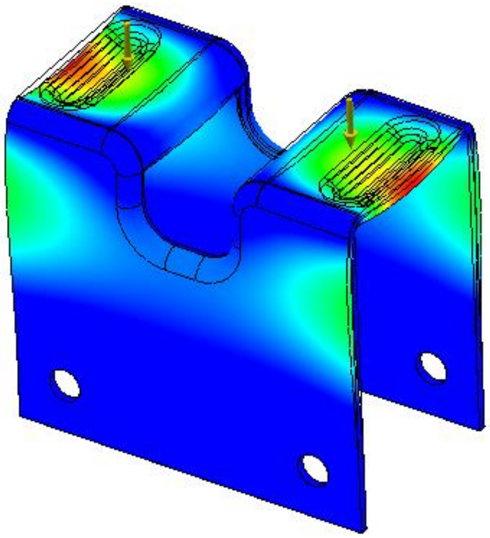


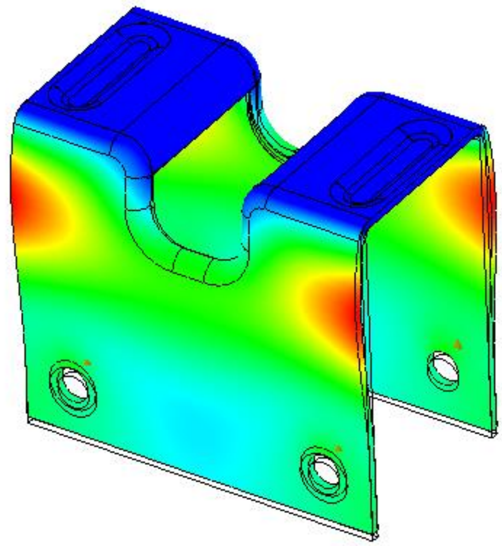
How we present our analysis to non-experts is important.

We can show animations as illustrations.

We can create reports to establish a chain-of-evidence.

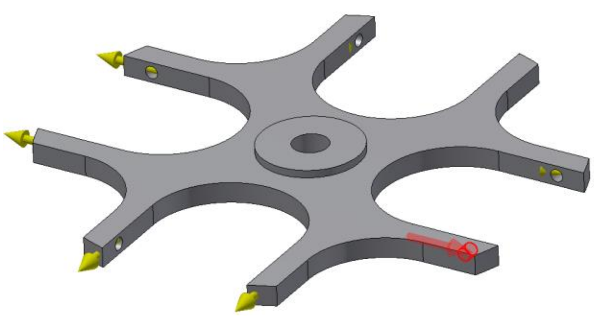
Demonstration – Automobile Jack Lift Plate (what does our experience tell us?)

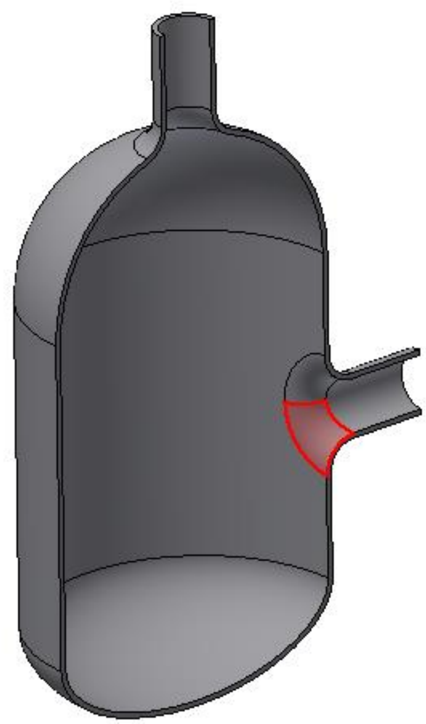




Notes on applying loads/constraints.

Adding pressure load to internal faces – watch out for face seams.





Location of the glyphs is not important.
There was an issue in 2015 SP0 with the arrow glyphs not showing correct direction.

Learning Objective 3 – Setting up FEA of simple assemblies in Autodesk Inventor.

