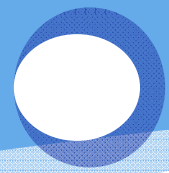


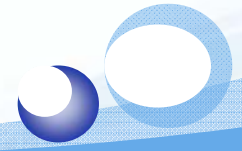


COSMOSWorks Practice

2008. 10. 1 컴퓨터 이용 설계 및 제작
Tae Ho Jang
Human-Centered CAD Lab.



What to Learn?



- ❖ COSMOSWorks in Solidworks
- ❖ Static Analysis
- ❖ Frequency Analysis
- ❖ Contact Analysis
- ❖ Buckling Analysis

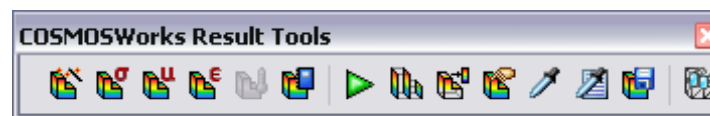
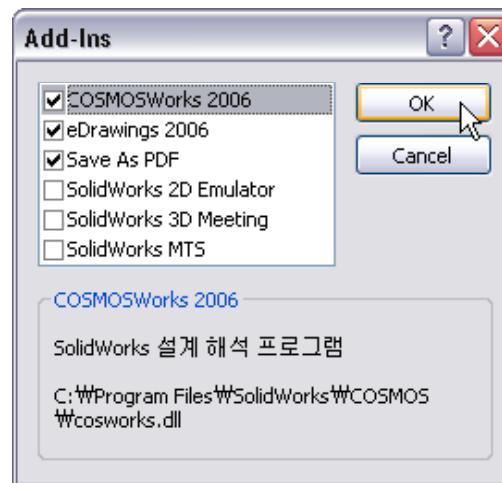
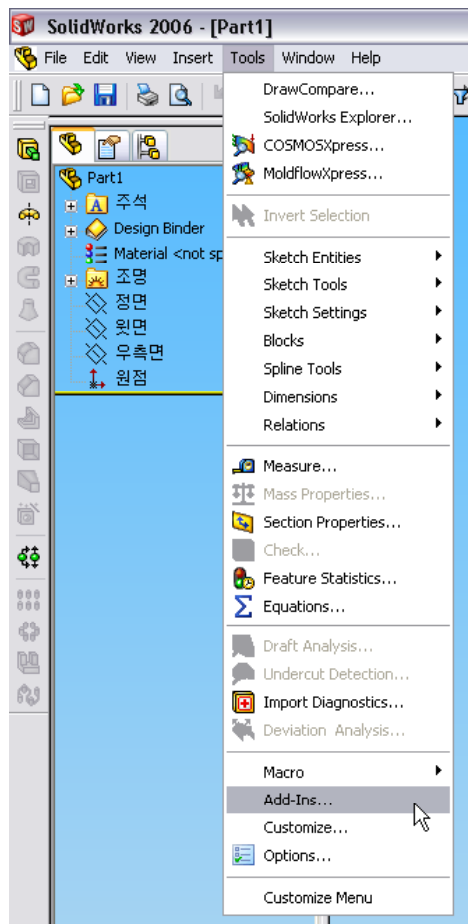


COSMOSWorks in Solidworks

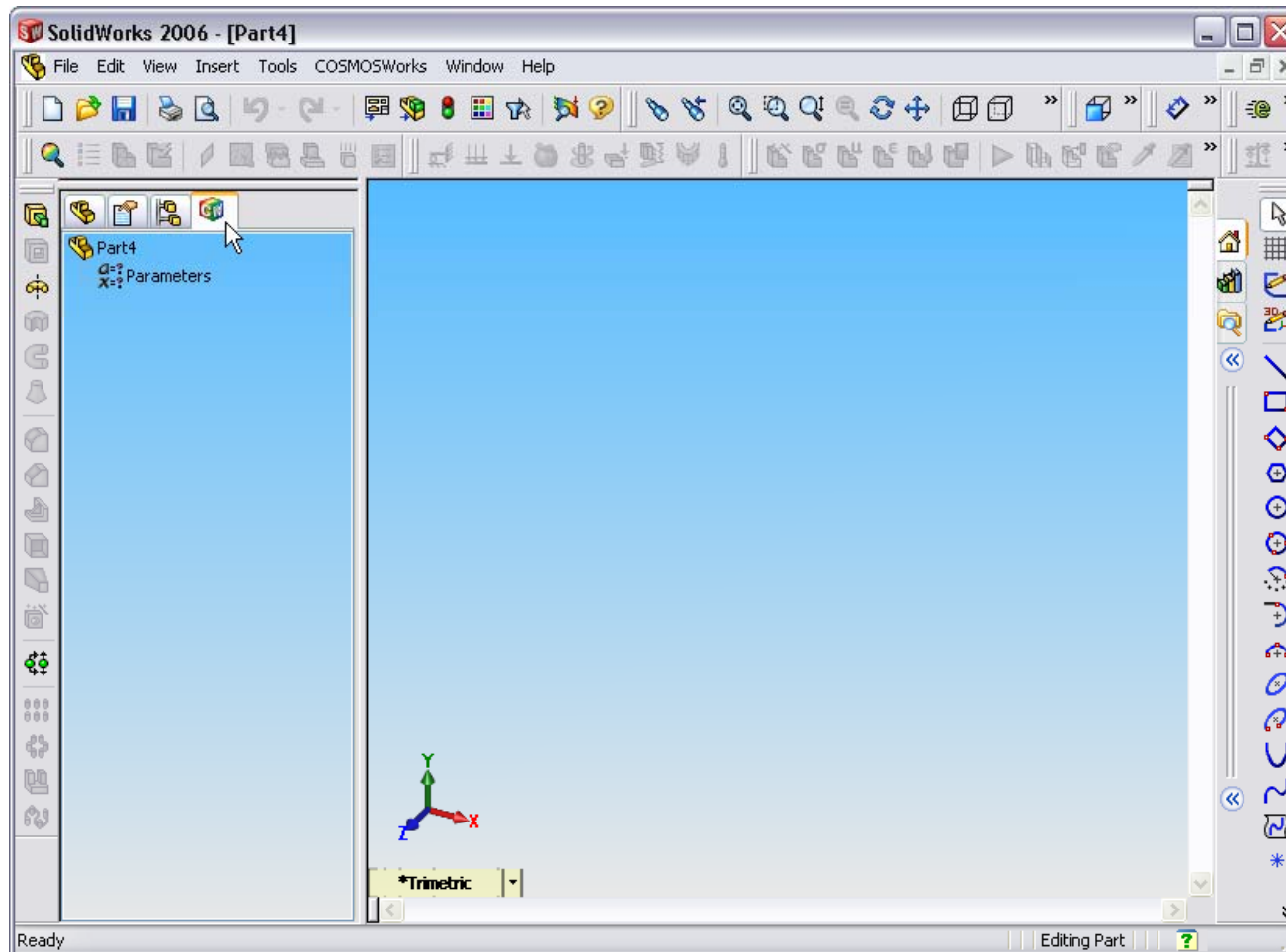


Add-in / Toolbar

Tool >> Add-in >> COSMOSWorks



COSMOSWorks Manager

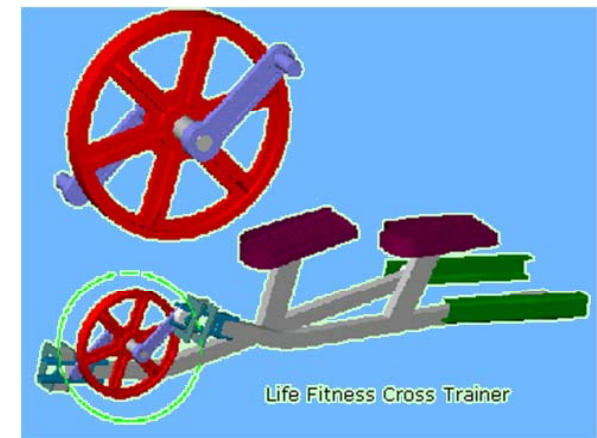




Static Analysis

Procedure

- ❖ Create a static analysis study
- ❖ Assign materials to the various components of the assembly
- ❖ Insert restraints and loads
- ❖ Mesh the assembly
- ❖ Run static analysis
- ❖ Visualize the static analysis result

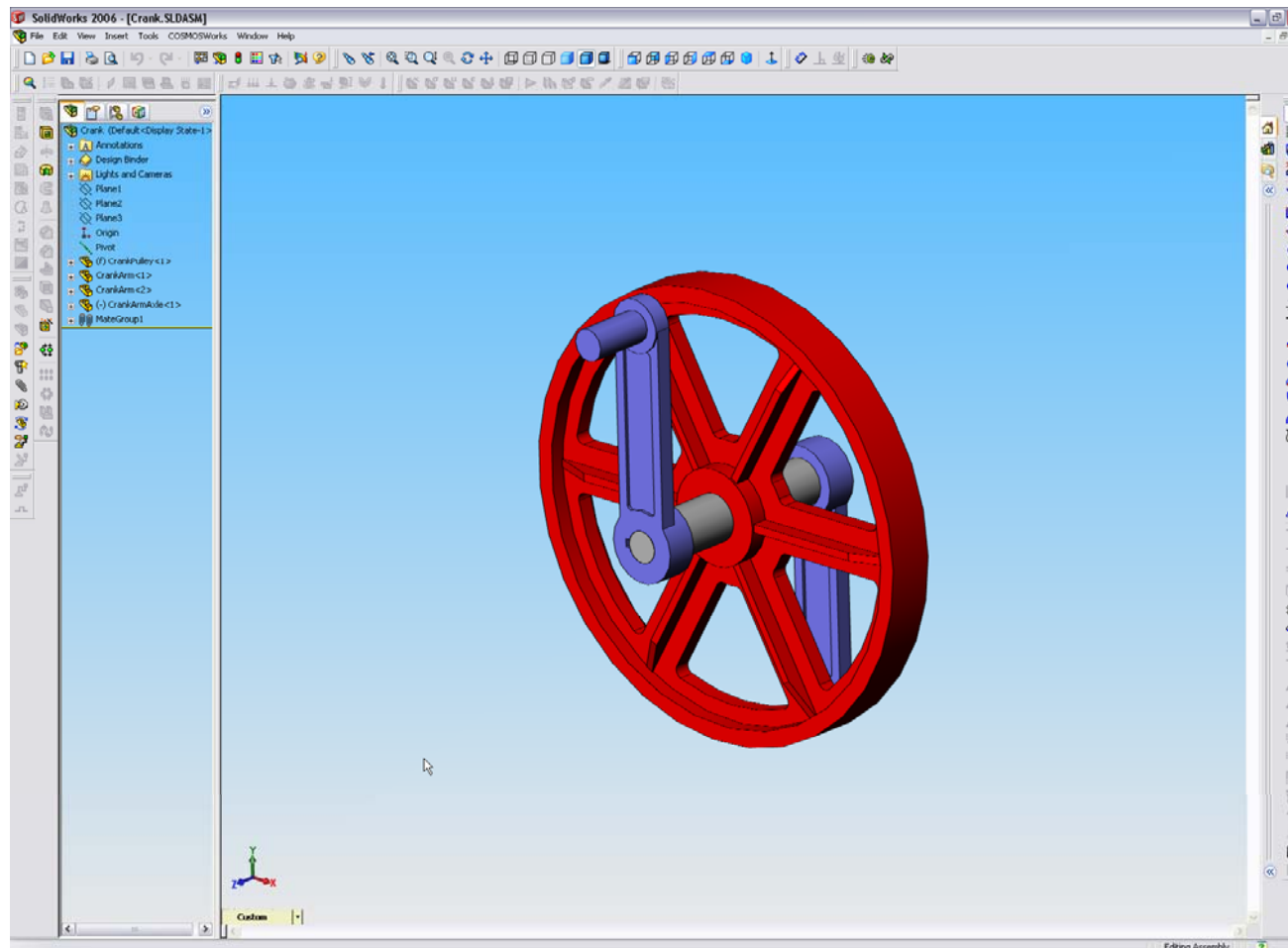




Open

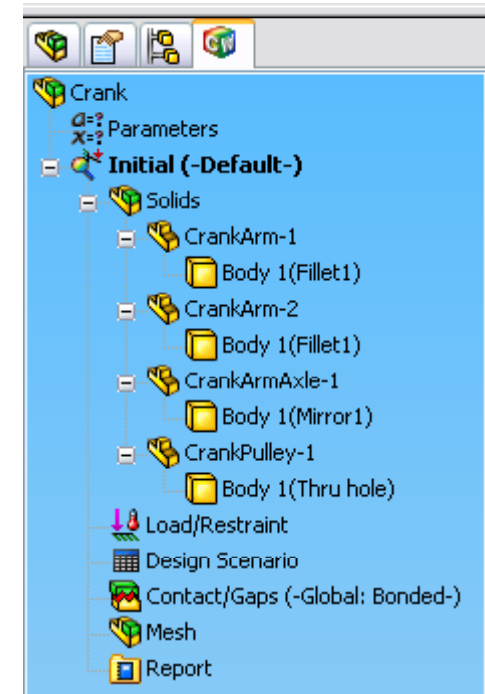
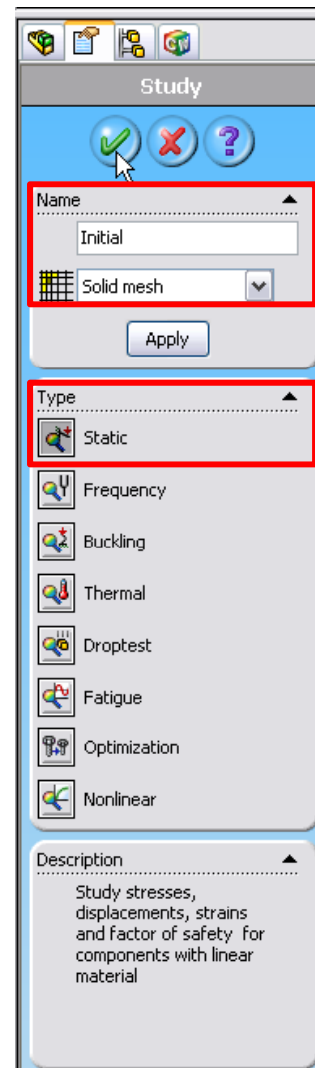
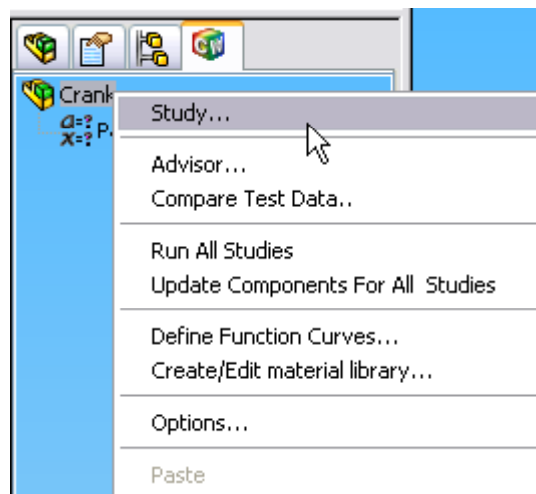
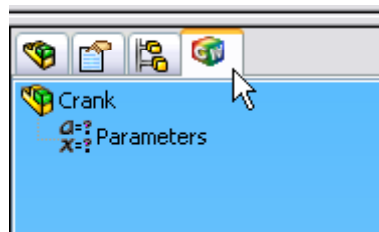


Crank.SLDASM



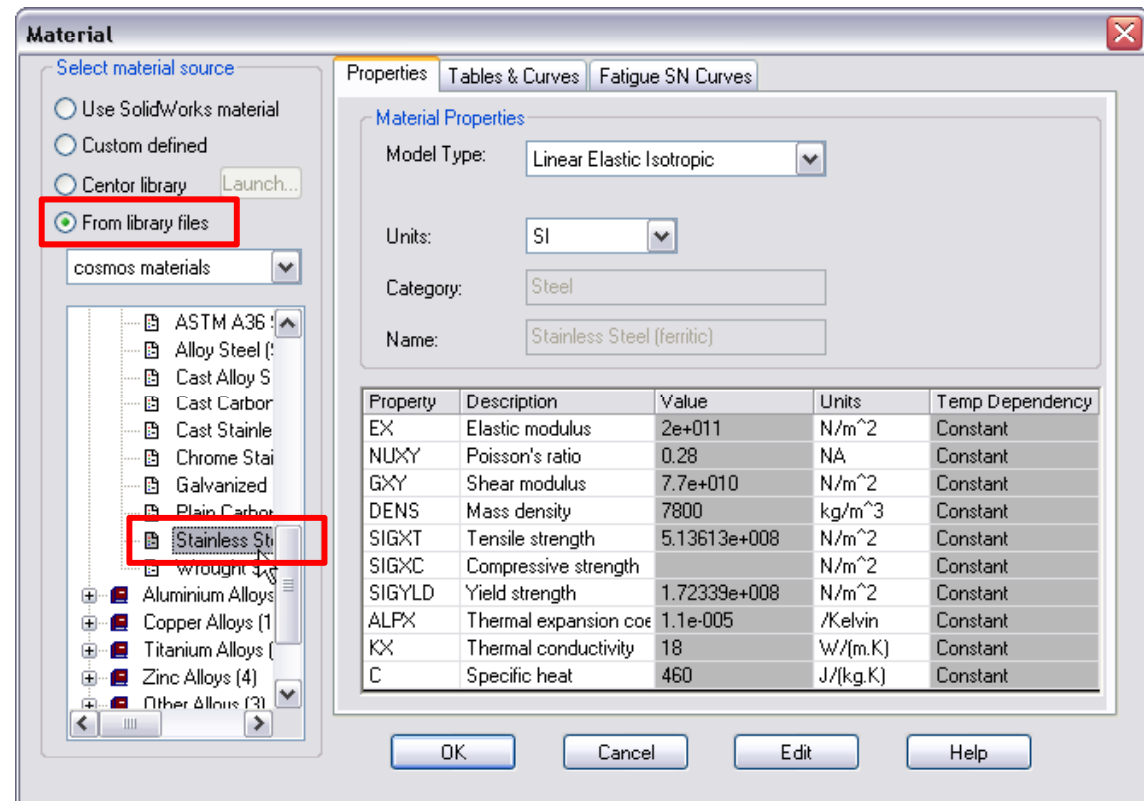
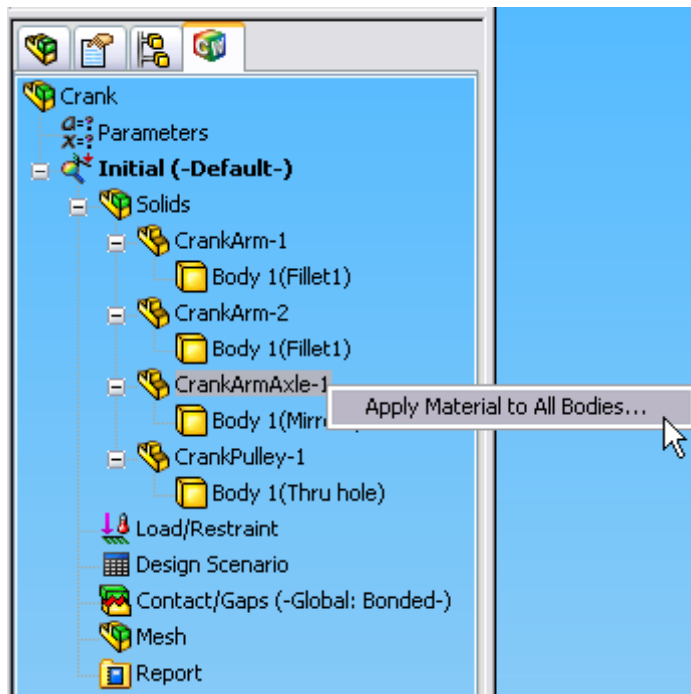
Create a Study

Study >> Solid Mesh & Static



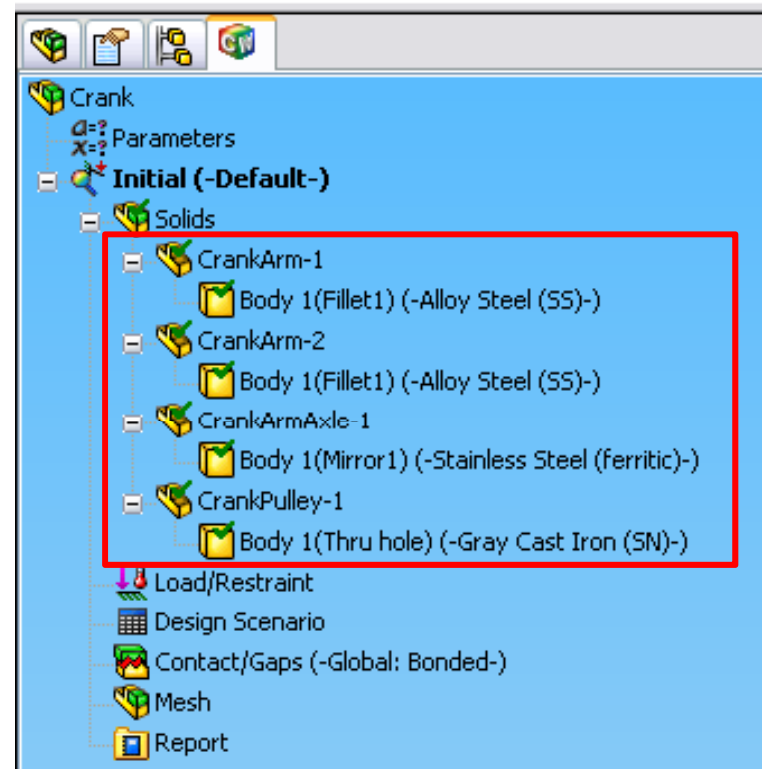
Apply Materials

CrankArmAxle-1 >> Apply Material to All Bodies >> Steel >> Stainless Steel



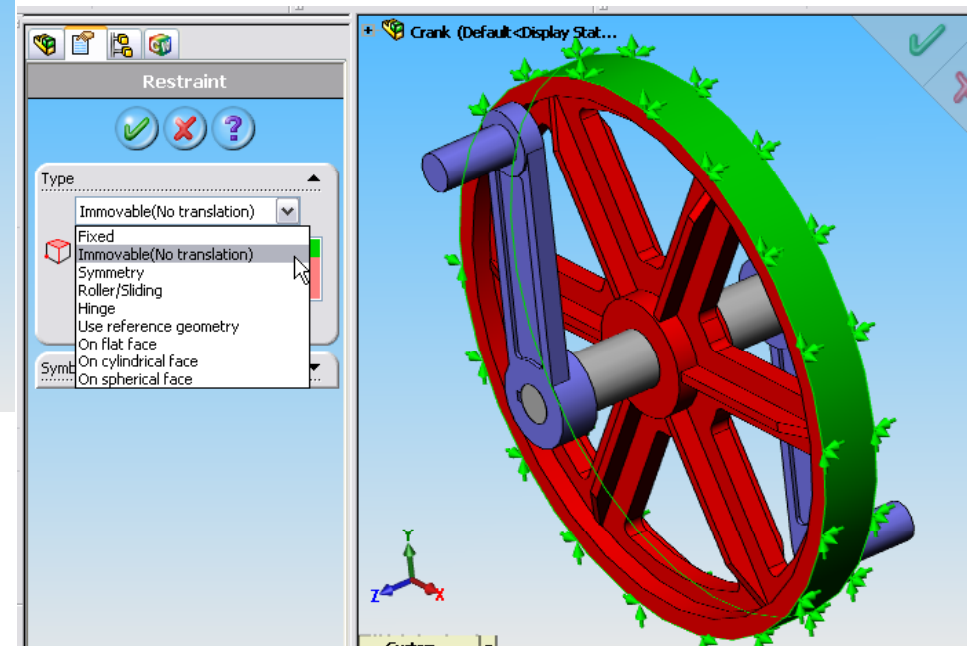
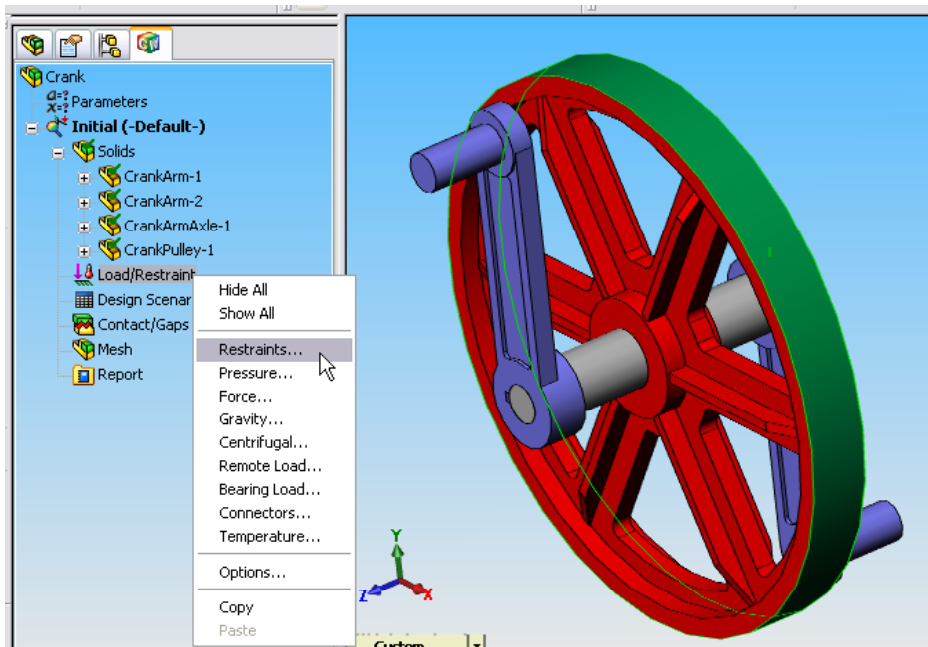
Apply Materials

