COSMOSWorks Practice

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

10011111



COSMOSWorks in Solidworks

Static Analysis

Frequency Analysis

Contact Analysis

Buckling Analysis

COSMOSWorks in Solidworks

1. CLORADIOD

Add-in / Toolbar

Tool >> Add-in >> COSMOSWorks







COSMOSWorks Manager



Static Analysis

0100000000000

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





Crank.SLDASM

SolidWorks 2006 - [Crank.SLDASM]		_ B 🔀
😵 File Edit View Insert Tools COSMOSWorks Window Help		- 8 ×
0 2 1 2 1 2 2 1 2 2 2 2 2 2 2 2 2 2 2 2	LQQQQ+DDDDDDDDDDDDDDDDDDDD	
QIEBEINEBENE → TOSANY1	KKKKOU > WKK/20 %	
Image: Section of the section of t	<image/>	2 10 10 10 10 10 10 10
Castom [+]		
		Editor According

Create a Study

Study >> Solid Mesh & Static



%	
Crank Crank Z=? P. X=? P.	Study Advisor Compare Test Data
	Run All Studies Update Components For All Studies
	Define Function Curves Create/Edit material library
	Options
	Paste





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

Crank	
	Run Design Stonario Update All Components Export
	Delete Details Properties
	Copy Paste Save all plots as JPEG files Save all plots as eDrawings

🔎 Linear S	Static V	2.95(2005/	180)	- C:\'	Crank-Initi	ial 🗖 🗆	X
		ltera	itive	Solve	: r		
Nodes:	15240	Elem	nents:	7704	D.O.F:	42642	
Iteration:					80% Comple	ed	
					core compily	5-	
Stop	Pause	PCG_Reset	Conv	g.Plot	Elapsed Time	e 00:00:02	

Stress Plot



Change Unit - psi

Plot1 >> Edit Definition >> psi



Animate

Plot1 >> Animate









Generating Report

Report >> Define



Cover Page	Cover Page
 ✓ Introduction ✓ Description ✓ File Information 	Logo File:
Materials Load & Restraint Information Contact Study Property	Browse Title: Stress analysis of Crank
 ✓ Stress Hesults ✓ Strain Results ✓ Displacement Results ✓ Deformation Results 	Author: Tae Ho Jang
Design Creck Results Design Scenario Results Conclusion Appendix	Company: HCCL
	Date:
eport path: C:₩Docume	is and Settings₩Sc
teport file name: Crank-Initial-1	Report format: HTML Microsoft Wo
Show report on OK	Automatically update all plots in JPEG files 👘 📃 Print version



Stress analysis of Crank	7. Stree	ss R	esults					
Author: Tae-guen Son Company: HCCL 1. Introduction 2. File Information 3. Materials 4. Load & Restraint Information 5. Study Property 9. Constant	3. Mate	Name Plot1	VON: von Mises stress	Min 0.0717146 N/m^2 Node: 3327	Location (0.265 in, -4.22274 in, -3.684 in)	Max 2,64092e+ N/m^2 Node: 191	Loca (0,641 in, 0,9058 1 in, 3,434	tion 187 319 in)
 7. <u>Stress Results</u> 8. <u>Strain Results</u> 9. <u>Displacement Results</u> 10. <u>Deformation Results</u> 11. <u>Design Scenario Results</u> 12. <u>Appendix</u> 		No. 1 9 2 0 3 0 4 0	Part Name CrankArm-1 Al CrankArm-2 Al CrankArmAxle-1 St CrankPulley-1	Material loy Steel (SS) loy Steel (SS) tainless Steel (fe ray Cast Iron (SN	1. 1. <u>rritic)</u> 1.!] <u>)</u> 9,:	Mass 17195 kg (17195 kg (52621 kg (25209 kg (Volume 0,000152201 m 0,000152201 m 0,000195668 m 0,00128501 m ²	1^3 1^3 1^3 `3