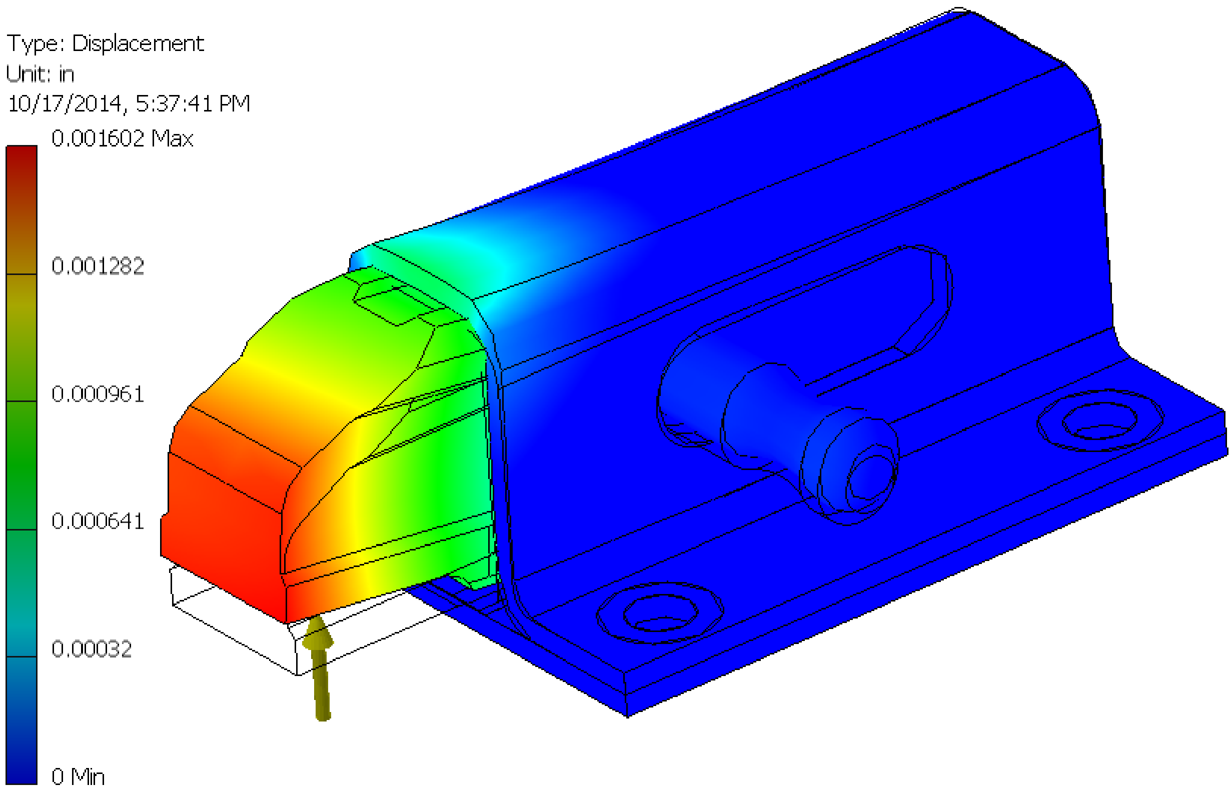
Frame Analysis – the Crane problem – a word about modeling technique.

Contacts

Validate your digital analysis model with empirical data for predictive purposes.

What is the important parameter to us? Calculated Stress? Displacement? Safety Factor? I would argue that it is almost always Displacement while most FEA users would pick one of the other two.

Demonstration – Latch Bolt Example (split faces)

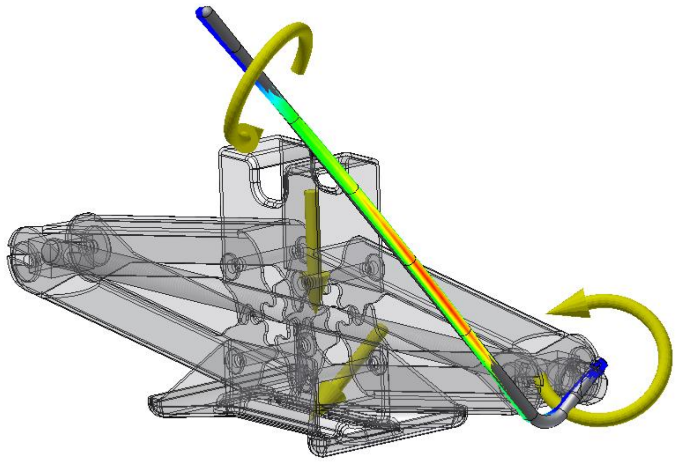


Local yielding can occur initially until the load is transmitted at lower stress by the full joint. (Younis, pg. 135 2014).