- ❖ CrankPully-1
 - Gray Cast Iron
- ❖ CrankArm-1
 - Alloy Steel
- ❖ CrankArm-2
 - Alloy Steel



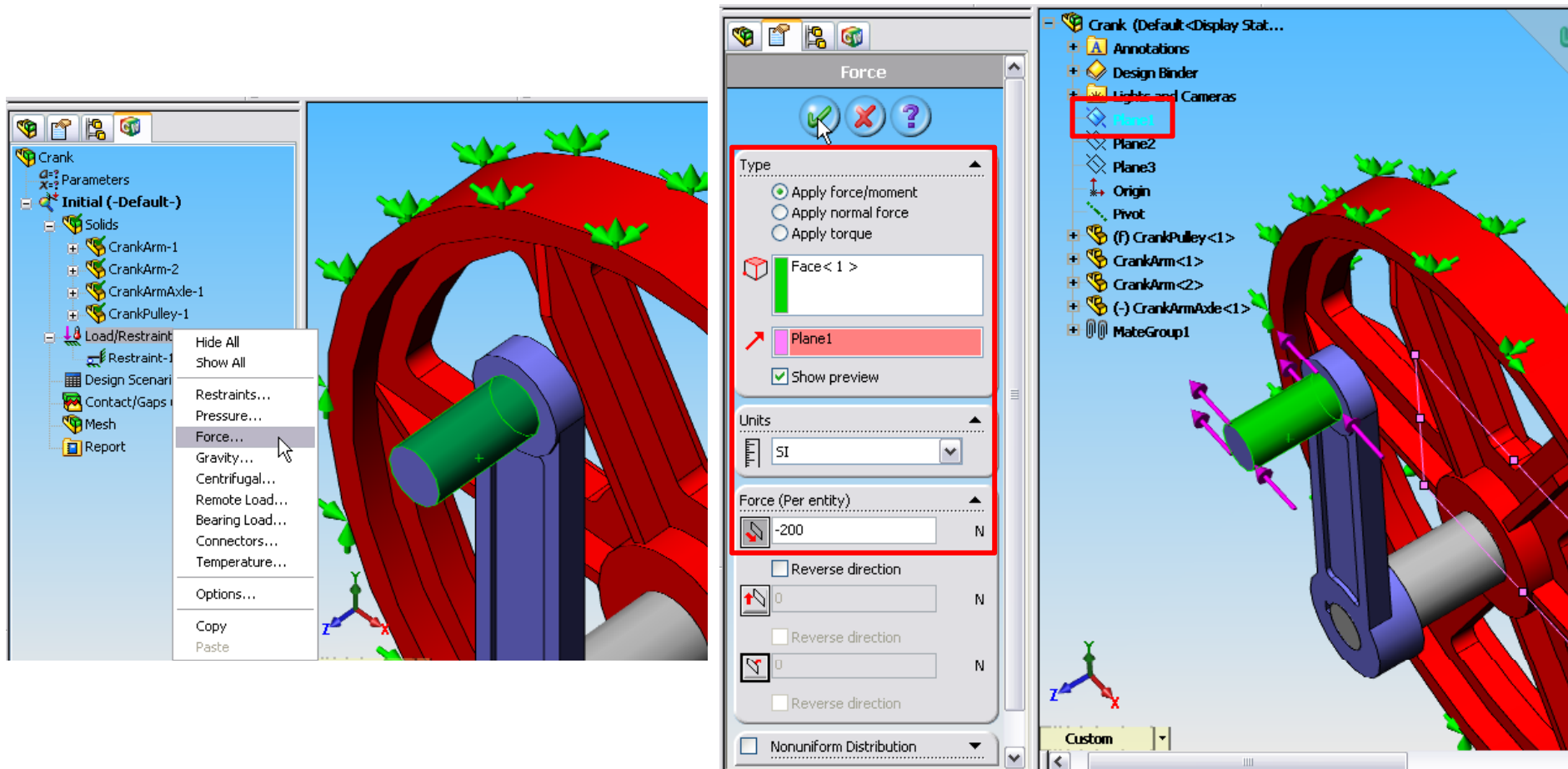
Apply Restraint - Immovable

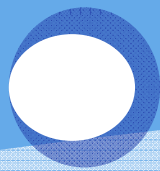
Load/Restraints >> Restraints >> Immovable



Apply Force

Load/Restraints >> Force >> Along direction >> - 200N

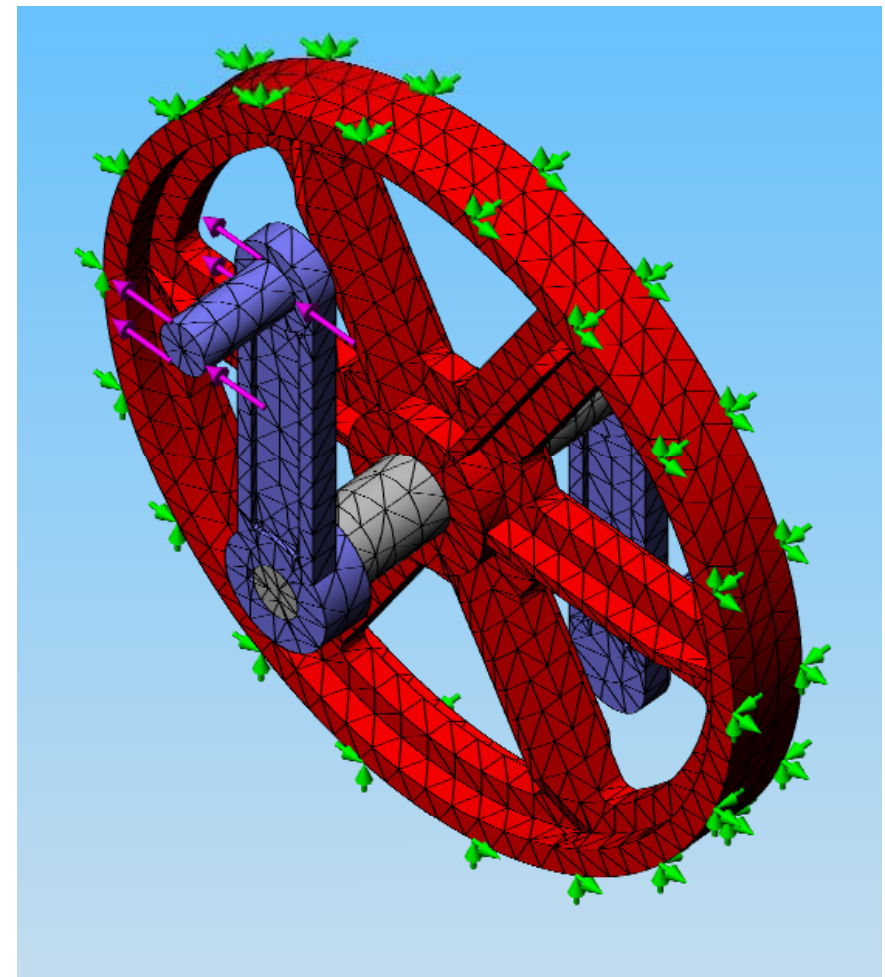
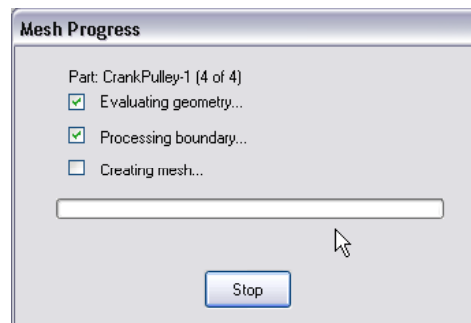
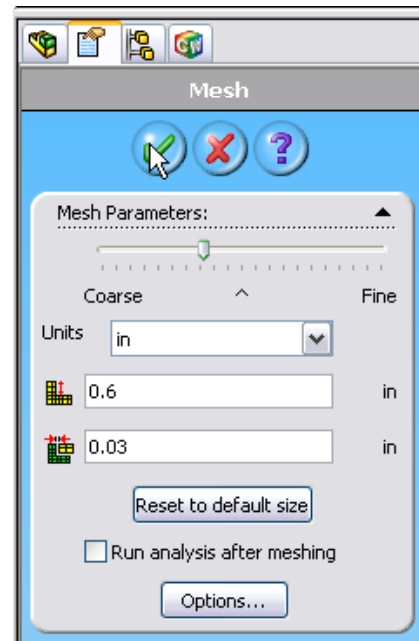
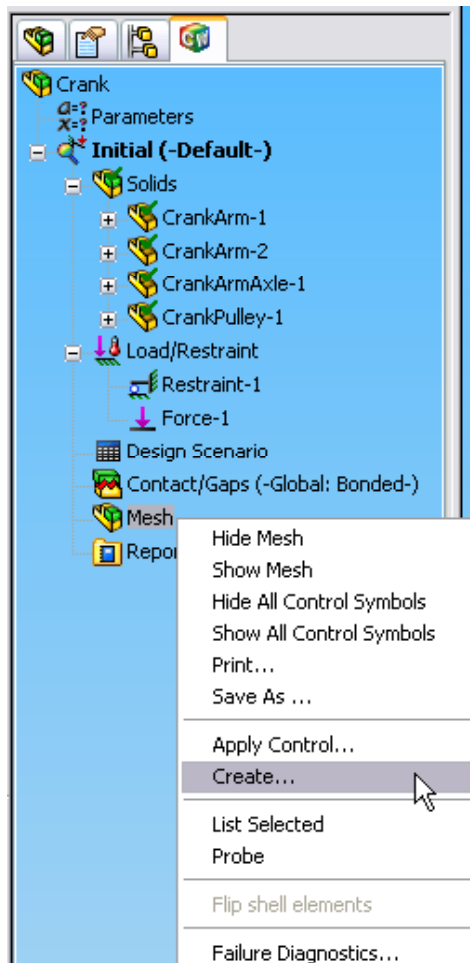




Create Mesh

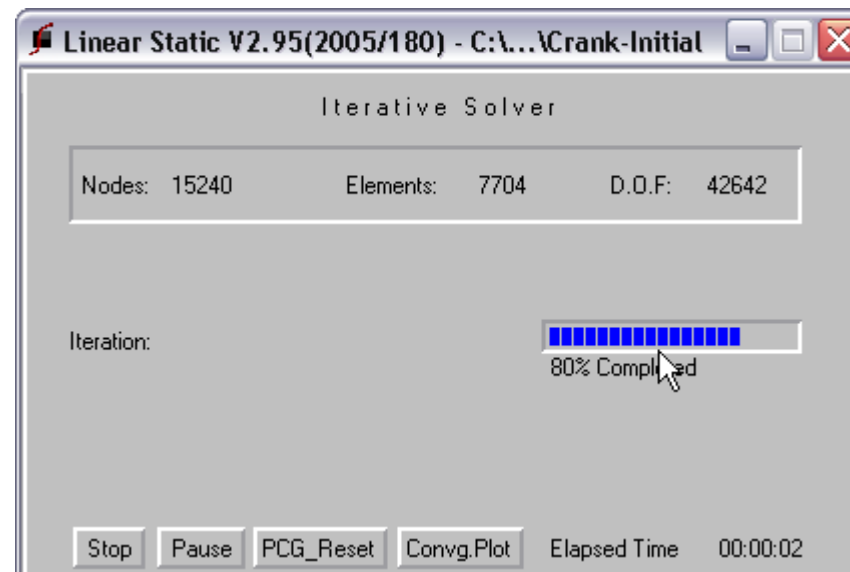


Mesh >> Create >> Global Size 0.6 in



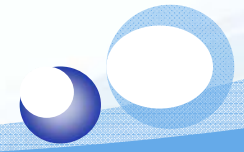
Run Static Analysis

Study >> Run





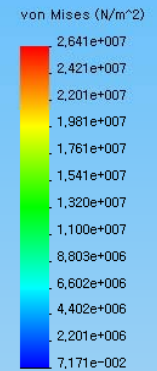
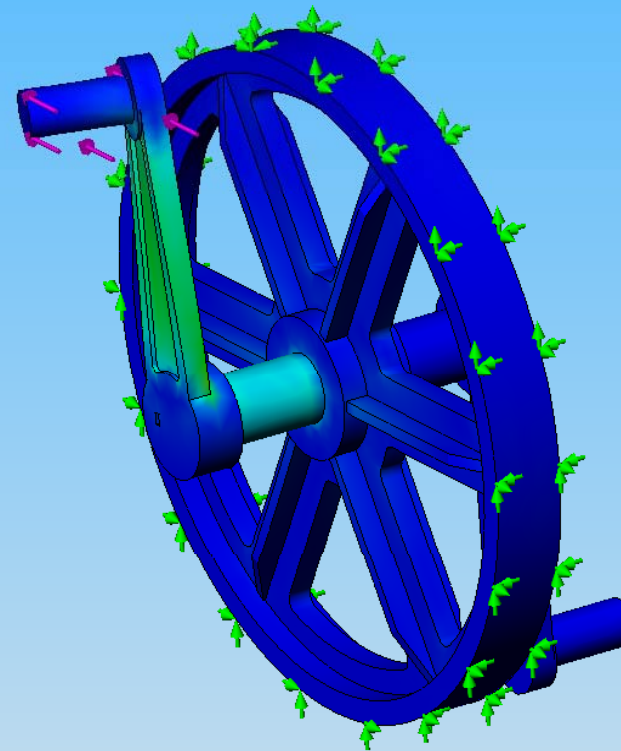
Stress Plot



Crank

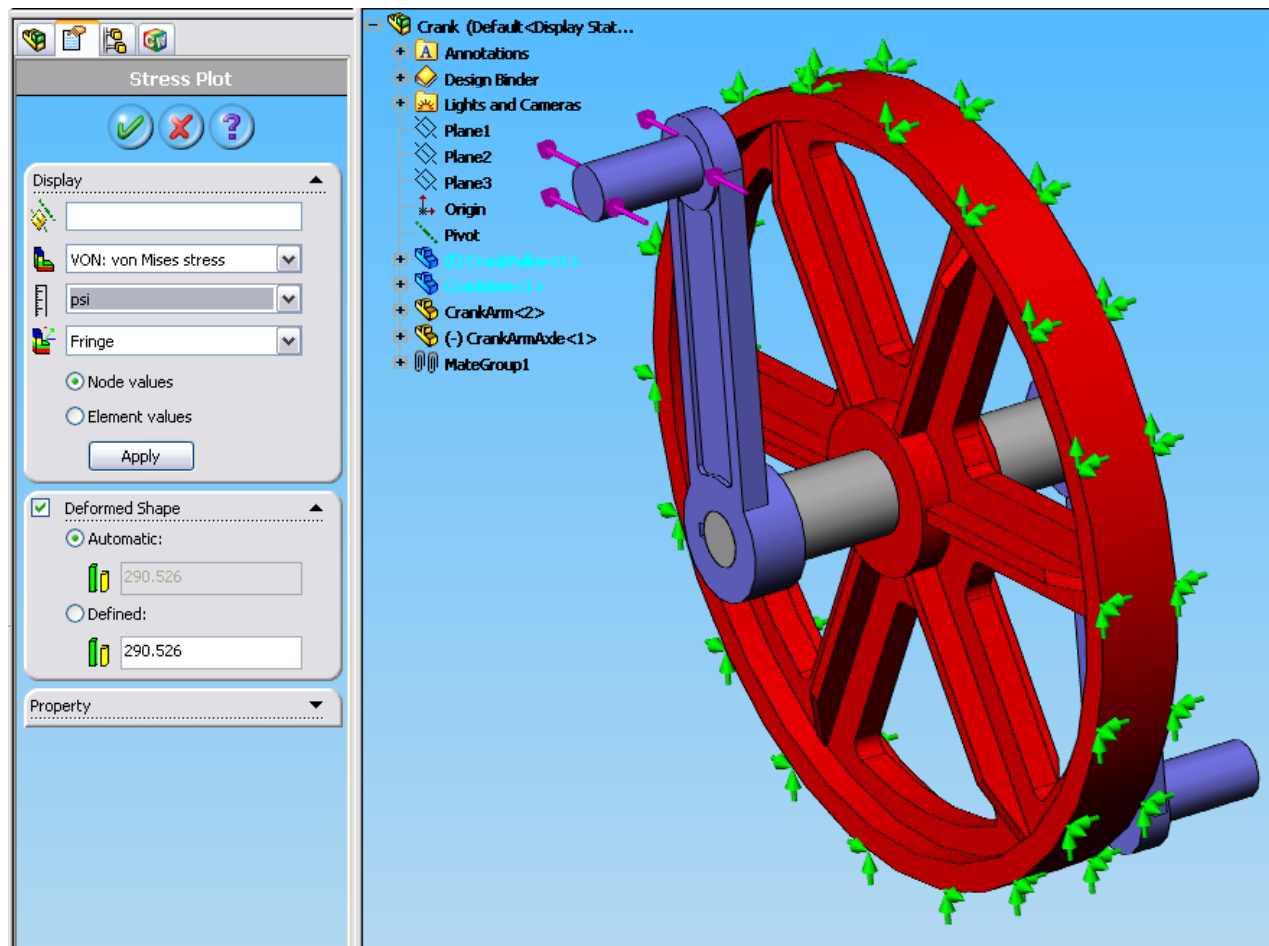
- Parameters
- Initial (-Default-)**
 - Solids
 - CrankArm-1
 - CrankArm-2
 - CrankArmAxle-1
 - CrankPulley-1
 - Load/Restraint
 - Restraint-1
 - Force-1
 - Design Scenario
 - Contact/Gaps (-Global: Bonded-)
 - Mesh
 - Report
 - Stress
 - Plot1 (-vonMises-)
 - Displacement
 - Plot1 (-Res disp-)
 - Strain
 - Plot1 (-Equivalent-)
 - Deformation
 - Plot1
 - Design Check
 - Plot1 (-FOS-)

Model name: Crank
Study name: Initial
Plot type: Static nodal stress Plot1
Deformation scale: 290,526



Change Unit - psi

Plot1 >> Edit Definition >> psi

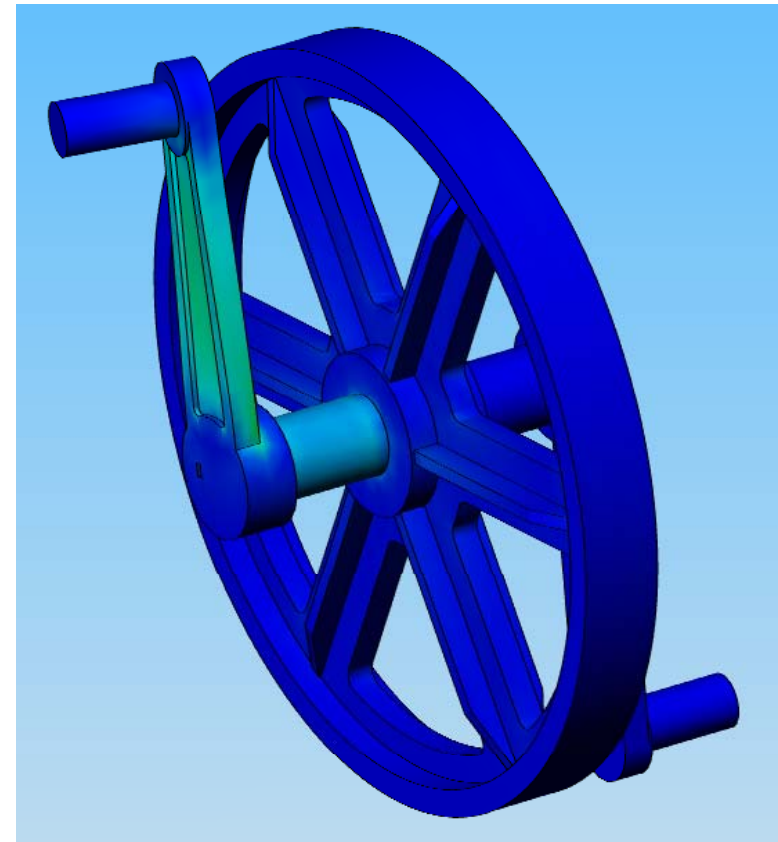
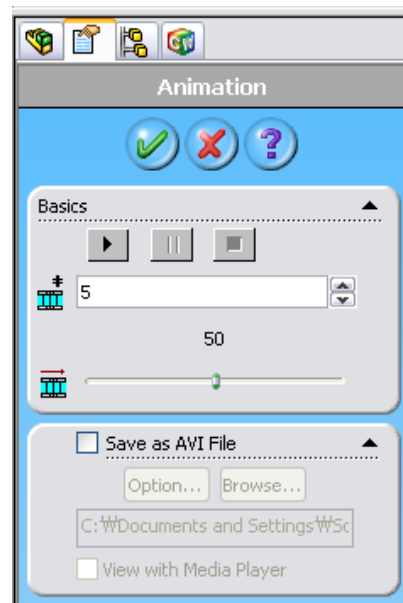
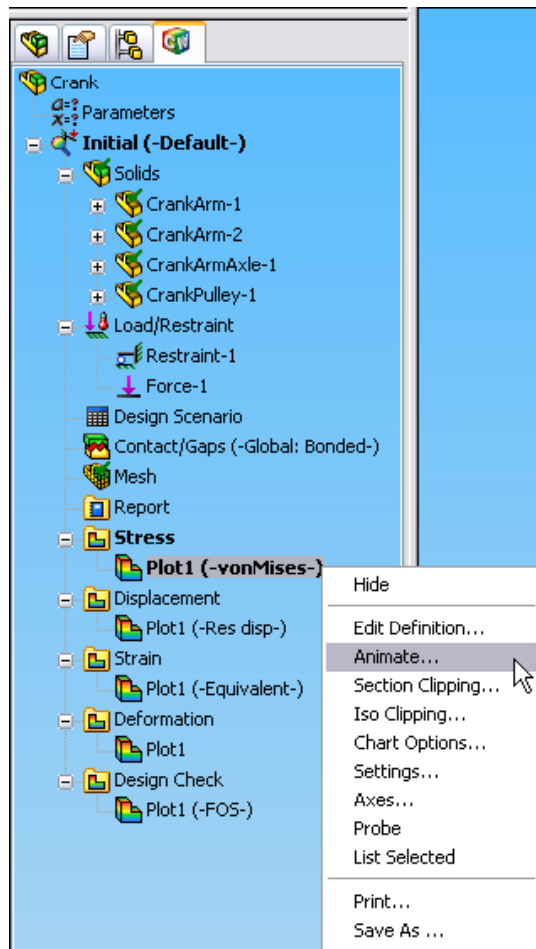