Save Report as Web Page

File >> Save As >> .mht

<i>6</i>	::Woc	ument	ts and Set	tings\Se	ontg\			
9	🔄 🕞 👻 🌈 C: \Documents and Settin							
File	Edit	View	Favorites	Tools	Help			
Ne	w Tab			Ctrl+1	-			
Ne	w Wind	low		Ctrl+N	J I			
Op	ben			Ctrl+0	>			
Ed	lit with I	Microsof	t Office Wo	rd				
Sa	ve			Ctrl+9	5			
Sa	ve As		•					
Cle	ose Tab	I	6	Ctrl+\	V			
Pa	ige Seti	q.						
Print				Ctrl+F	>			
Pri	int Prev	iew						
Se	nd				•			
Im	port an	id Expor	t					
Pr	operties	5						
W	ork Offl	ine						
E×	it							





Frequency Analysis

1 010011110000

The COLOR OF THE STREET

Procedure

Create a frequency analysis study
 Run frequency analysis
 Visualize frequency analysis study



Add New Study

Study >> Solid Mesh & Frequency

🧐 😭 😵	
Crank	Study Advisor Compare Test Data Run All Studies Update Components For All Studies Define Function Curves Create/Edit material library
Mesh	Options
🕂 📋 Report	Paste
⊕ Displacem ⊕ Displacem ⊕ Strain ⊕ ⊡ Deformati ⊕ ⊡ Design Ch	ent on eck

🎕 👔 😫 🚳 📃 👘
Study
Name
Freq
Solid mesh 💌
Apply
Туре
Katic
<table-of-contents></table-of-contents>
4 Thermal
Croptest
🕎 Fatigue
(Continuization
Konlinear
Description
Study resonant frequencies and mode
shapes

👒 😭 😫 🚳
😘 Crank
A=? Parameters
🛓 🗬 Initial (-Default-)
🕂 📢 Solids
😛 🚂 Load/Restraint
Design Scenario
R Contact/Gaps (-Global: Bonded-)
- 🍯 Mesh
🗉 📋 Report
🗉 🛅 Displacement
🗄 🛅 Strain
Deformation
主 🛅 Design Check
🛓 🔍 Freq (-Default-)
🚊 🧐 Solids
🛓 🤏 CrankArm-1
🛓 🧐 CrankArm-2
🗉 🧐 CrankArmAxle-1
🗉 🧐 CrankPulley-1
Load/Restraint
🔤 🎹 Design Scenario
- 🧖 Contact/Gaps (-Global: Bonded-)
Report

Drag & Drop

Apply Materials, Restraints and Force









Run Frequency Analysis

Freq >> Run

🧐 😭 😫 🞯	
Grank	
	Run Run Design Scenatio Update All Components Export Delete Details Properties
🖵 📋 Report	Copy Paste Save all plots as JPEG files Save all plots as eDrawings

Mash Drogram			
Mesti Progress			🧐 😭 😫 🚳
Part: CrankArmAxle-1 (3 of 4)			
Evaluating geometry			4:? Parameters
Processing boundary			🗉 発 Initial (-Default-)
			🛓 🔍 Freq (-Default-)
Creating mesn	N		🛓 🥎 Solids
	4		🚊 🕌 Load/Restraint
			Restraint-1
			Force-1
Stop			🎹 Design Scenario
			🔀 Contact/Gaps (-Global: Bonded-)
			🧐 Mesh
📁 Linear Analysis V2.95(2005/180) - C	:\\Crank-Fre	g 🗕 🗆 🗙	Report
Iterative Solv	er.		Plot1 (-Res disp - Mode Shape 1-)
			Plot2 (-Res disp - Mode Shape 2-)
Nodes: 19058 Elements: 9716	6 D.O.F:	53304	Plot3 (-Res disp - Mode Shape 3-)
· · · · · · · · · · · · · · · · · · ·			Plot4 (-Res disp - Mode Shape 4-)
			Plot5 (-Res disp - Mode Shape 5-)
1			E Correction
Iteration:	55% Completed		Plot1 (-Mode Shape 1-)
			Plot2 (-Mode Shape 2-)
			Plots (-Mode Shape 3-)
			Plots (-Mode Shape 5-)
			mile Hots (Hidde Shape 3-)
Stop Pause	Elapsed Time	00:00:07	



List Result - Mode Shape

COSMOSWorks >> List Results >> Modes



Ľ	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



Model name: Crank Study name: Freq Plot type: Frequency Plot1 Mode Shape : 1 Value = 170,77 Hz Deformation scale: 0,0314235