Dynamic Simulation

Exporting Motion Loads from Dynamic Simulation to the FEA environment.

Demonstration – Car Jack Handle



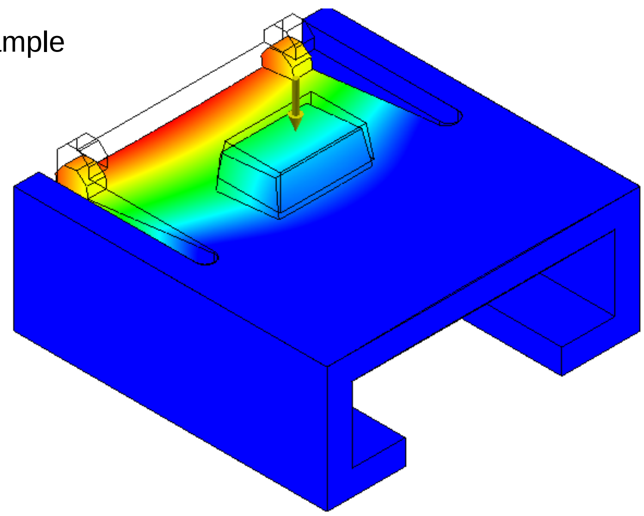
What implication do these twisting forces suggest for the Carabiner analysis from above?

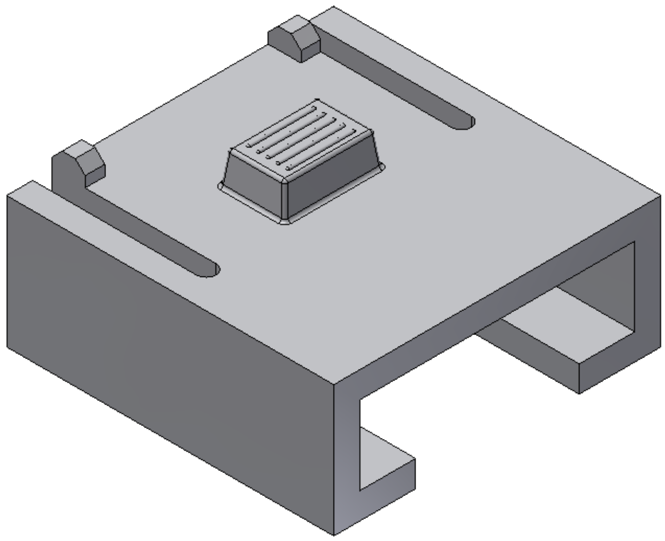
Learning Objective 4 – Learn how to do parametric dimension FEA test of part iterations in Inventor software.

Optimizing a Part.