Animate

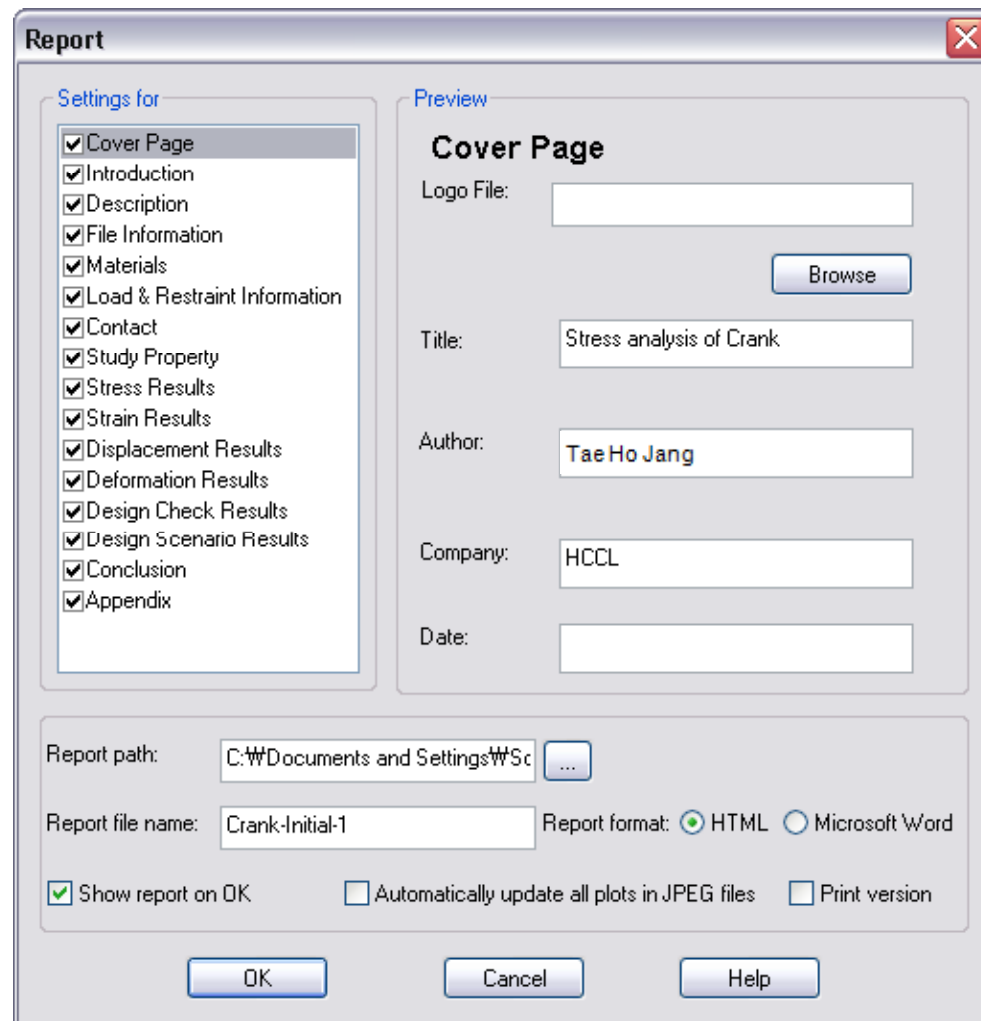
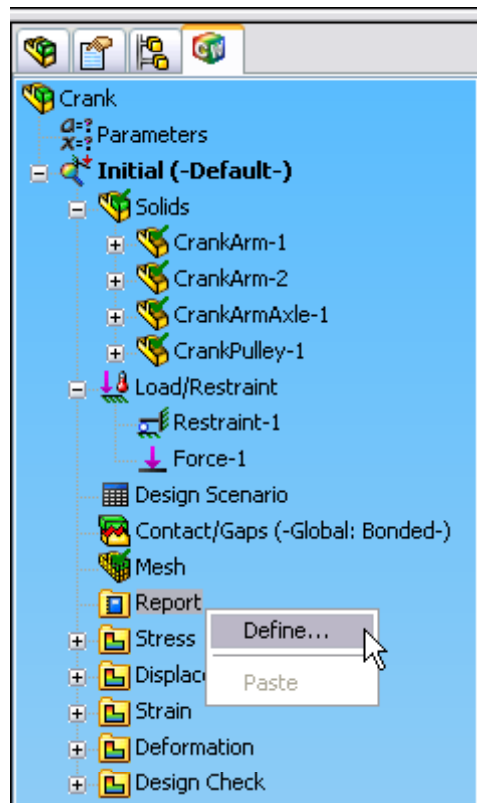


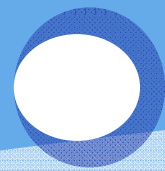
Plot1 >> Animate



Generating Report

Report >> Define





Report Example



Stress analysis of Crank

Author: Tae-guen Son

Company: HCCL

- [1. Introduction](#)
- [2. File Information](#)
- [3. Materials](#)
- [4. Load & Restraint Information](#)
- [5. Study Property](#)
- [6. Contact](#)
- [7. Stress Results](#)
- [8. Strain Results](#)
- [9. Displacement Results](#)
- [10. Deformation Results](#)
- [11. Design Scenario Results](#)
- [12. Appendix](#)

7. Stress Results

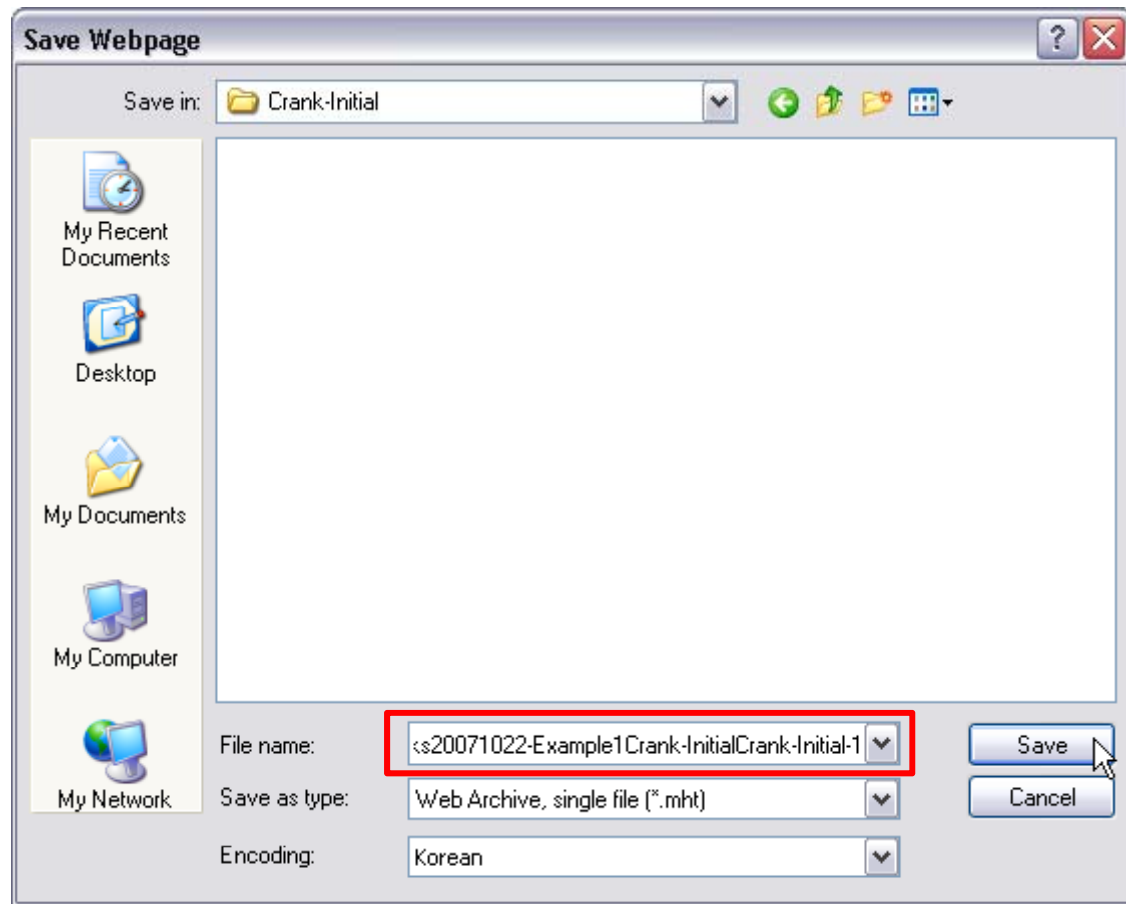
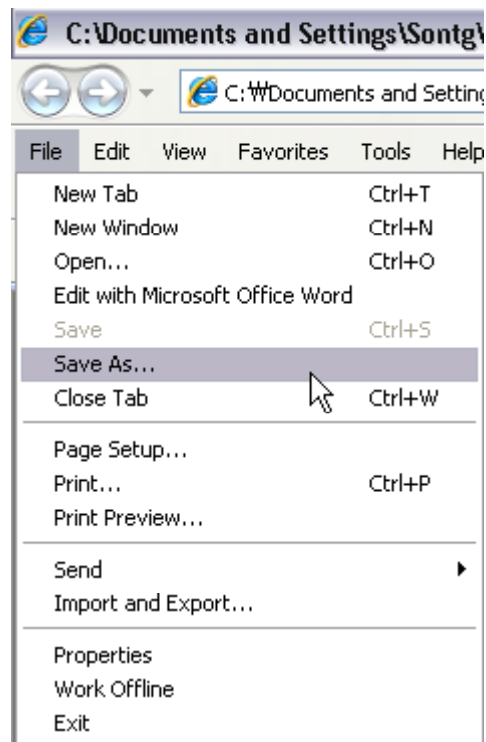
Name	Type	Min	Location	Max	Location
Plot1	VON: von Mises stress	0,0717146 N/m ² Node: 3327	(0,265 in, -4,22274 in, -3,684 in)	2,64092e+007 N/m ² Node: 1911	(0,641187 in, 0,905819 in, 3,434 in)

3. Materials

No.	Part Name	Material	Mass	Volume
1	CrankArm-1	Alloy Steel (SS)	1,17195 kg	0,000152201 m ³
2	CrankArm-2	Alloy Steel (SS)	1,17195 kg	0,000152201 m ³
3	CrankArmAxle-1	Stainless Steel (ferritic)	1,52621 kg	0,000195668 m ³
4	CrankPulley-1	Gray Cast Iron (SN)	9,25209 kg	0,00128501 m ³

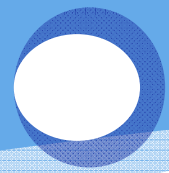
Save Report as Web Page

File >> Save As >> .mht



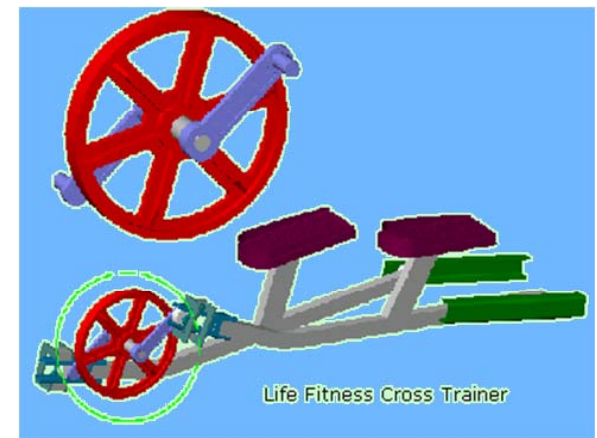


Frequency Analysis



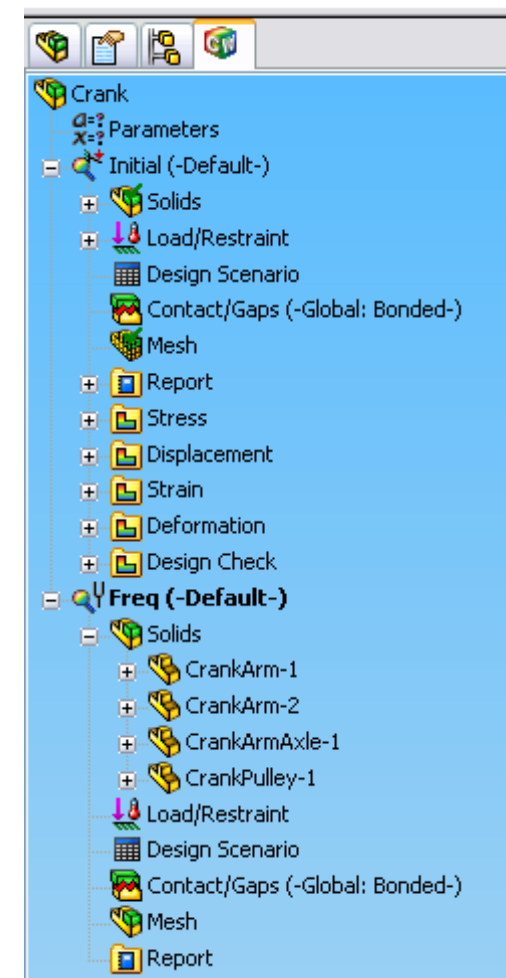
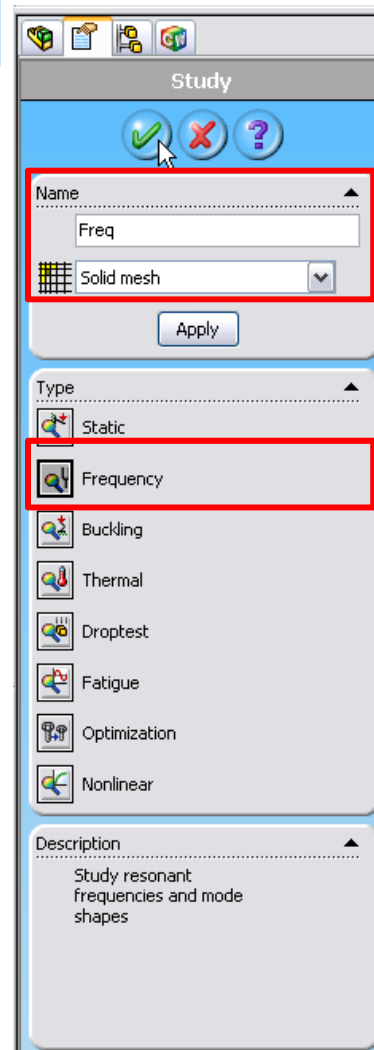
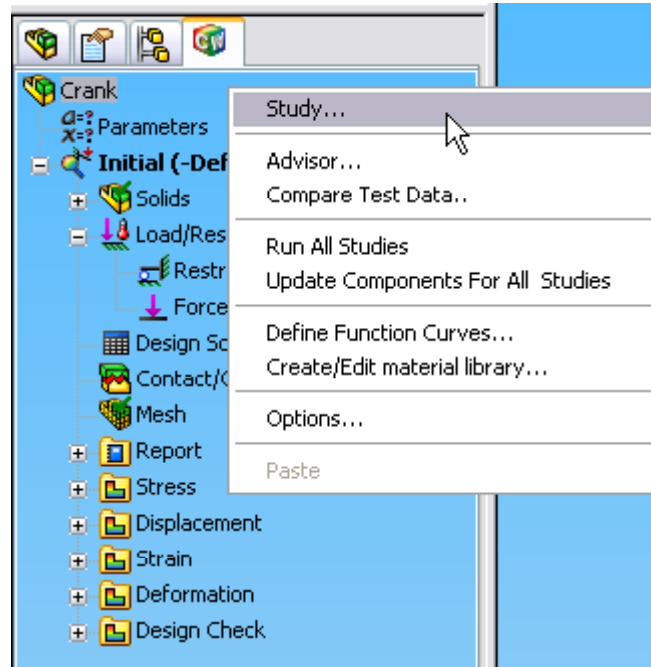
Procedure

- ❖ Create a frequency analysis study
- ❖ Run frequency analysis
- ❖ Visualize frequency analysis study



Add New Study

Study >> Solid Mesh & Frequency

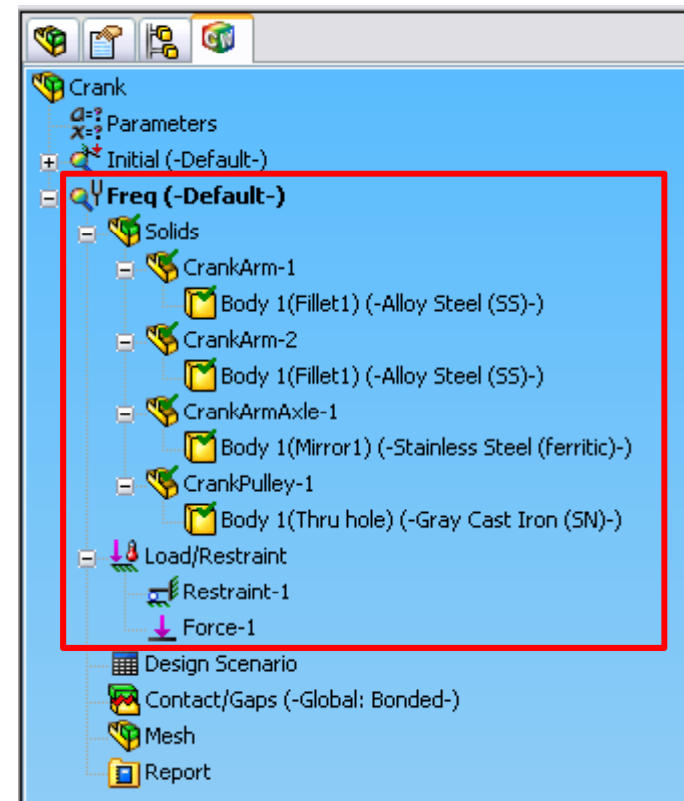
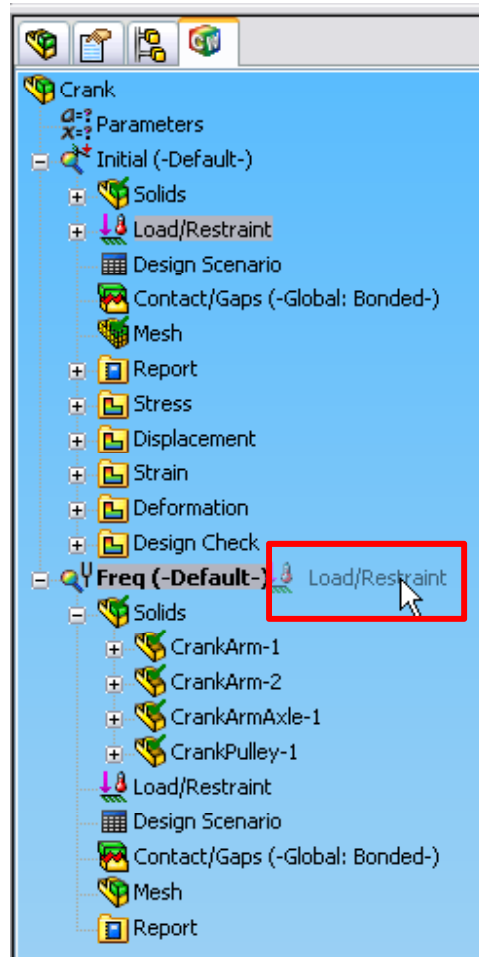
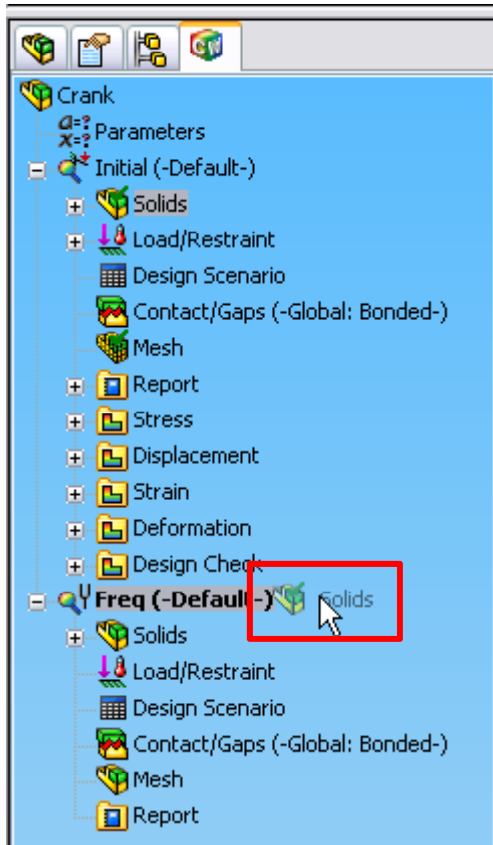