Procedure

Set a global Contact/Gaps options

- Mesh an assembly
- Run static analysis

Visualize the stress results





Imbalance.SLDASM

SolidWorks 2006 - [Imbalance.SLDASM]	
We File Edit View Insert Tools COSMOSWorks Window Help	×8-
	10 d d d d d d d d d d d d d d d d d d d
Borner (Ortal Classe State) Borner (Ortal Classe State) Comparison Comparison	
,	Editing Assembly 7

50-

Create a Study

Study >> Solid Mesh & Static





👒 省 😫 🚳
🕎 Imbalance
⊿=? x=? Parameters
🛓 ╃ ImBalance (-Default-)
🚊 🧐 Solids
🚊 🧐 Bearing-1
Body 1(Fillet1)
😑 🧐 Bearing-2
Body 1(Fillet1)
😑 🧐 Shaft-1
Body 1(Split Line1)
🛛 👯 Load/Restraint
🛄 Design Scenario
Report

Properties

ImBalance >> Properties >> FFEPlus Solver



tatic 🛛 🔀					
Options Adaptive Flow/Thermal Effects Remark					
Gao/Contact					
Include global friction Friction coefficient: 0.05					
Ignore clearance for surface contact					
Large displacement					
Solver					
O Direct sparse Use inplane effect					
○ FFE Use soft spring to stabilize model					
⊙ FFEPlus					
Results folder C:\Documents and Settings\Sontg\Desk					

Define Materials

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

COSMOSWorks Window Help		Matorial					
COSMOSWorks Window Help Study Advisor Loads/Restraint Drop Test Setup Result Options Contact/Gaps Shells	Apply Material to All Create/edit Material Library Update Components For All Studies Update All Components Customize Menu	Material Select material source Use SolidWorks material Custom defined Centor library From library files cosmos materials	Properties Material F Model T Units:	ables & Curves Fatigur roperties ype: Linear Elastic I SI	e SN Curves	•	
Run Plot Results		AISI 316 Sta	Name:	Alloy Steel (SS))		
Flot Results Ites and the second		→ AISI 4130 S → AISI 4130 S → AISI 4130 S → AISI 4340 S → AISI 4340 S → AISI 4340 S → AISI Type 3 → AISI Type 3 → AISI Type 4 → AISI Type 4 → AISI Type 4 → AISI Type 4 → AISI AISI Type 3 → Cast Alloy Steel (→ Cast Alloy S → Cast Carbor → Cast Stainle	Property EX NUXY GXY DENS SIGXT SIGXC SIGYLD ALPX KX C	Description Elastic modulus Poisson's ratio Shear modulus Mass density Tensile strength Compressive strength Yield strength Thermal expansion coe Thermal conductivity Specific heat	Value 2.1e+011 0.28 7.8999998e+010 7700.0001 7.2382562e+008 6.20422e+008 1.3e-005 50 460	Units N/m ² NA N/m ² kg/m ³ N/m ² N/m ² N/m ² /Kelvin W/(m.K) J/(kg.K)	Temp Dependency Constant Const
Export Import Motion Loads Options Help Research About COSMOSWorks Customize Menu	Plene2	Chrome Stail	q	K Cance	I Edi	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 Benott



Restraint - Immovable

Load/Restraints >> Restraints >> Immovable



👒 👕 😫 🞯			
Restraint			
Туре			
Immovable(No translation)			
Face<1> Face<2>			
Show preview			
Symbol settings			



Force - Normal to Plane

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



Symbol settings



Contact/Gaps

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





Create Mesh

Mesh >> Create >> 0.4 in









Run Static Analysis

ImBalance >> Run



Linear Static V2	.95(2005/180) -	C:\\Imba	lance	
	Iterative	Solver		
Nodes: 14240	Elements:	8583	D.O.F:	40650
Iteration:				
		70% (Completed	
				Þ
Stop Pause	PCG_Reset Convg	g.Plot Elaps	ed Time	00:00:03



Stress Plot

Model name: Imbalance

Imbalance Imbalance Imbalance Imbalance

- 🛓 🕂 ImBalance (-Default-)
 - 🛓 ആ Solids
 - 🖨 🍕 Bearing-1
 - Body 1(Fillet1) (-Alloy Steel (SS)-)
 - E K Bearing-2 Body 1(Fillet1) (-Alloy Steel (SS)-)
 - 🖶 🍕 Shaft-1
 - Body 1(Split Line1) (-Alloy Steel (SS)-)
 - 🛓 👯 Load/Restraint