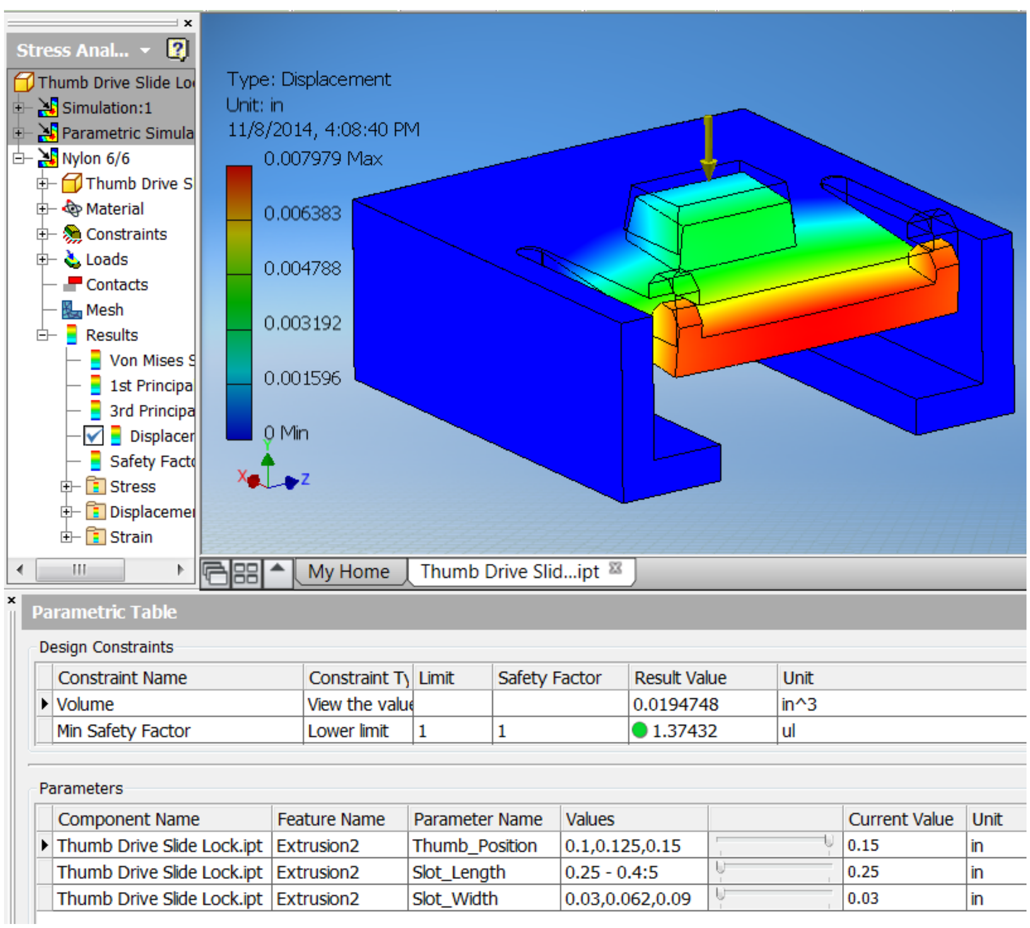
You can set up semi-automatic testing of iterations of your design. With cloud-based computing – I predict that this will become real-time and transparent in the future. With the setting of a few base parameters – expert systems will be continually analyzing our design in the background. The opportunity to get in on the ground flow is now – that is why you should be learning this stuff.

Demonstration – Thumb Slide Example





Notice that the part has features that are not relevant to our design analysis. Including these non-relevant features in the mesh would only increase the analysis time without any value.



Final Thoughts… …do we know what we think we know?

Conclusion: What have we learned?

- Recognize that units are important.
- Recognize when differences are important.
- Recognize the limits of the "ball field" in which we are operating.
  - Isotropic material properties.
  - Relatively rigid materials (not highly elastic, not brittle*).
  - In the linear portion of the stress/strain curve.
  - Relatively small deformations.
  - Slowly applied loads. (not impact loads)

- Recognize that Safety Factor is not a test of "breaking".
  - Cannot "extrapolate" information with SF<0 because this is a non-linear portion of the curve, outside of the limits of Inventor linear static stress analysis.

And my final thought.

Parse* these statements:

"Hand calculations serve as a "sanity check" for our digital analysis."

"Good correlation between the FEA and the hand calculation gives us a high degree of confidence in the FEA solution."