Drag & Drop

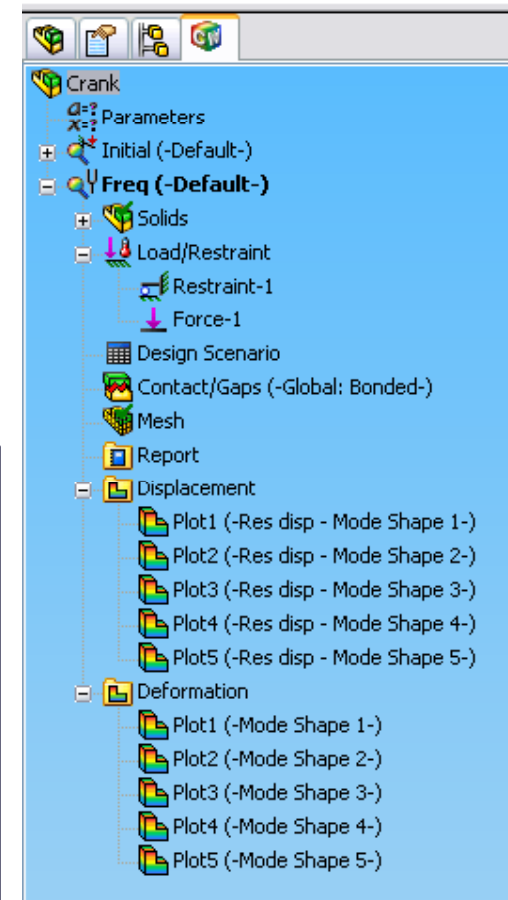
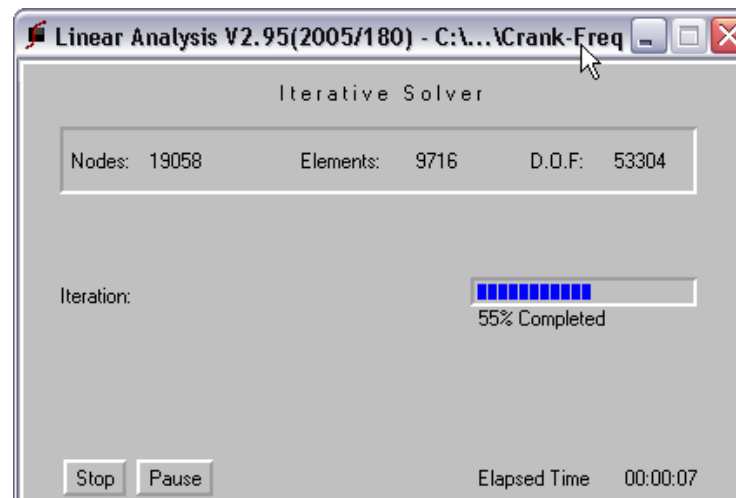
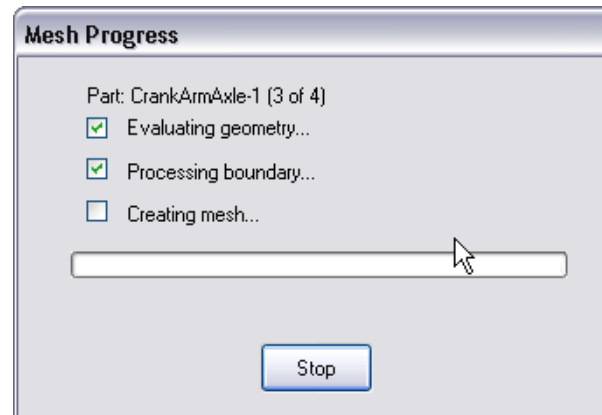
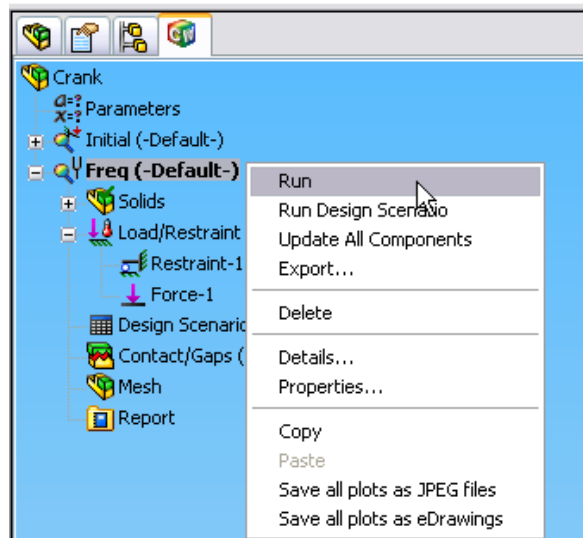


Apply Materials, Restraints and Force



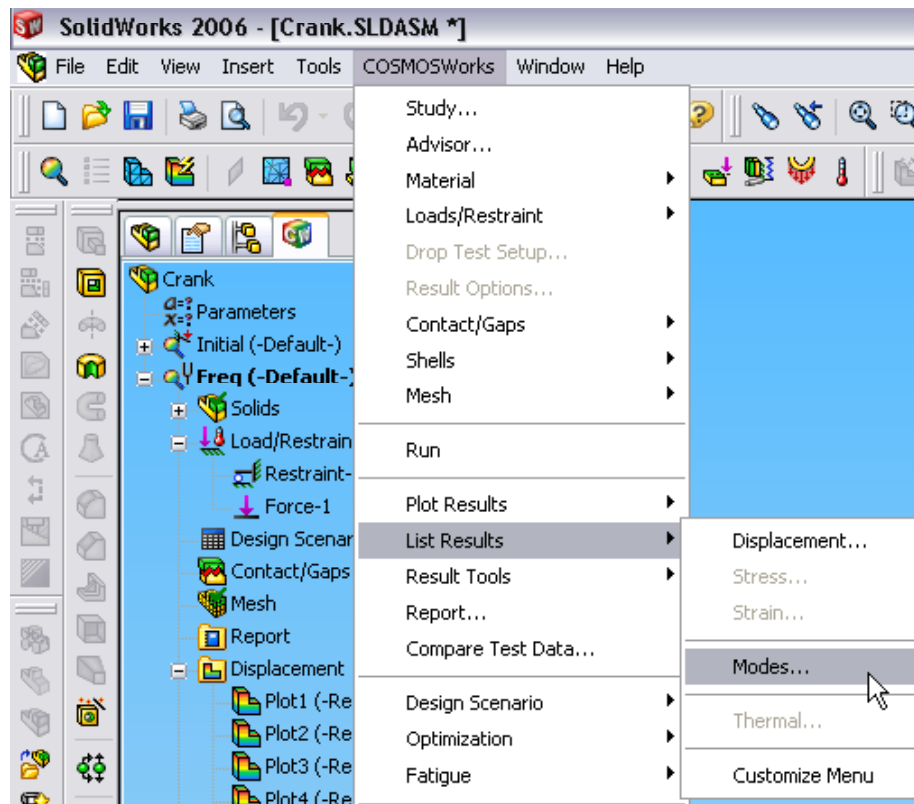
Run Frequency Analysis

Freq >> Run



List Result - Mode Shape

COSMOSWorks >> List Results >> Modes



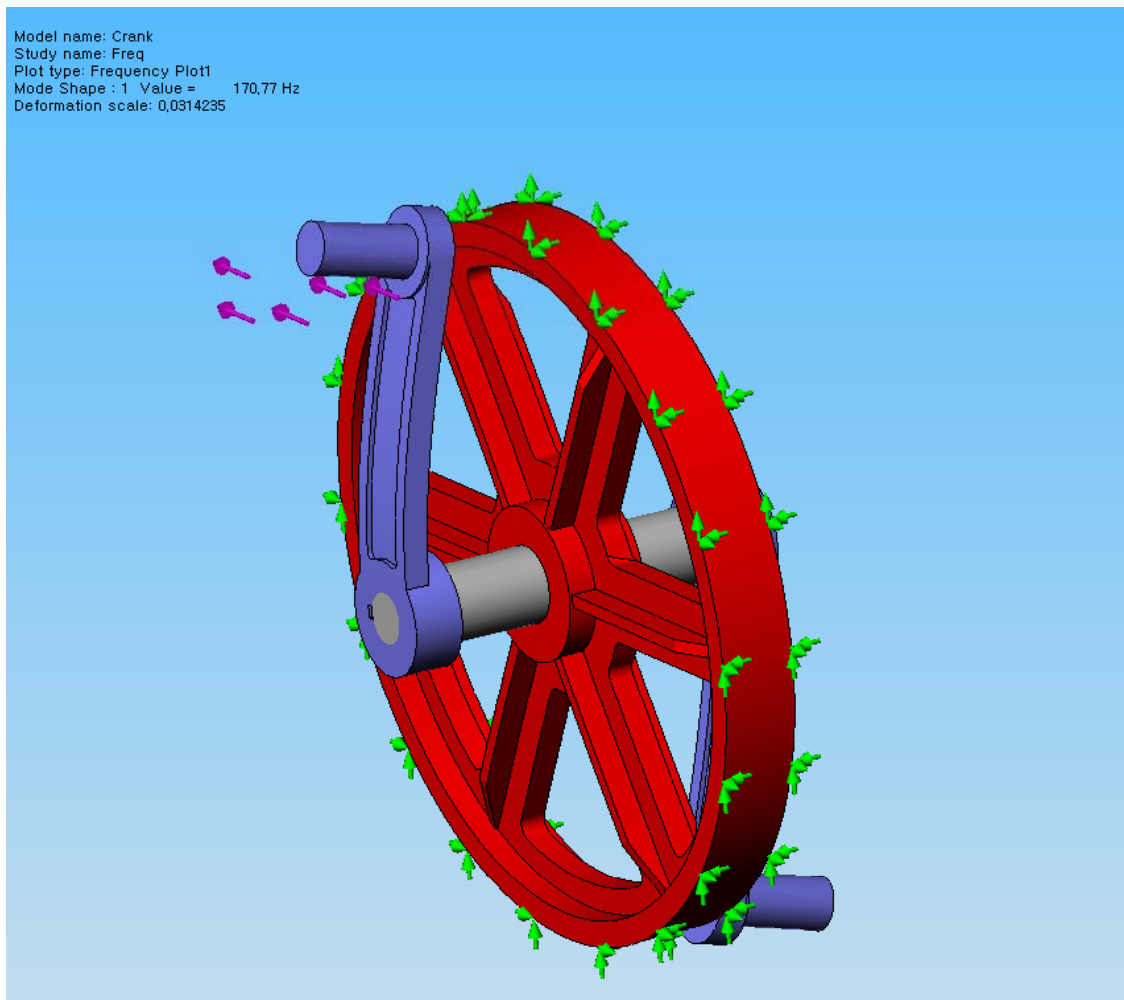
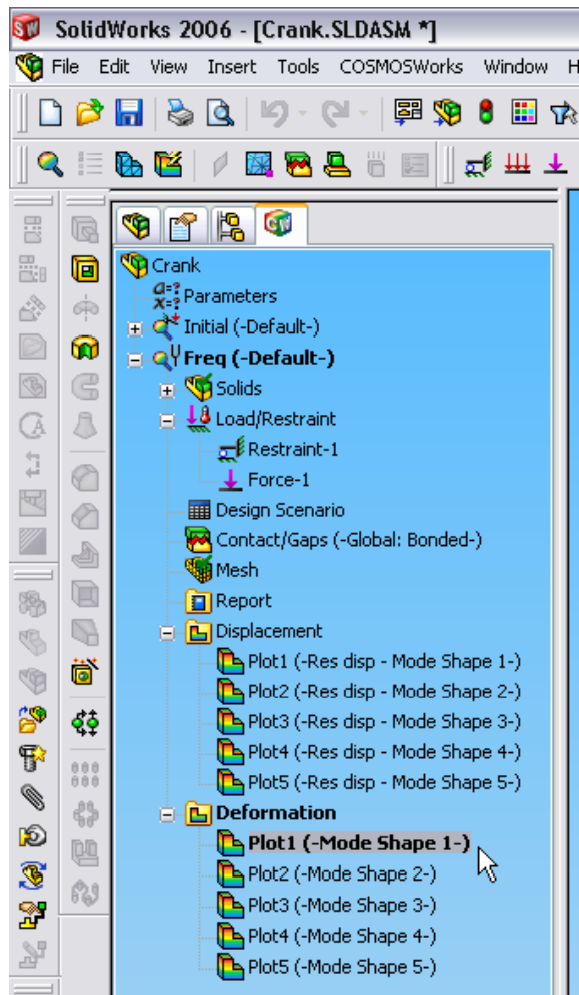
List Modes

Study name: Freq

Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	1073	170.77	0.0058559
2	1136.6	180.89	0.0055283
3	2143.9	341.22	0.0029307
4	2681.1	426.71	0.0023435
5	2836.1	451.37	0.0022155

Close Save Help

Deformation Plot



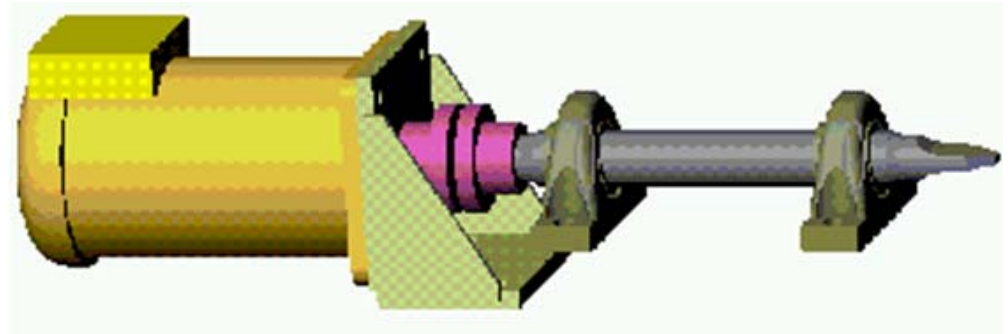


Contact Analysis



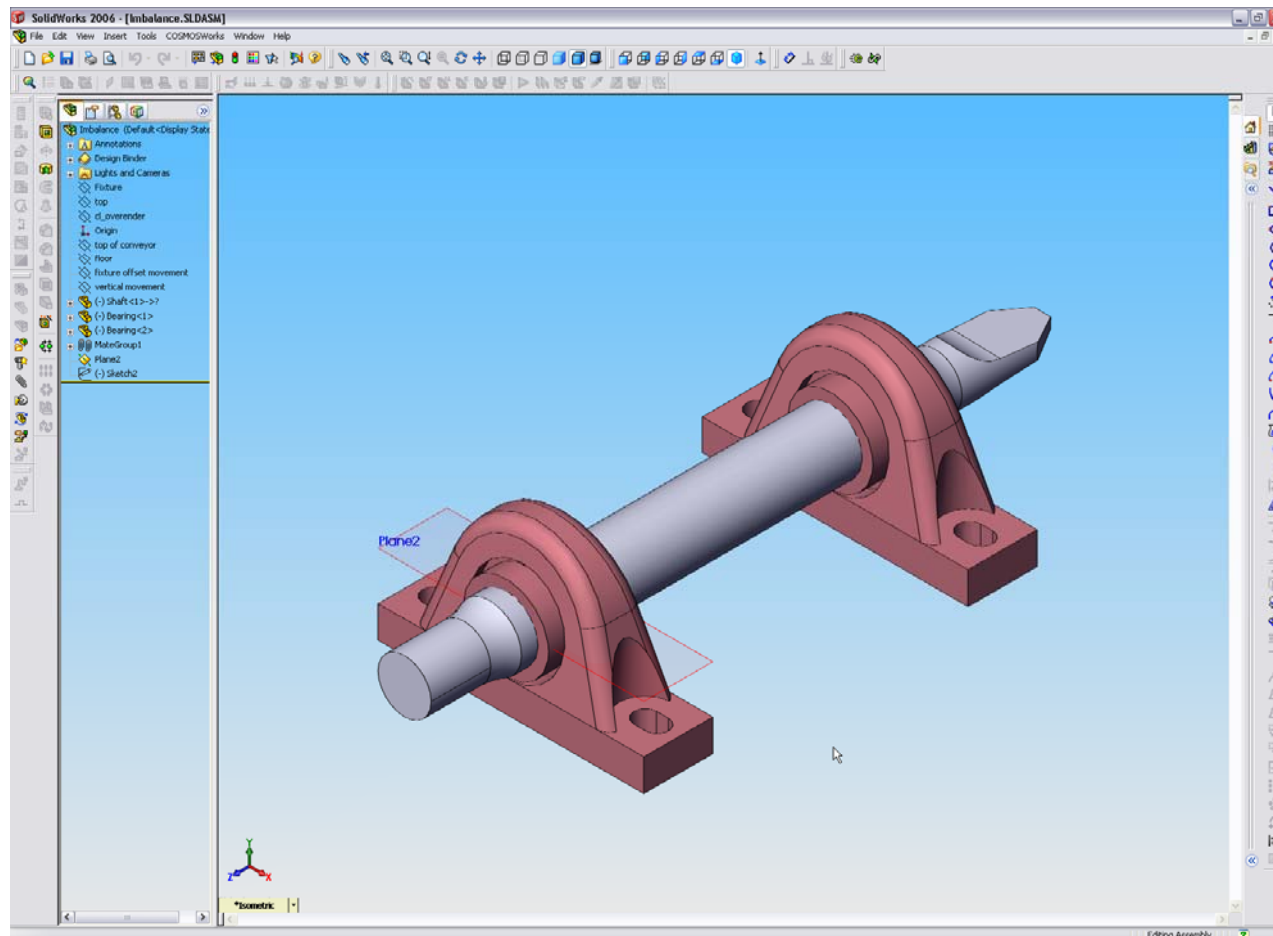
Procedure

- ❖ Set a global Contact/Gaps options
- ❖ Mesh an assembly
- ❖ Run static analysis
- ❖ Visualize the stress results



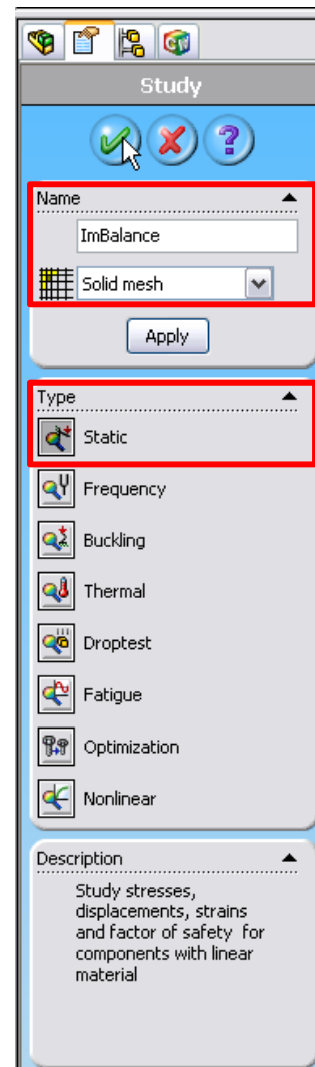
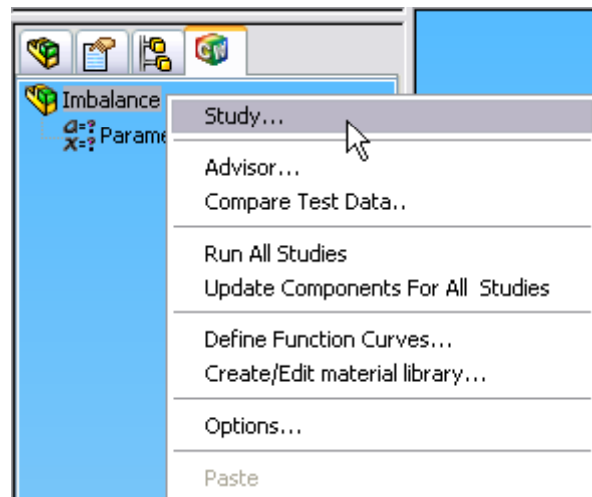
Open

Imbalance.SLDASM



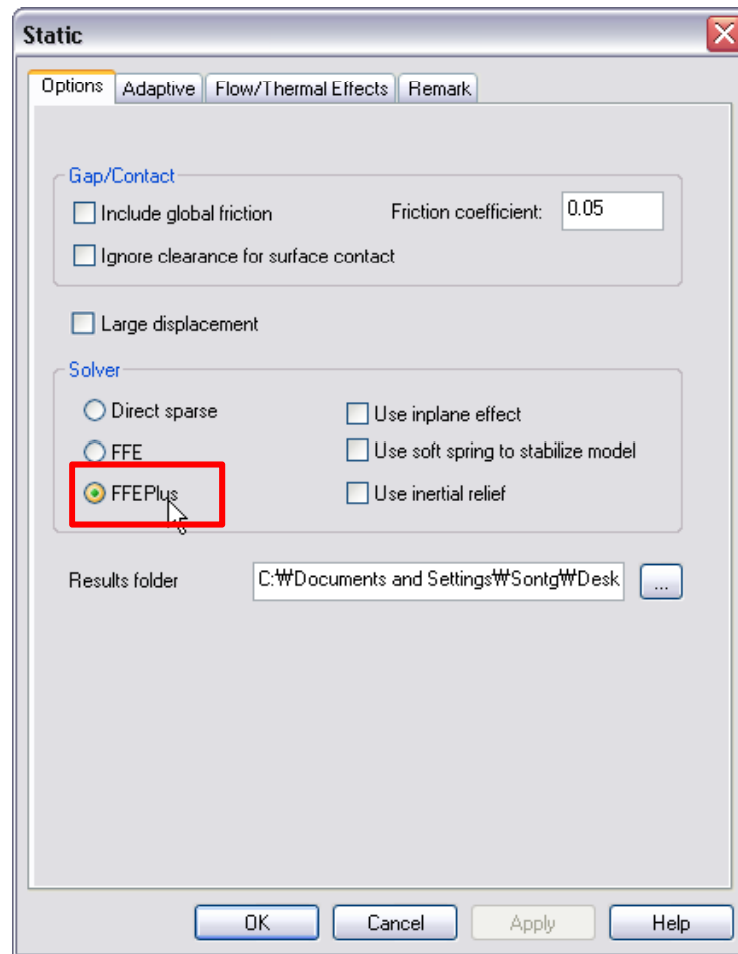
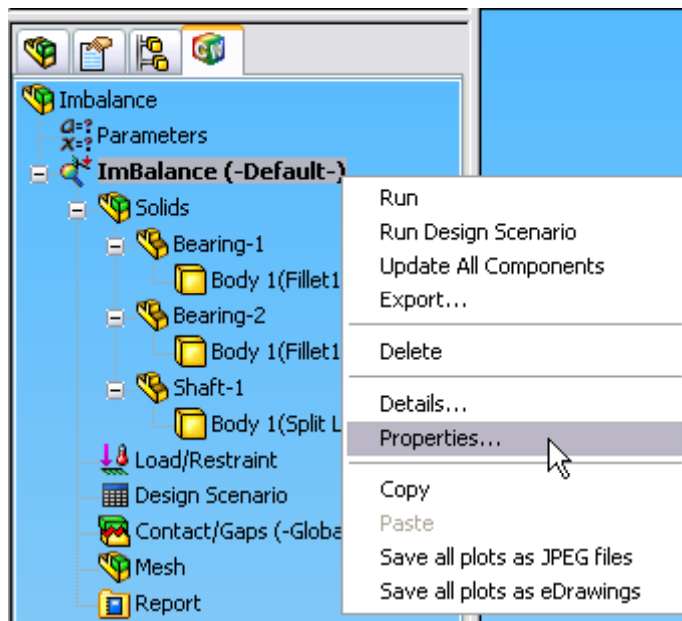
Create a Study

Study >> Solid Mesh & Static



Properties

ImBalance >> Properties >> FFEPlus Solver



Define Materials

COSMOSWorks >> Material >> Apply Materials to All... >> Alloy Steel

The screenshot displays the COSMOSWorks interface. On the left, the 'Material' menu is open, with 'Apply Material to All...' selected. The 'Material' dialog box is open, showing the 'From library files' option selected. The 'Material Properties' tab is active, displaying the following table:

