

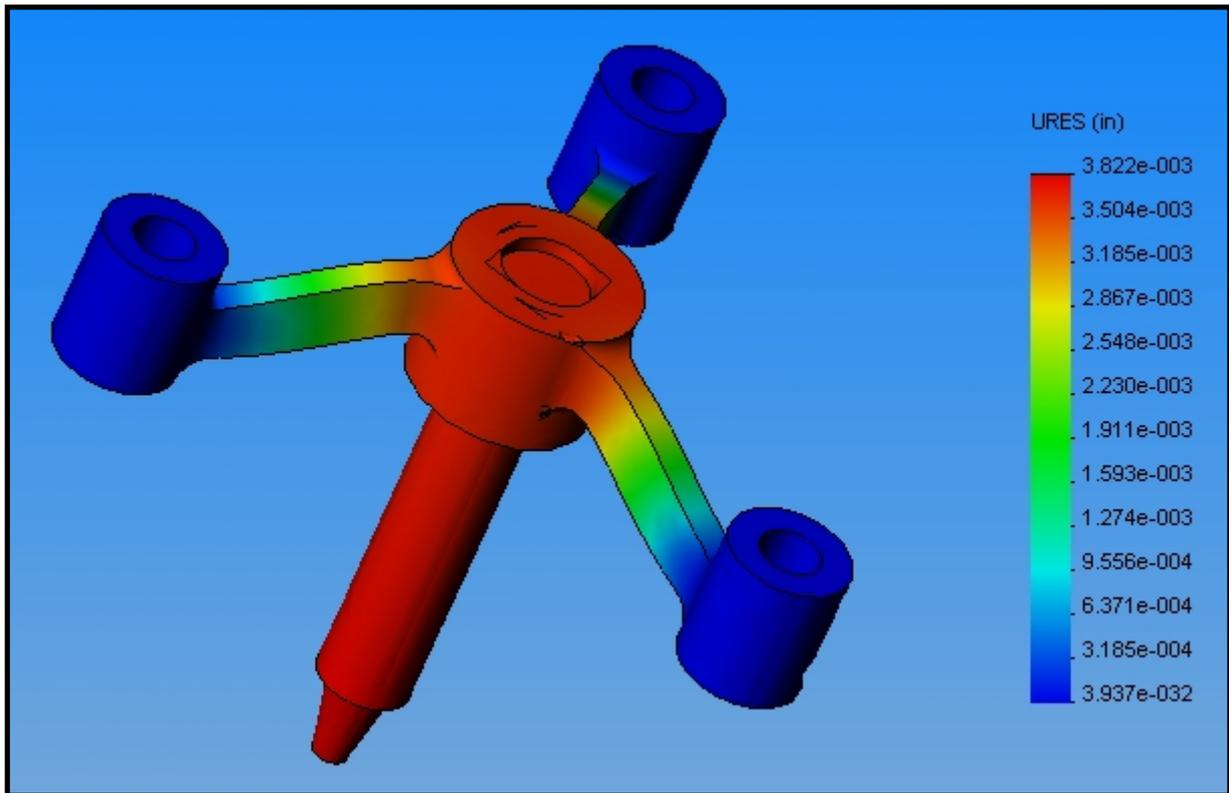
CAD/CAM

Homework 2

FEM Analysis in COSMOSWorks

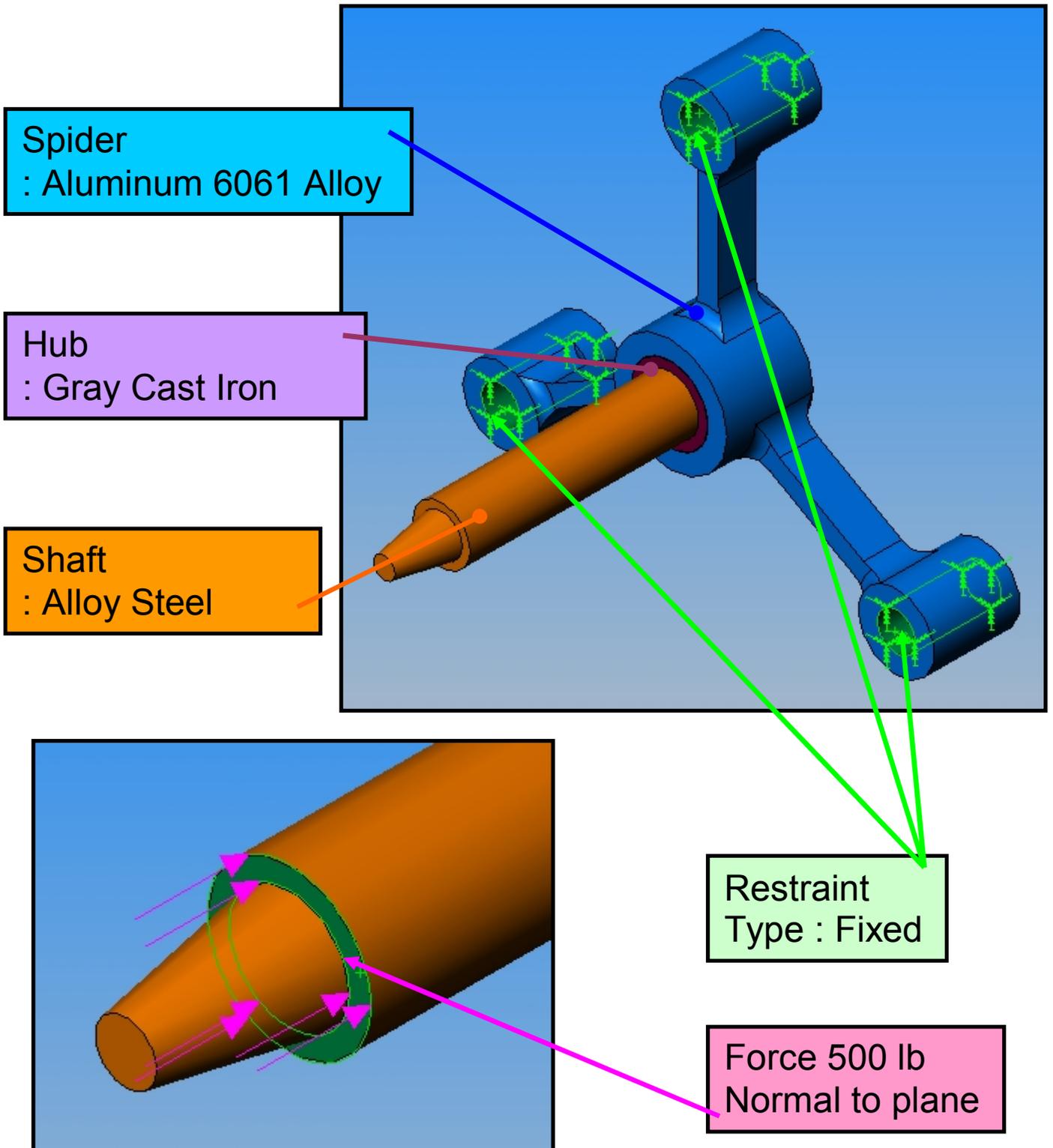
Due Date : 2006.10.24. 23:59:59

Static Analysis



- REQUIREMENT:
 - Open fem.SLDASM file.
 - Make a Study [Static, Solid mesh].
 - Apply Material / Load / Restraint.
 - As shown in NEXT PAGE
 - Use English(IPS) as system of units
 - Create Mesh [Global size : 0.2 in, Automatic transition].
 - Run the static analysis [FFEPlus].
 - Plot the stress [von Mises (psi)].
 - Generate a study report.
 - Type your name in AUTHOR
 - Check Automatically update all plots in JPEG files

Solids / Load / Restraint



Submission Details

- Due : Oct 24, 2006, 23:59:59.
 - Credit : 10 points.
 - Delay penalty : 2 point per 24 hours.

- Objects on demand :
 - SolidWorks Assembly/Part files
 - Report file (save as `***.mht`)
 - Write a short REPORT.TXT file which describes your implementation.

- Submit : <http://eng.snu.ac.kr>
 - You must compress above all files as a Zip file named "HW2_(student ID).zip"
 - Ex) HW2_99446123.zip
or HW2_200012345.zip

- Question :
 - E-mail : zeroneo1@snu.ac.kr
 - Tel : 02 - 880 - 7447