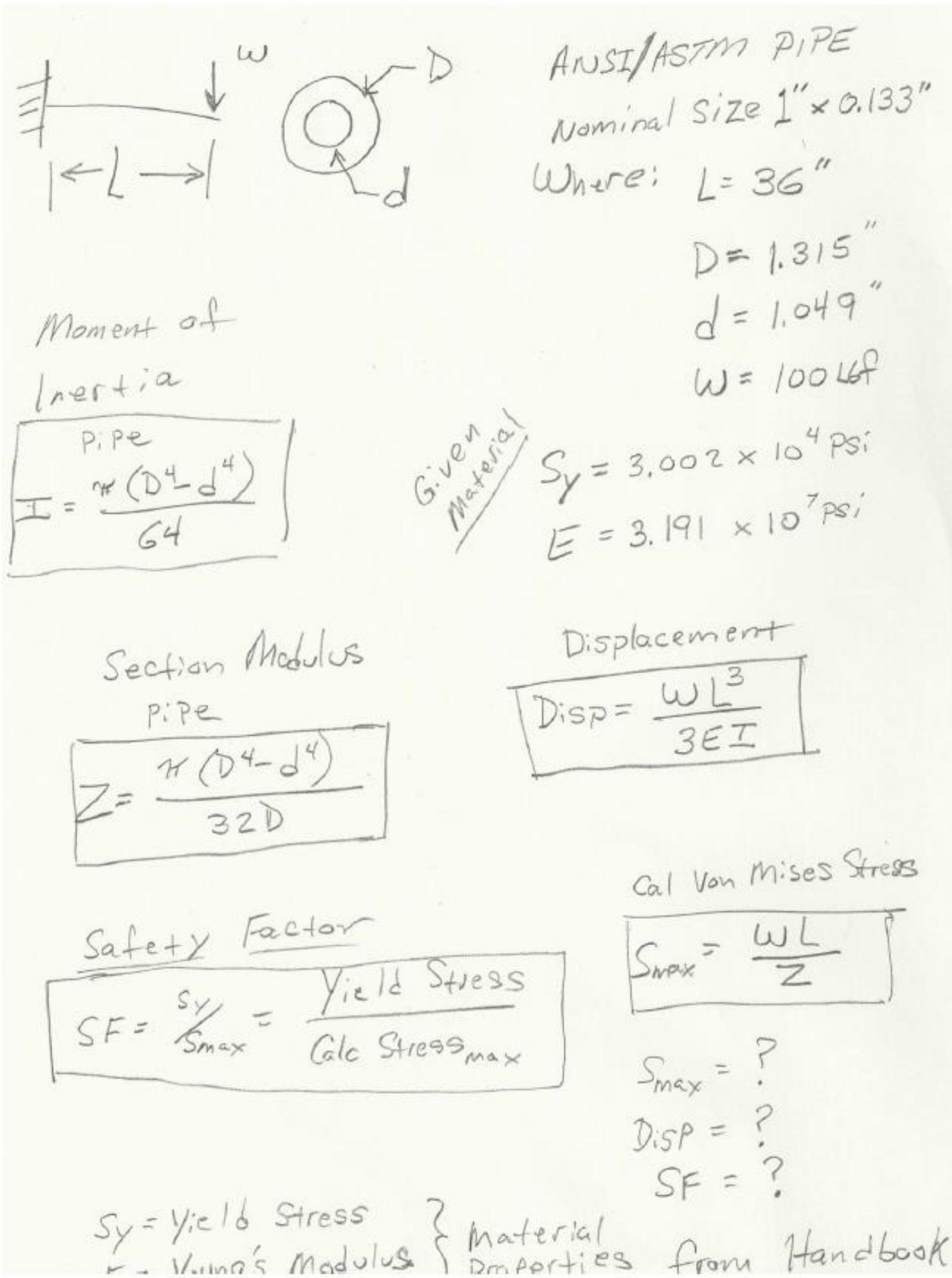
*Parse

verb

3. …to analyze (something, as a speech or behavior) to discover its implications or uncover a deeper meaning.

If you have some good hand calculation examples of other than trivial geometry – please forward them to jmather@pct.edu

Appendix: Example Hand Calculations



Hand calculation results:
S_{Max}=27,108 psi
Disp=0.558”
SF=1.1

References:

Inventor Quick Start – Perform Static Stress Analysis
http://akn.autodesk.com/maps/ENU/869946164#n=1391619412&c=987182764

Statics and Strength of Materials, 7th Ed. H.W. Morrow

Up and Running with Autodesk Inventor Professional 2014: Part 1 – Stress and Frame Analysis. Wasim Younis.