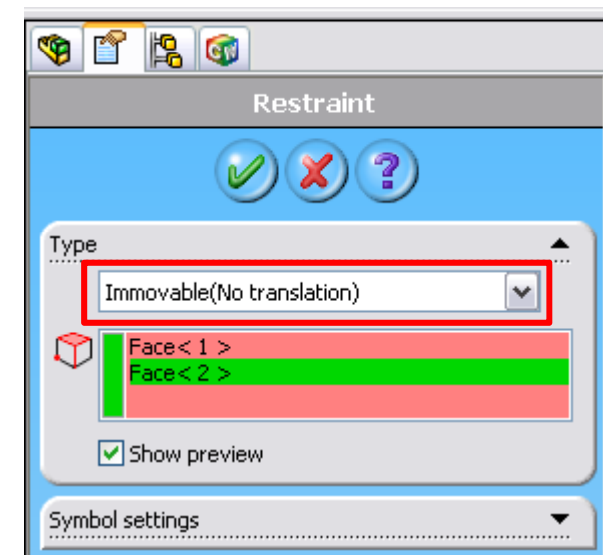
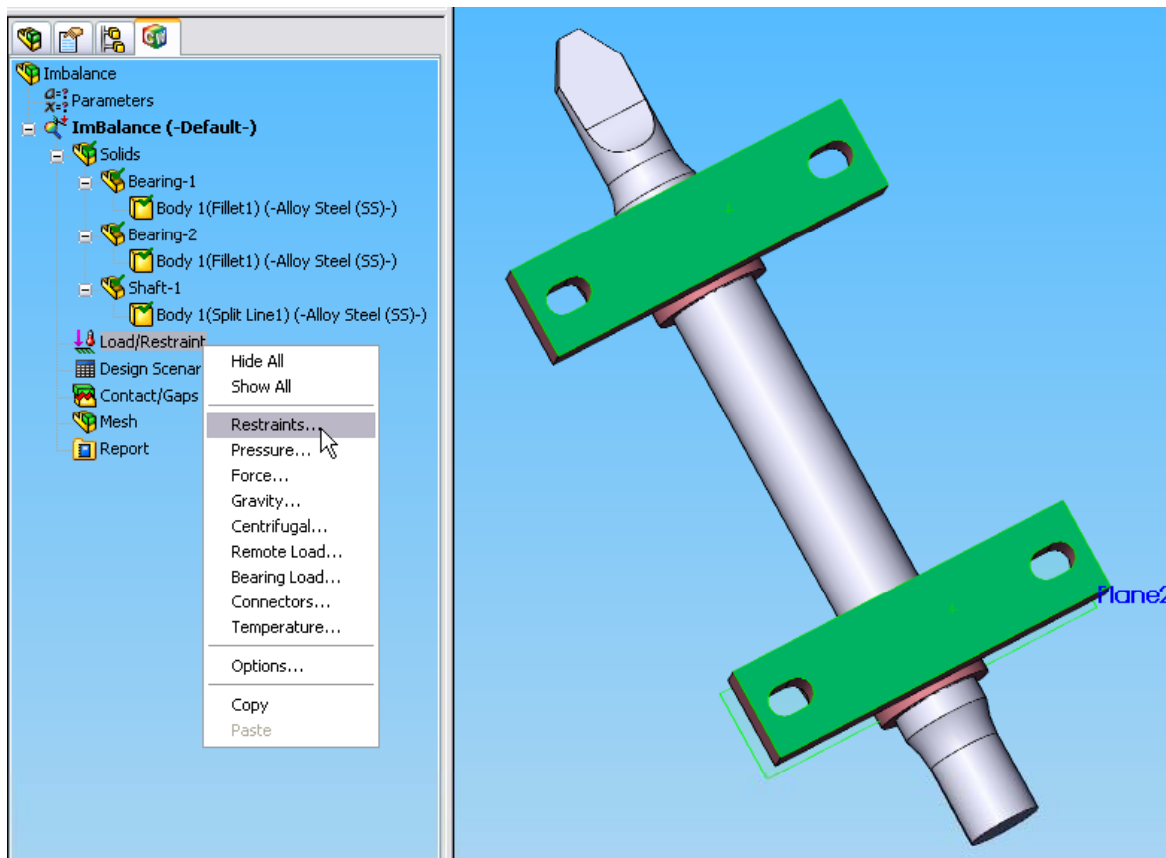
Property	Description	Value	Units	Temp Dependency
EX	Elastic modulus	2.1e+011	N/m ²	Constant
NUXY	Poisson's ratio	0.28	NA	Constant
GXY	Shear modulus	7.89999998e+010	N/m ²	Constant
DENS	Mass density	7700.0001	kg/m ³	Constant
SIGXT	Tensile strength	7.2382562e+008	N/m ²	
SIGXC	Compressive strength		N/m ²	
SIGYLD	Yield strength	6.20422e+008	N/m ²	
ALPX	Thermal expansion coe	1.3e-005	/Kelvin	
KX	Thermal conductivity	50	W/(m.K)	
C	Specific heat	460	J/(kg.K)	

The CAD model tree on the right shows the following structure:

- Imbalance
 - Parameters
 - Imbalance (-Default-)
 - Solids
 - Bearing-1
 - Body 1(Fillet1) (-Alloy Steel (SS)-)
 - Bearing-2
 - Body 1(Fillet1) (-Alloy Steel (SS)-)
 - Shaft-1
 - Body 1(Split Line1) (-Alloy Steel (SS)-)
 - Load/Restraint
 - Design Scenario
 - Contact/Gaps (-Global: Bonded-)
 - Mesh
 - Report

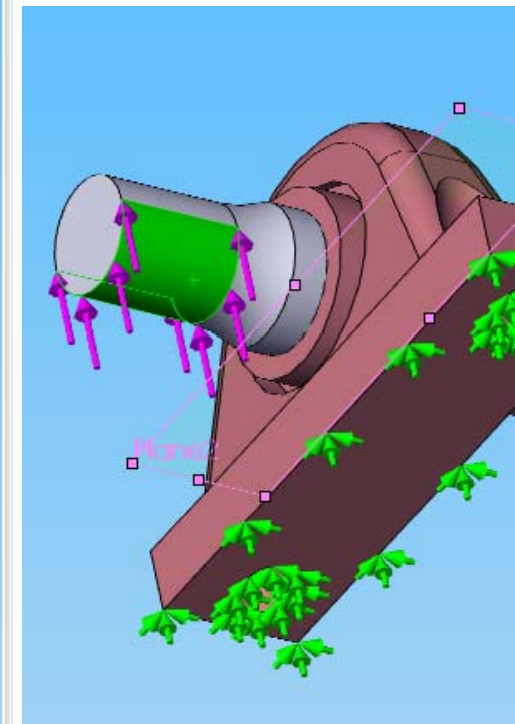
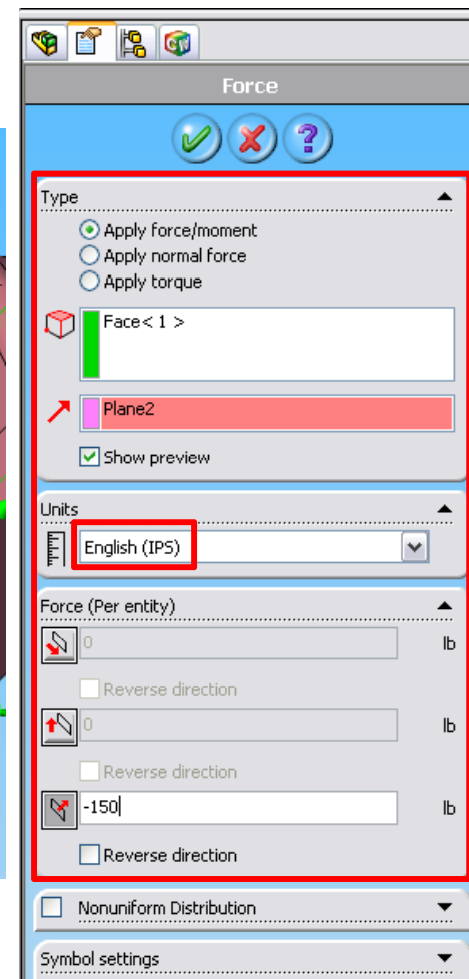
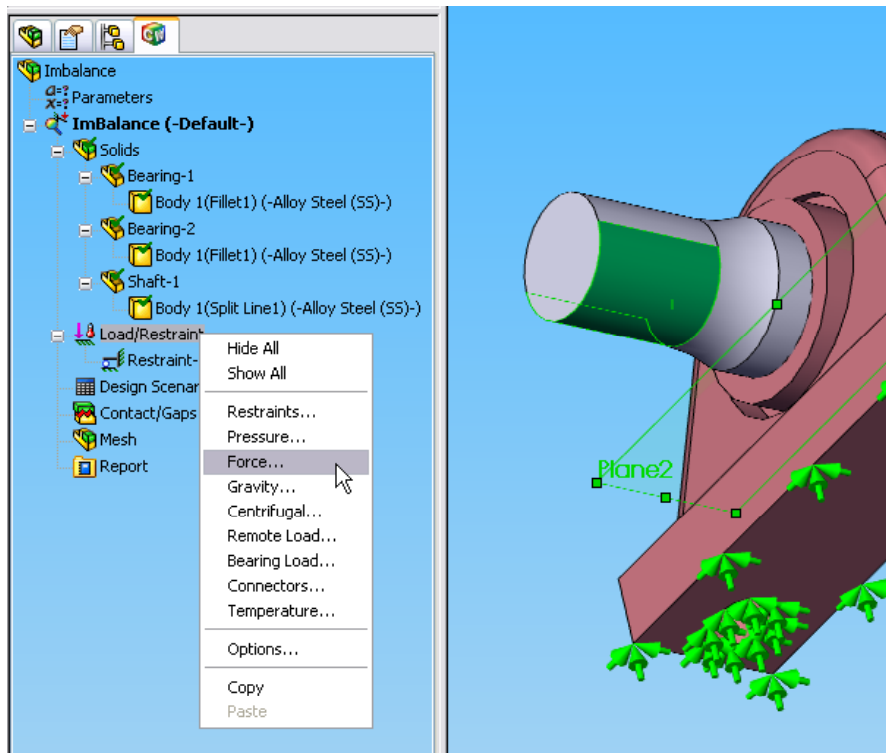
Restraint - Immovable

Load/Restraints >> Restraints >> Immovable



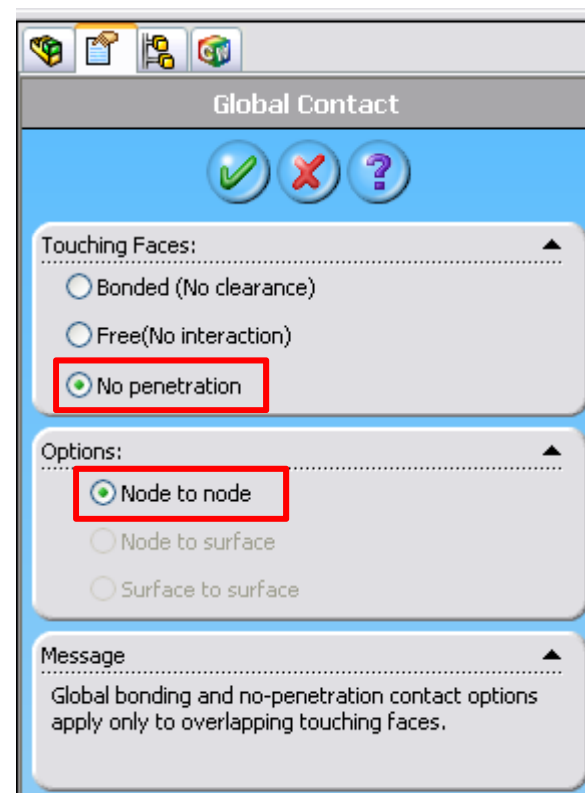
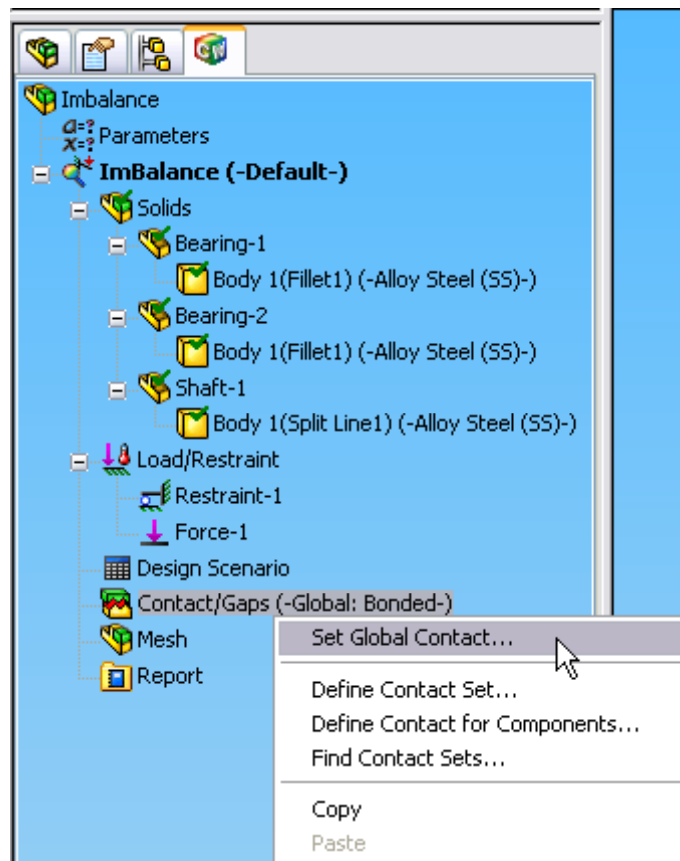
Force - Normal to Plane

Load/Restraints >> Force >> Normal to Plane >> - 150 lb



Contact/Gaps

Contact/Gaps >> Set Global Contact >> No Penetration >> Node to Node

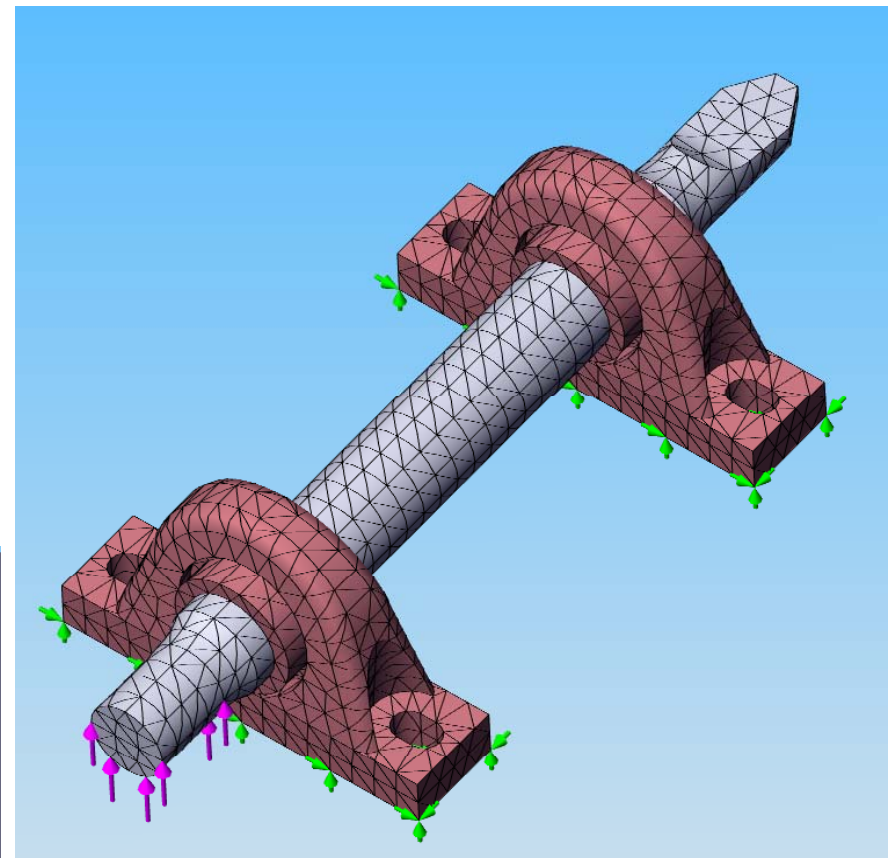
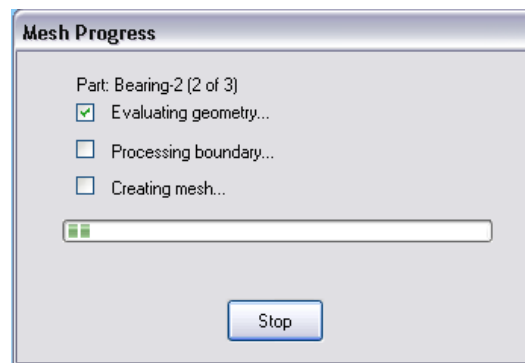
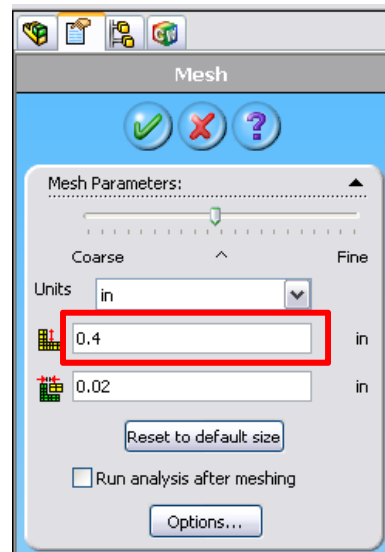
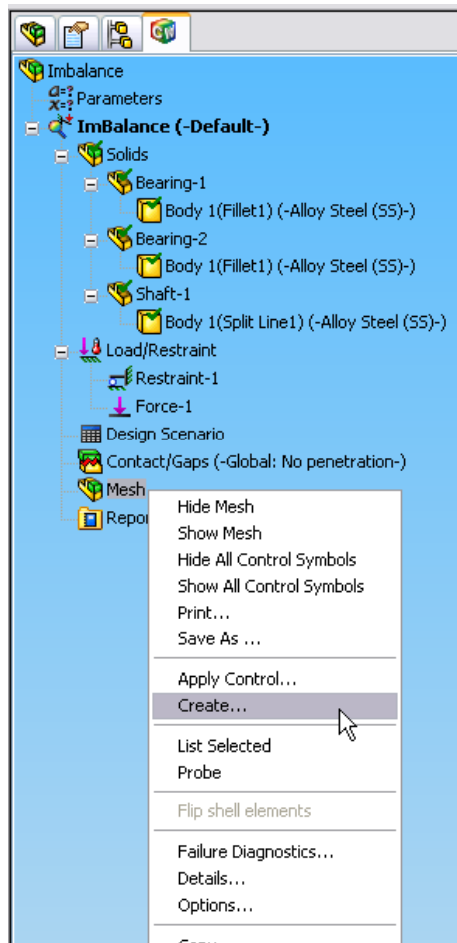




Create Mesh

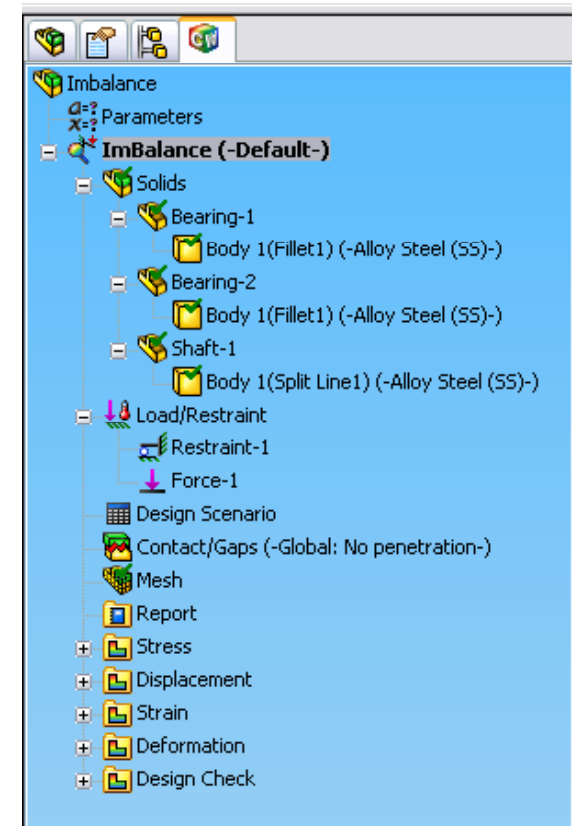
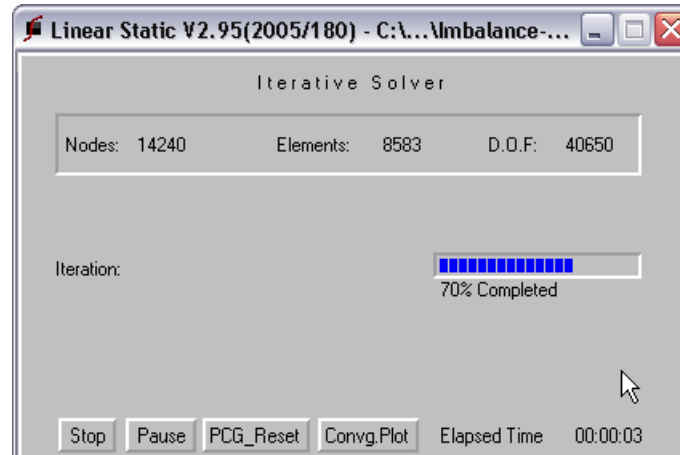
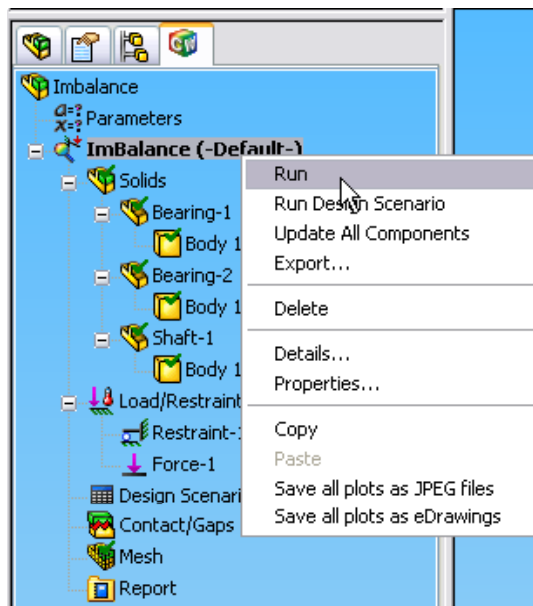


Mesh >> Create >> 0.4 in



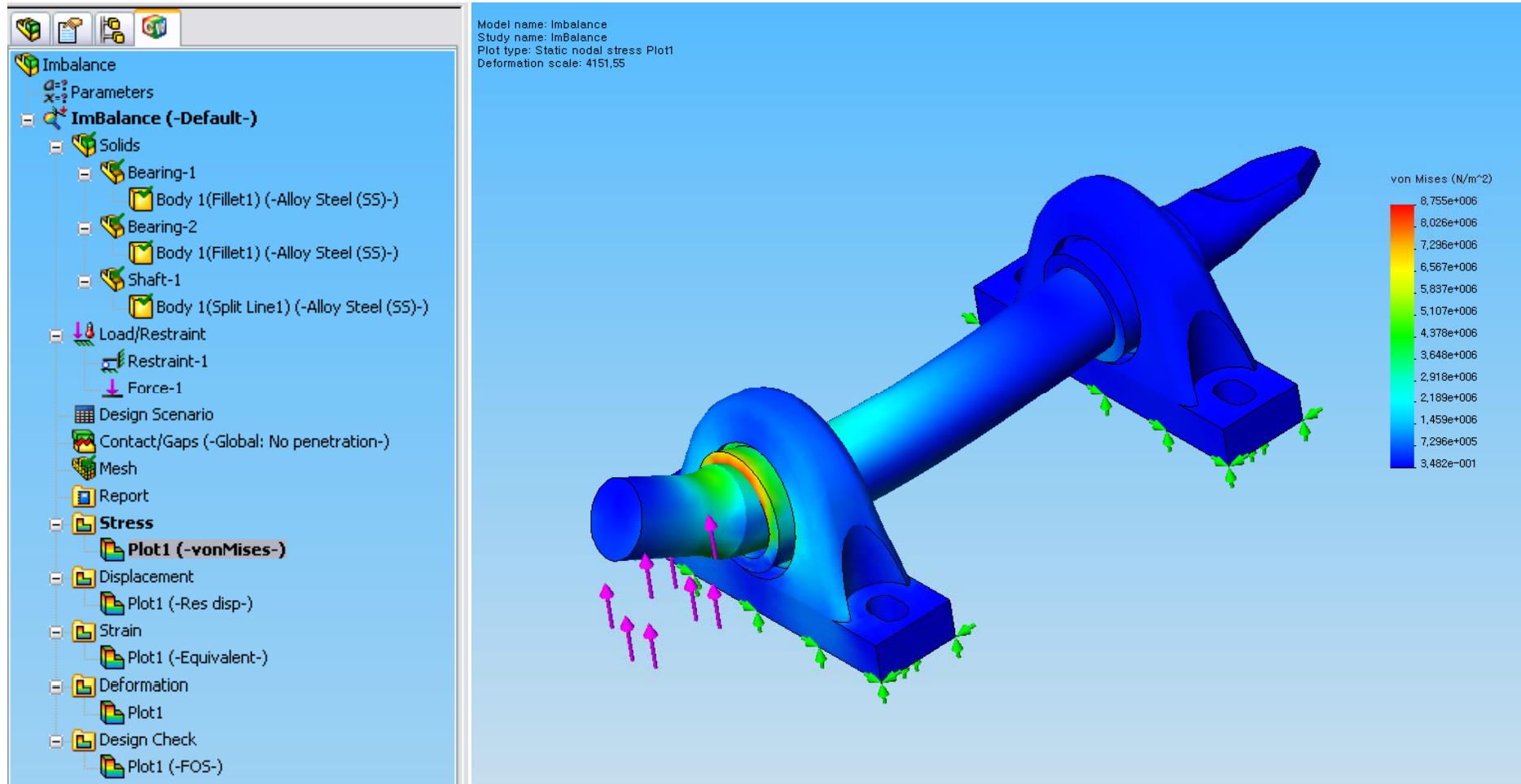
Run Static Analysis

ImBalance >> Run





Stress Plot

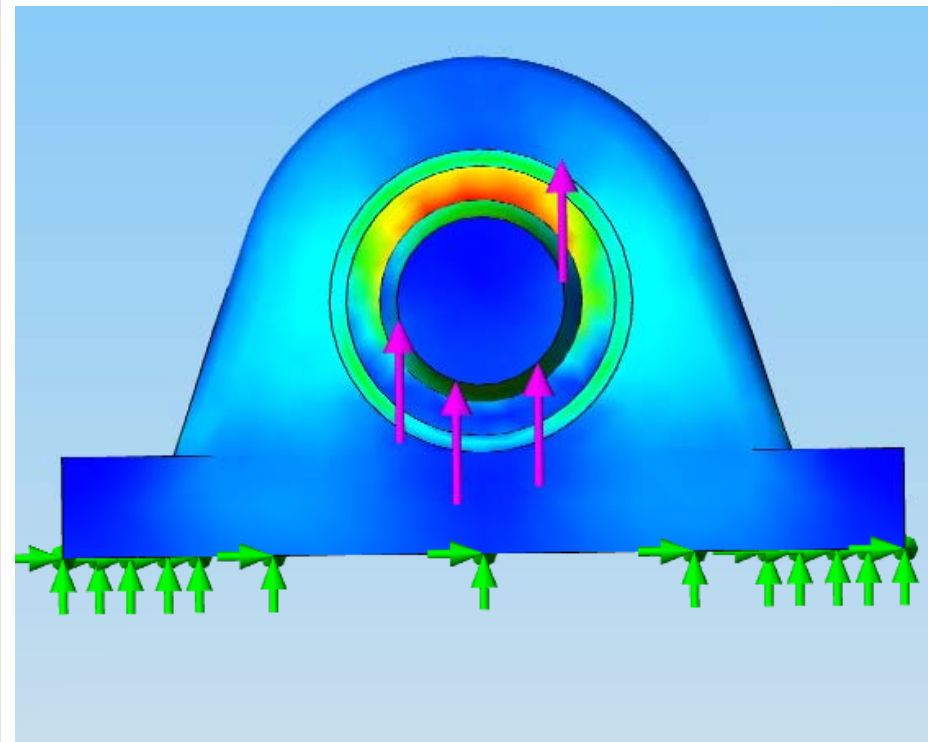
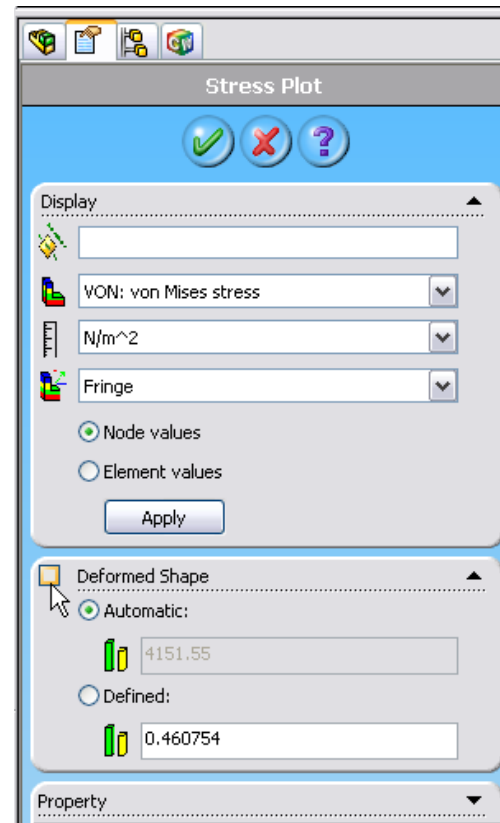
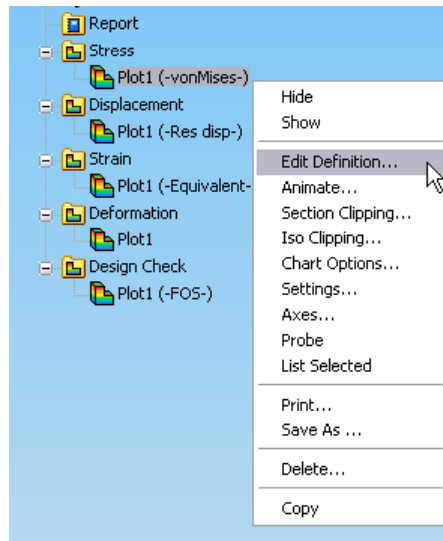


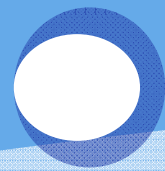


Deformed Shape



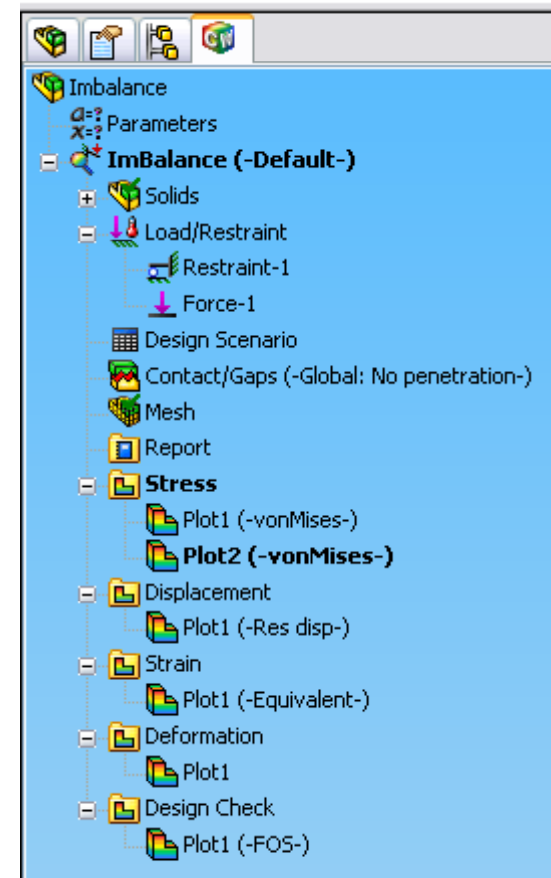
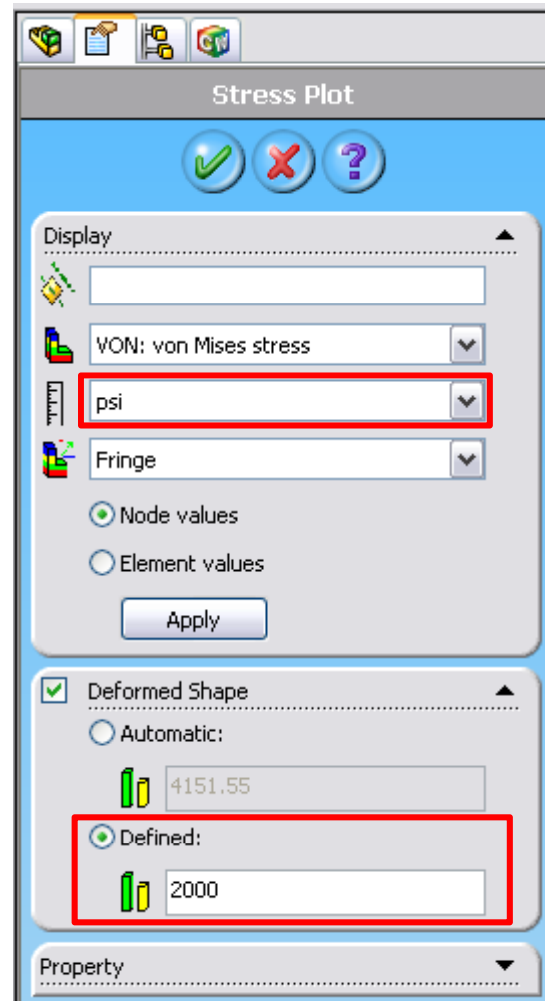
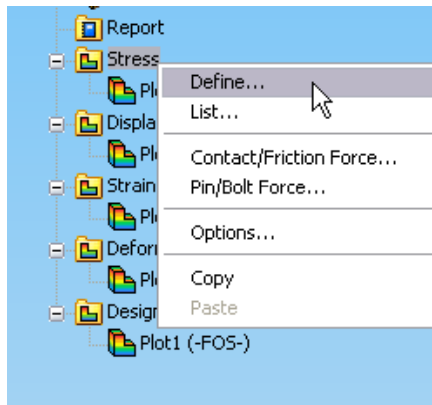
Stress Plot >> Edit Definition >> No Scale Factor





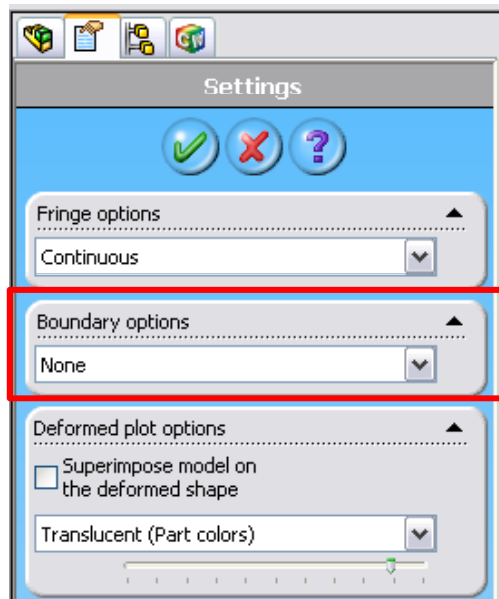
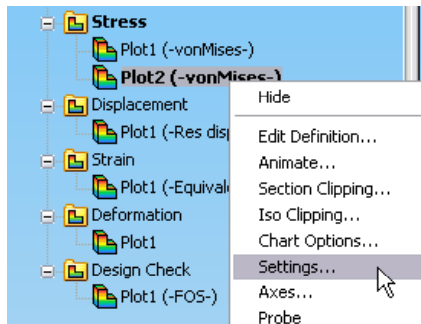
Create New Plot

Stress >> Define

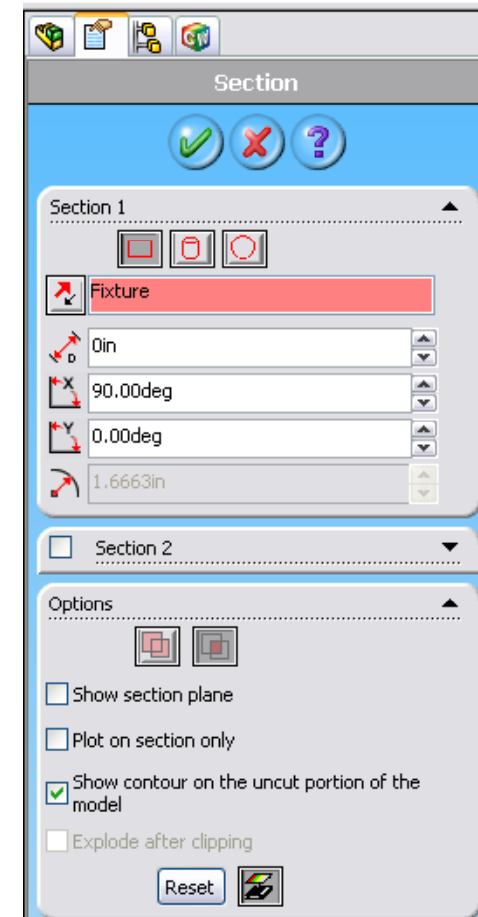
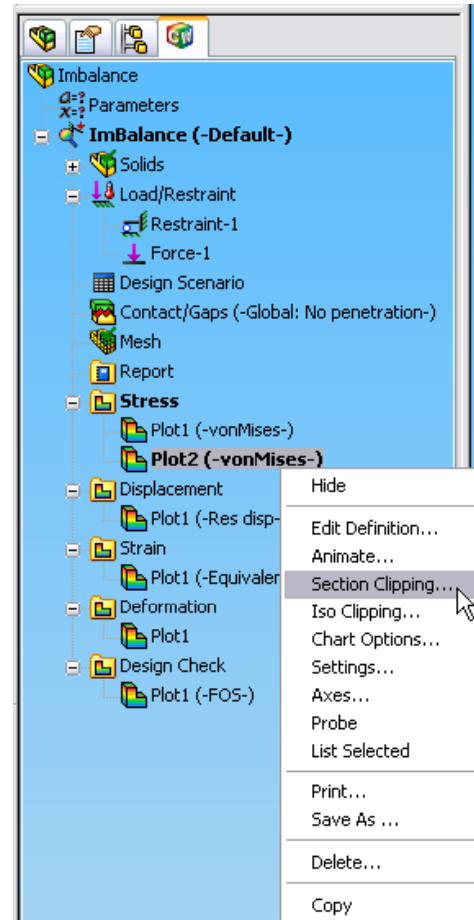


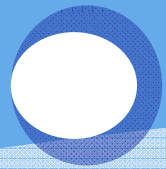
Plot Setting

Setting >> No Boundary



Section Clipping

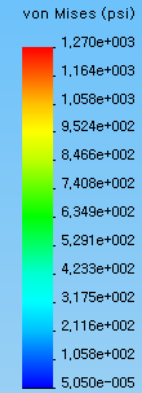
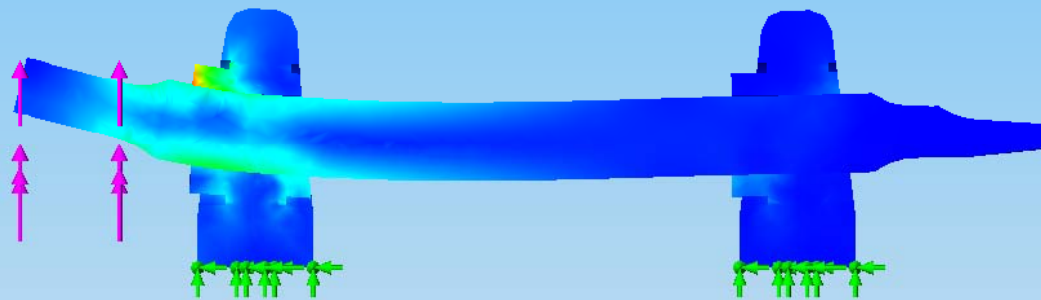




Section View

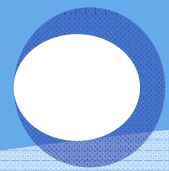


Model name: Imbalance
Study name: ImBalance
Plot type: Static nodal stress Plot2
Deformation scale: 2000





Buckling Analysis

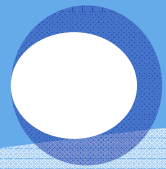


Procedure



- ❖ Create a buckling analysis study
- ❖ Assign material to the part
- ❖ Insert restraints and pressure loading
- ❖ Mesh the part
- ❖ Run buckling analysis
- ❖ Visualize the buckling analysis results by listing critical factors and plotting buckling modes
- ❖ Remesh the part with mesh controls, rerun the analysis, and compare the results

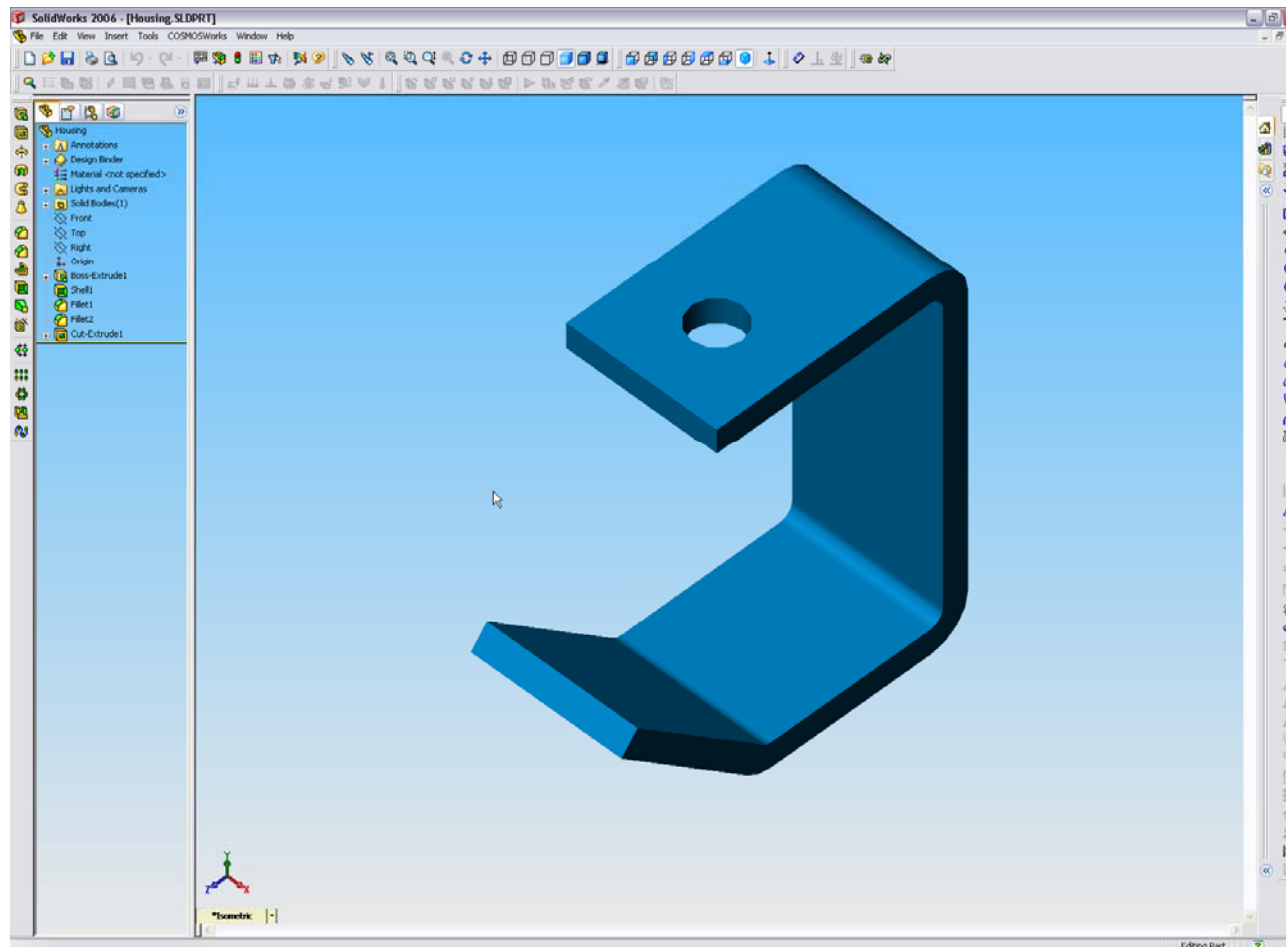




Open

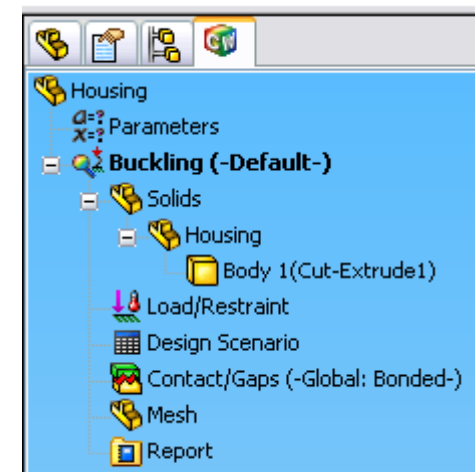
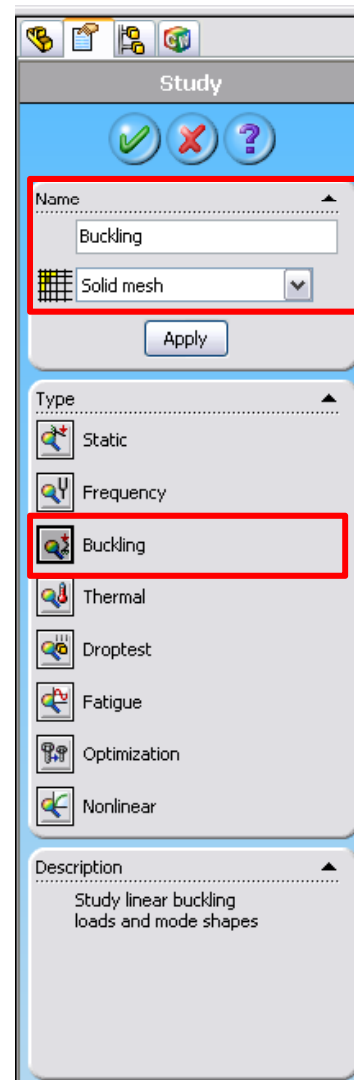
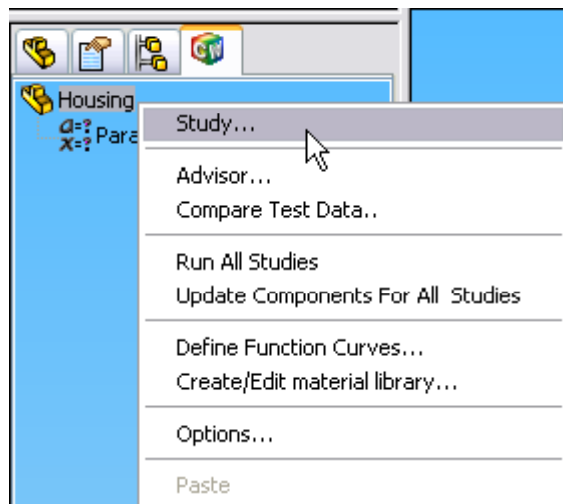


Housing.SLDPRT



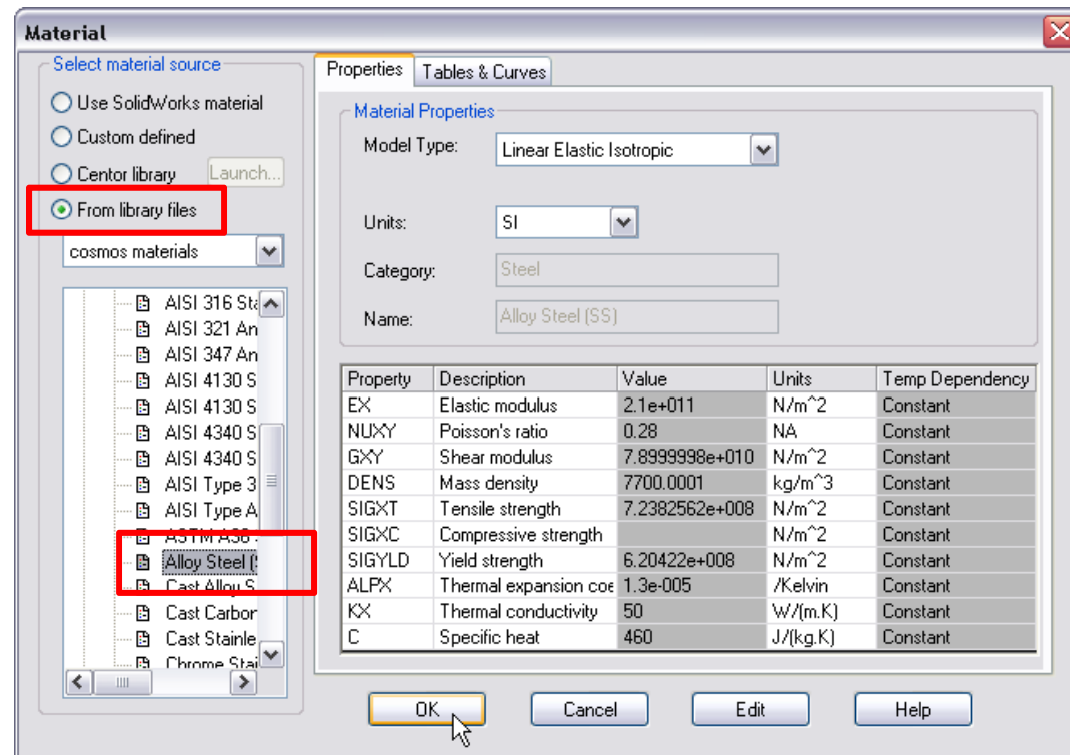
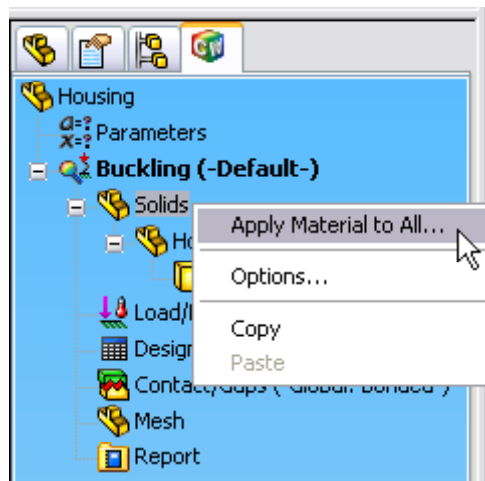
Create a Study

Study >> Solid Mesh & Buckling



Define Material

Solid >> Apply Material to All... >> Alloy Steel

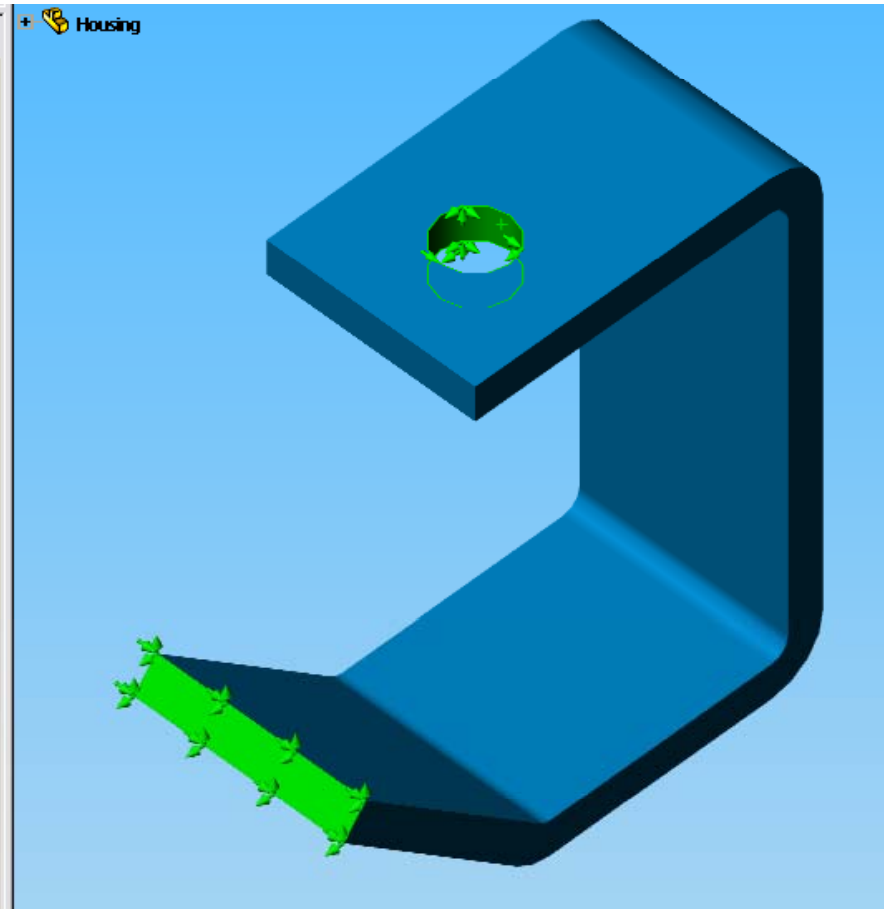
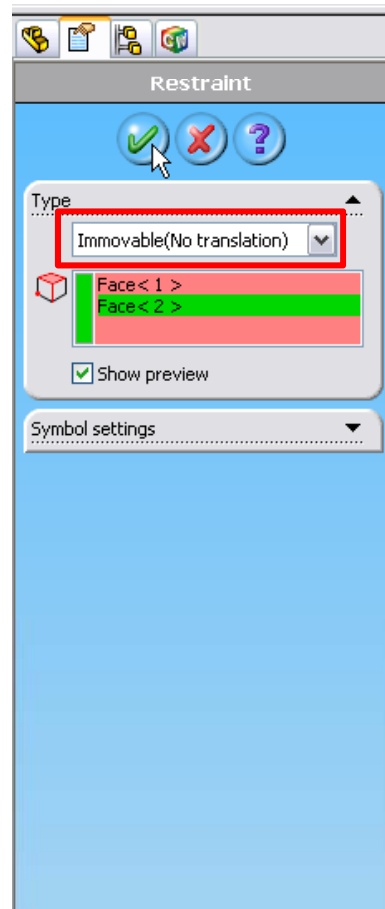
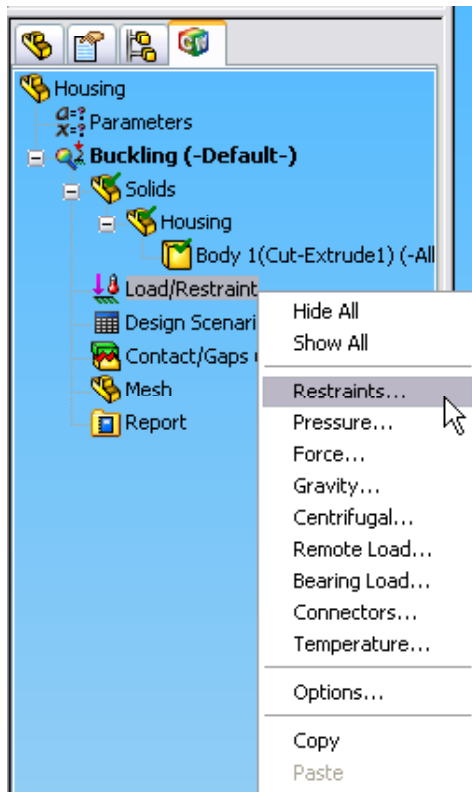


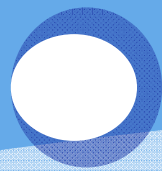


Restraints

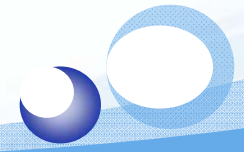


Load/Restraints >> Restraints >> Immovable

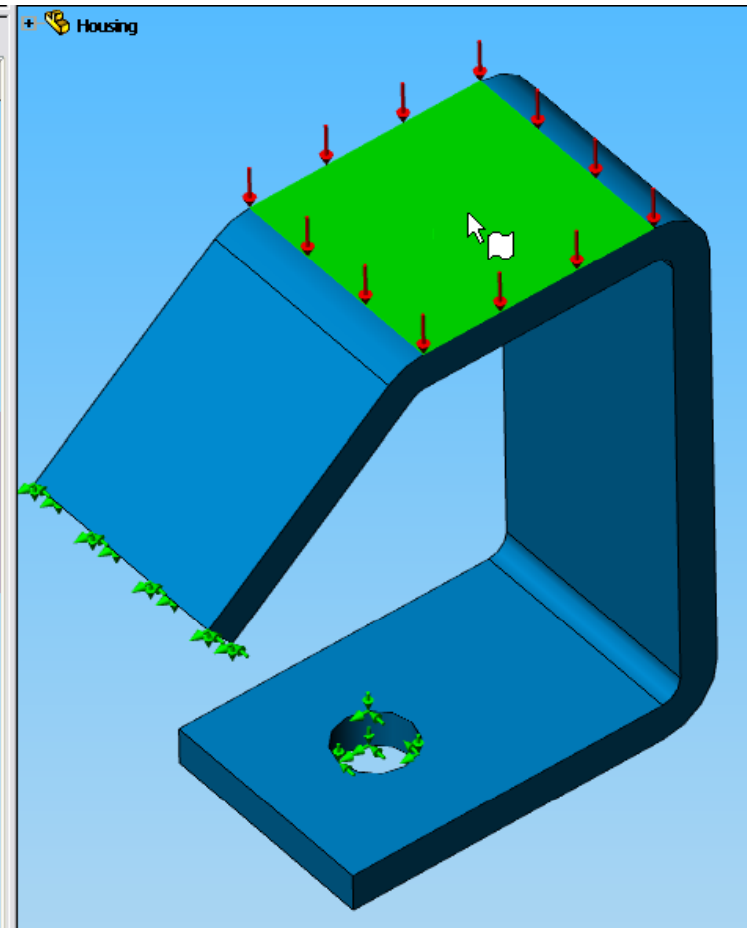
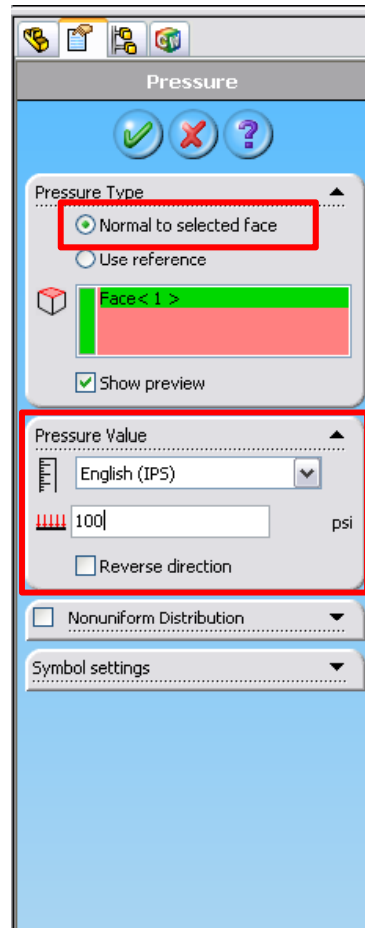
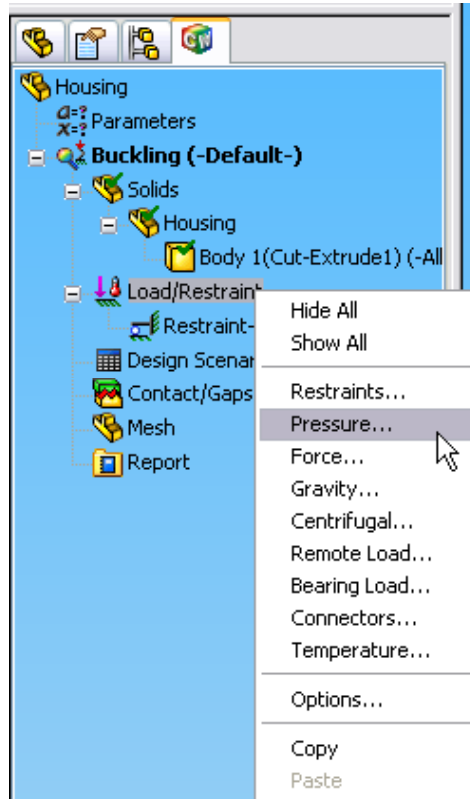




Pressure

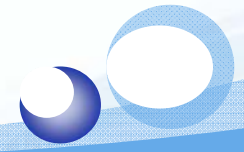


Load/Restraints >> Pressure >> Normal to -100 psi

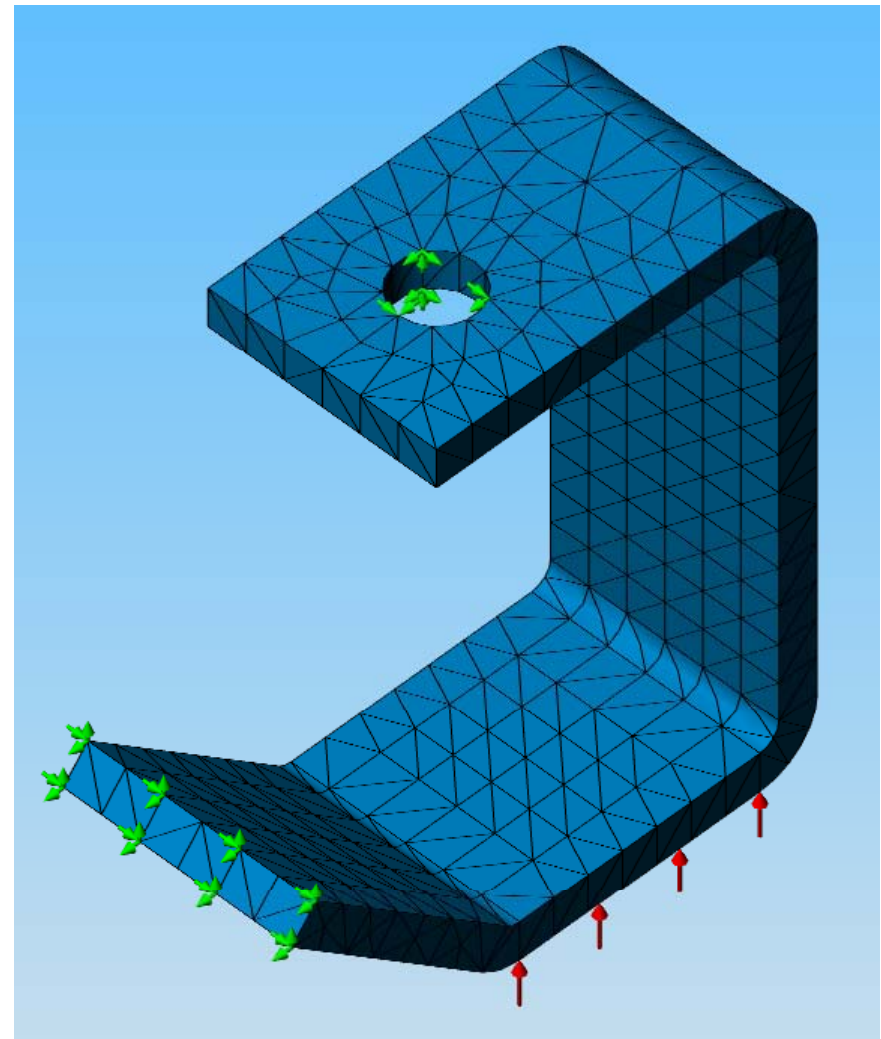
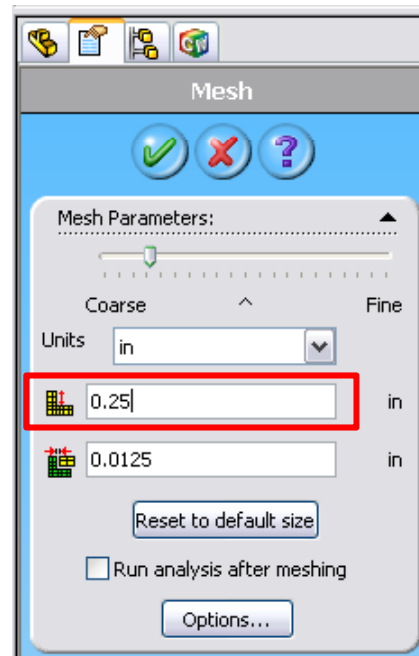
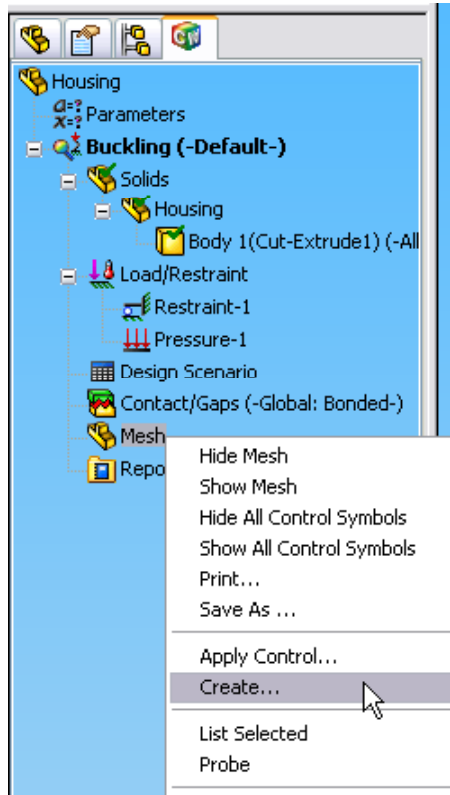




Create Mesh

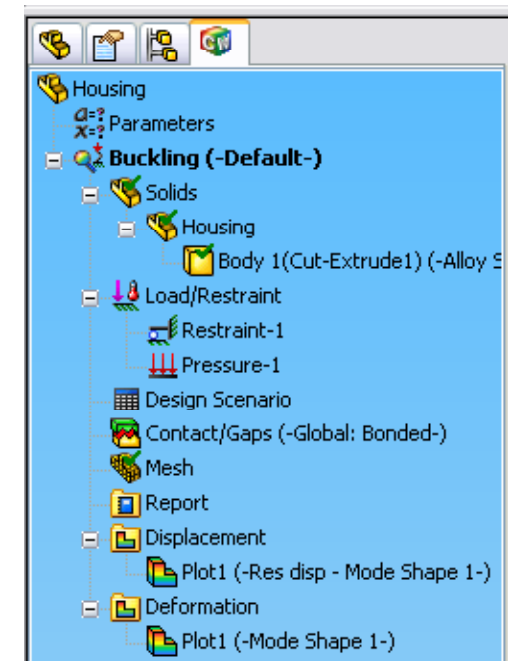
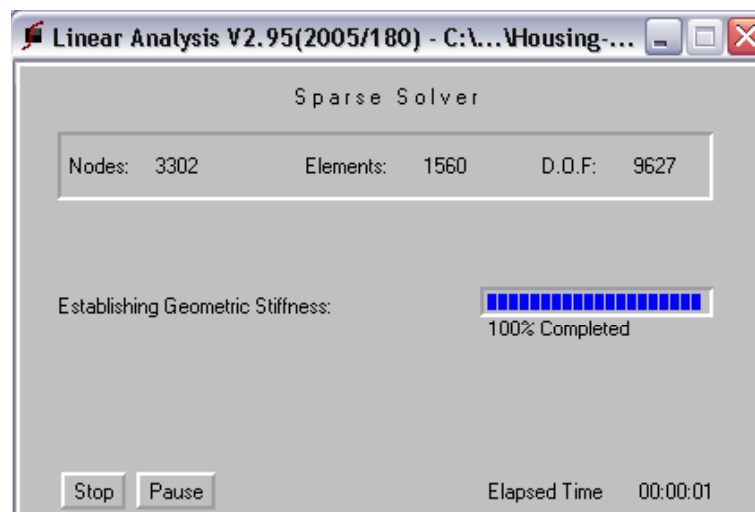
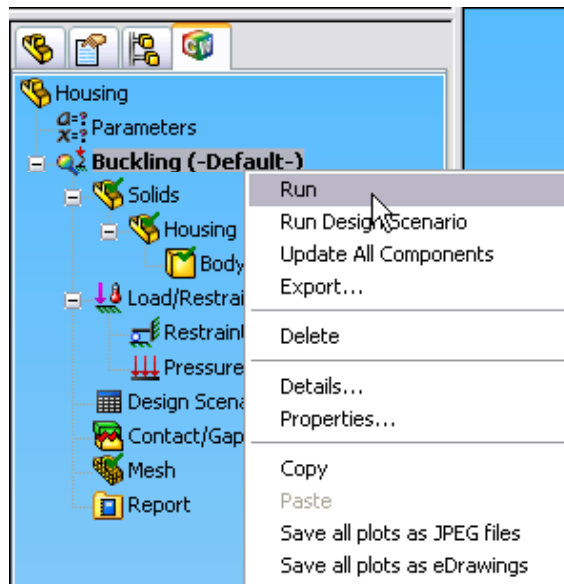


Mesh >> Create >> 0.25 in



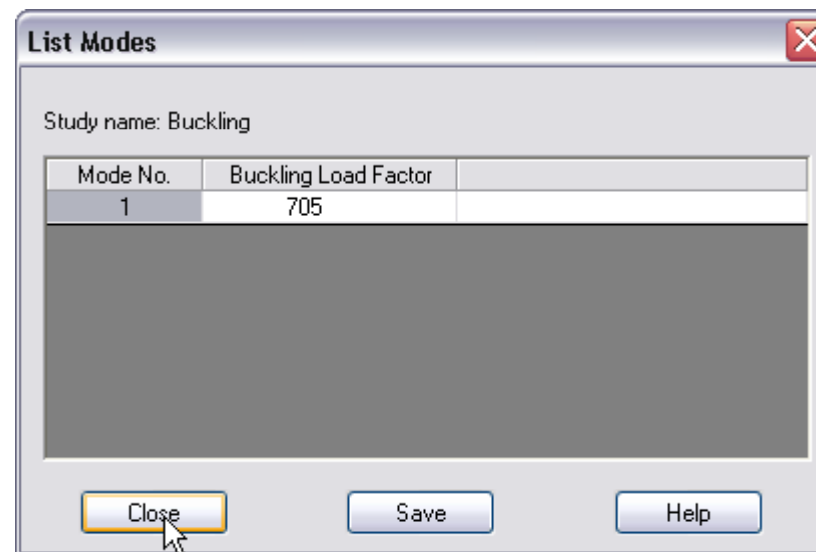
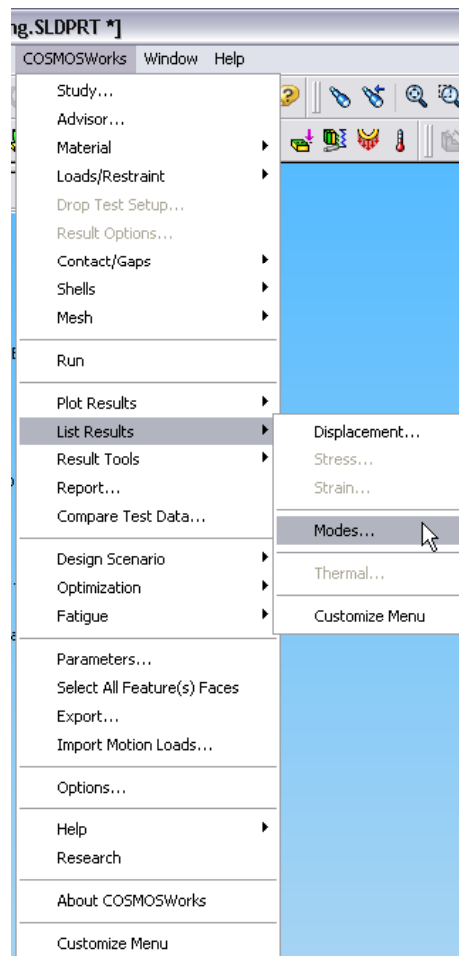
Run Buckling Analysis

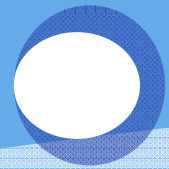
Buckling >> Run



List Result - Mode Shape

COSMOSWorks >> List Result >> Modes

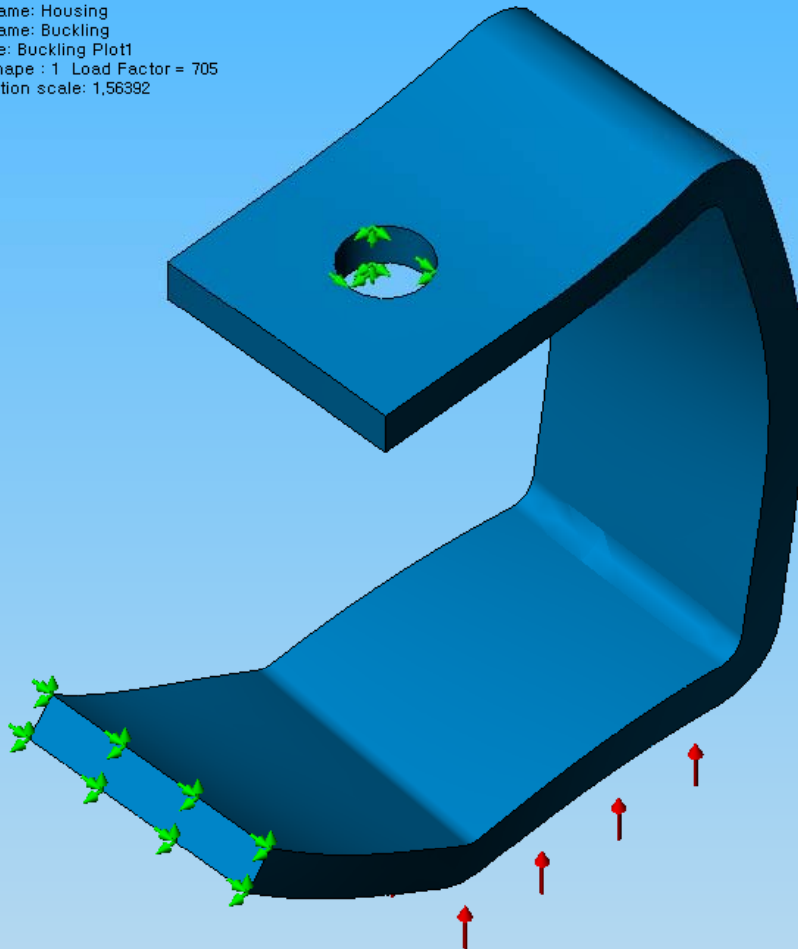




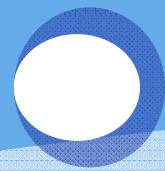
Deformation Plot



Model name: Housing
Study name: Buckling
Plot type: Buckling Plot1
Mode Shape : 1 Load Factor = 705
Deformation scale: 1,56392



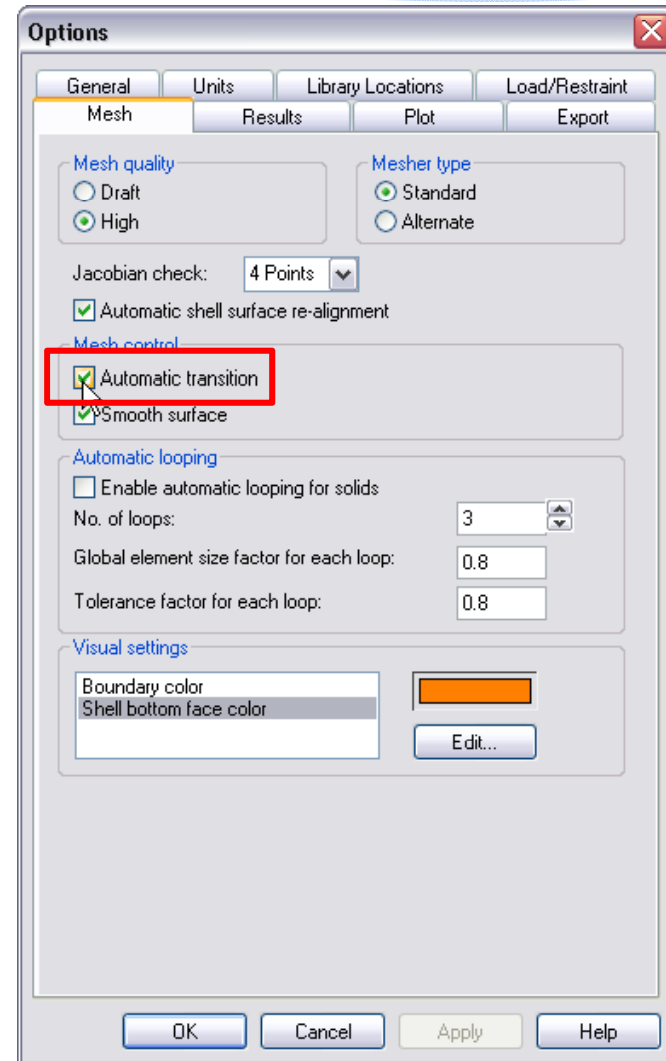
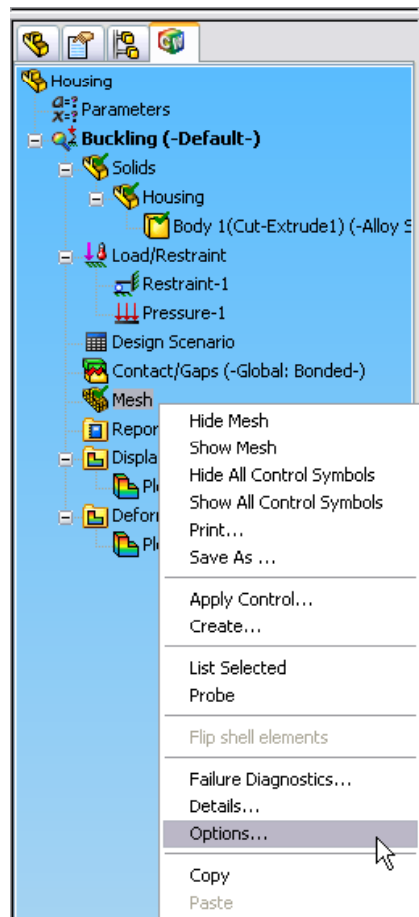
Critical Load Factor = 705
Critical Buckling Load = 70,500 psi
(Pressure * Critical Load Factor)



Mesh Control



Mesh >> Options >> Automatic transition

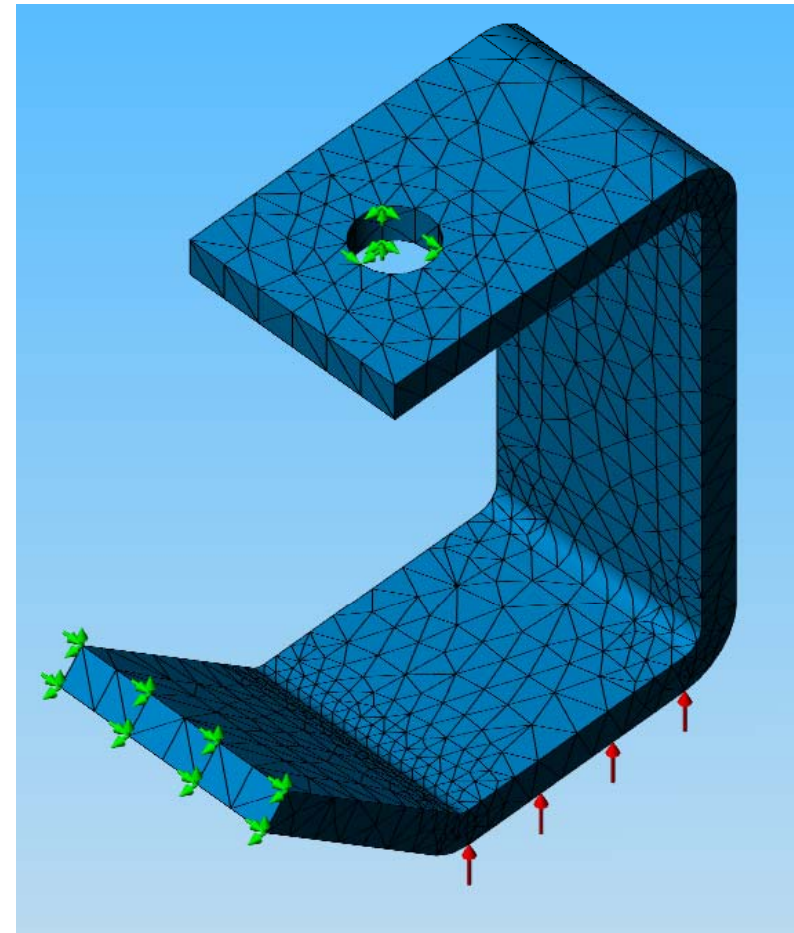
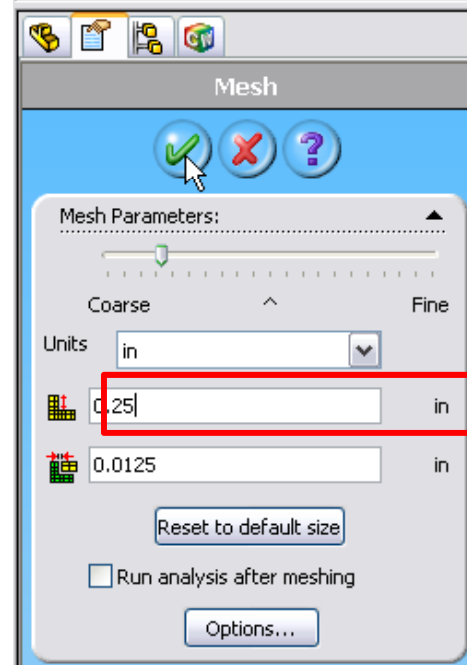
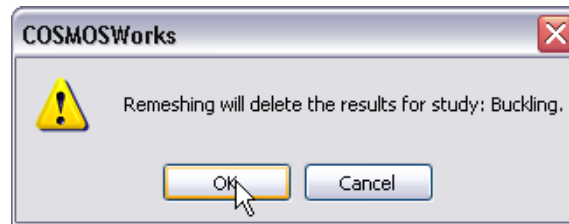
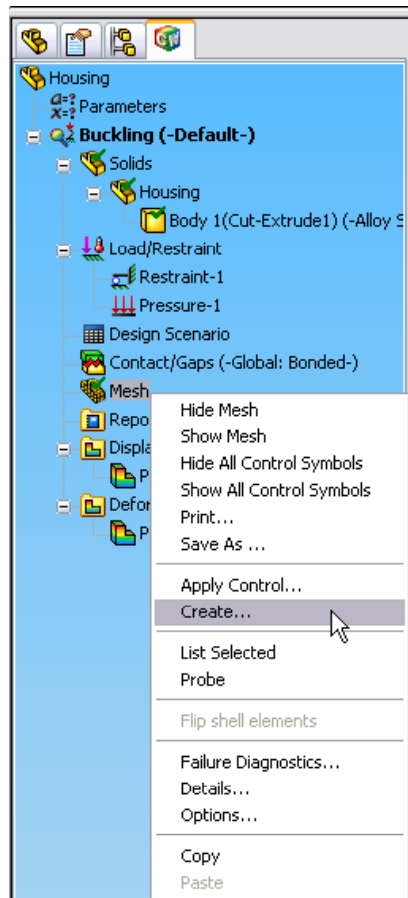




Remesh

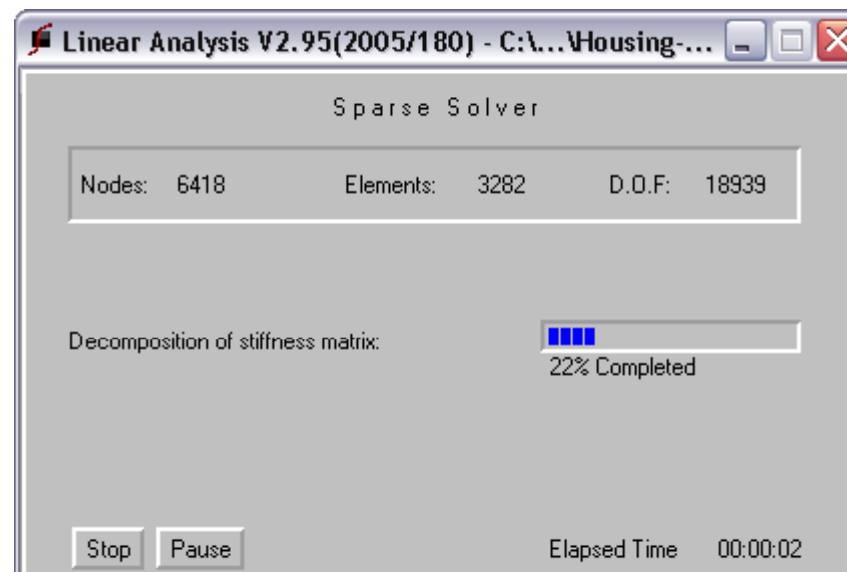
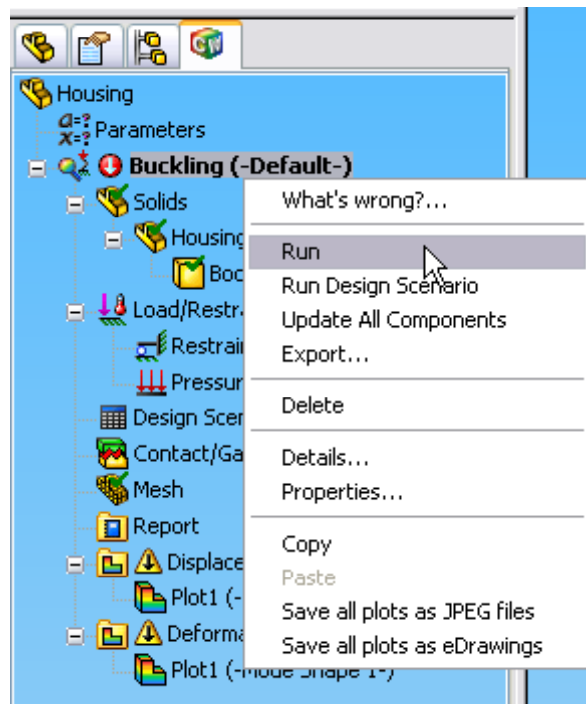


Mesh >> Create >> 0.25 in



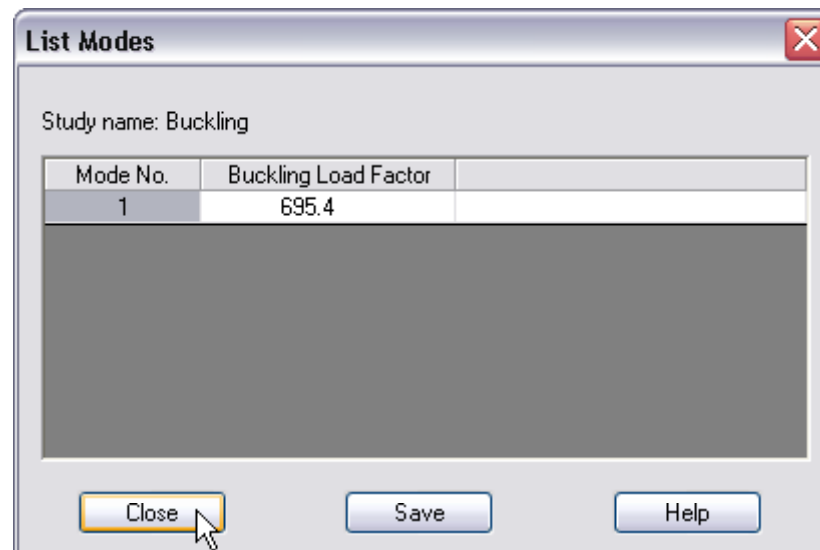
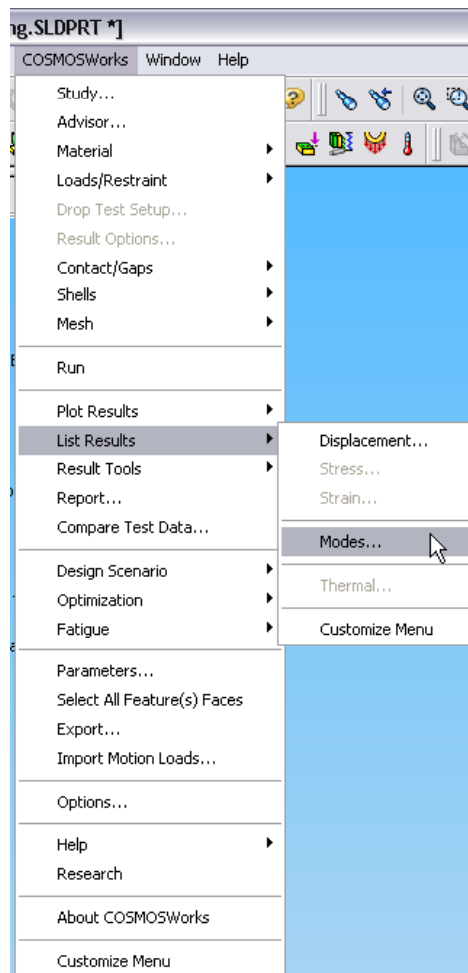
Rerun Buckling Analysis

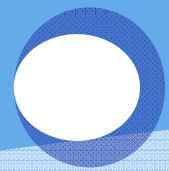
Buckling >> Run



List Result - Mode Shape

COSMOSWorks >> List Result >> Modes

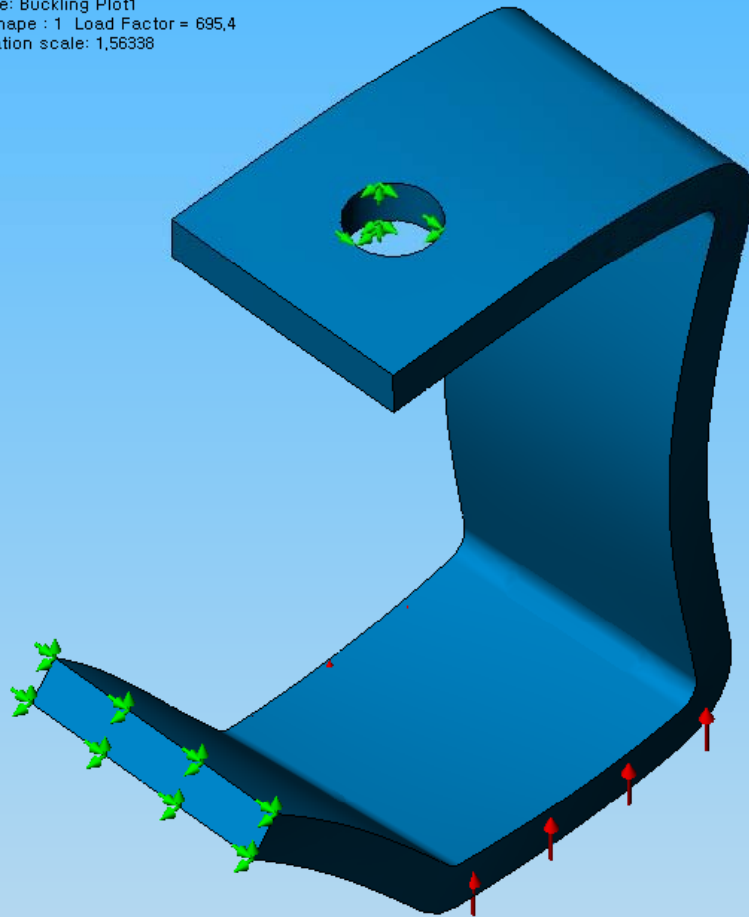




Deformation Plot



Model name: Housing
Study name: Buckling
Plot type: Buckling Plot1
Mode Shape : 1 Load Factor = 695.4
Deformation scale: 1,56338



Critical Load Factor = 695.4
Critical Buckling Load = 69,540 psi
(Pressure * Critical Load Factor)



Thank You !