- 🔀 Contact/Gaps (-Global: No penetration-)
- Mesh



Plot1





Deformed Shape

Stress Plot >> Edit Definition >> No Scale Factor



Create New Plot

Stress >> Define



🧐 🖀 🚳					
Stress Plot					
X ?					
Display					
🕒 VON: von Mises stress 🛛 👻					
F psi					
📔 Fringe 💌					
⊙ Node values					
O Element values					
Apply					
Deformed Shape Automatic:					
4151.55					
• Defined:					
2000					
Property					



Plot Setting

Setting >> No Boundary





🧐 😭 😭 N Imbalance Parameters 🖕 ≹ ImBalance (-Default-) 🛓 🚾 Solids | 🛓 🕌 Load/Restraint Restraint-1 👍 Force-1 - 🗰 Design Scenario 🔞 Contact/Gaps (-Global: No penetration-) Mesh Report 😑 📴 Stress Plot1 (-vonMises-) Plot2 (-vonMises-) Hide 🛓 📴 Displacement 🔁 Plot1 (-Res disp-Edit Definition... 🛓 📴 Strain Animate... 陷 Plot1 (-Equivaler Section Clipping... 🛓 📴 Deformation Iso Clipping... μt - 🔁 Plot 1 Chart Options... 🚊 🛅 Design Check Settings... Plot1 (-FOS-) Axes.... Probe List Selected Print... Save As ... Delete... Сору

Section Clipping





Model name: Imbalance Study name: ImBalance Plot type: Static nodal stress Plot2 Deformation scale: 2000 von Mises (psi) 1,270e+003 1,164e+003 1,058e+003 9,524e+002 8,466e+002 7,408e+002 6,349e+002 5,291e+002 4,233e+002 3,175e+002 2,116e+002 1,058e+002 5,050e-005

T THE T



Buckling Analysis

1 010011110000

1990000000

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



Material						
Select material source Properties Tables & Curves						
OUse SolidWorks material	O Use SolidWorks material					
O Custom defined	Model Tupe: Linear Flactic Lateria					
Centor library			sotropic			
• From library files	Units:	SI	~			
cosmos materials 💌	Category	Category: Steel				
🗈 AISI 316 Sta 🔨	Name:	Alloy Steel (SS)				
B AISI 347 An	Property	Description	Value	Units	Temp Dependency	
B AISI 4130 S	EX	Elastic modulus	2.1e+011	N/m^2	Constant	
- 🖻 AISI 4340 S	NUXY	Poisson's ratio	0.28	NA	Constant	
🖻 AISI 4340 S	GXY	Shear modulus	7.8999998e+010	N/m^2	Constant	
AISI Type 3 🗏	DENS	Mass density	7700.0001	kg/m^3	Constant	
AISI Type A	SIGXT	Tensile strength	7.2382562e+008	N/m^2	Constant	
ASTM AS6	SIGXC	Compressive strength		N/m^2	Constant	
- 🗈 Alloy Steel (SIGYLD	Yield strength	6.20422e+008	N/m^2	Constant	
- B Cast Allou S	ALPX	Thermal expansion cos	1.3e-005	/Kelvin	Constant	
🗠 🗈 Cast Carbor	KX	Thermal conductivity	50	W/(m.K)	Constant	
🗠 🗈 Cast Stainle	С	Specific heat	460	J/(kg.K)	Constant	
🕒 🖪 Chrome Stai						
Cancel Edit Help						

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

😵 😭 😫		
🎨 Housing		
Q=? X=? Parameters		
😑 科 Buckling (-Def	ault-)	
🛓 🔏 Solids	Run	
🚽 🍕 Housing	Run DesighScenario	
🛛 🗖 Body	Update All Components	
👝 🞎 Load/Restrai	Export	
🔤 🛒 Restrainl	Delete	
	Details	
Contact/Gan	Properties	
Mesh	Сору	
Report	Paste	
	Save all plots as JPEG files	
	Save all plots as eDrawings	

		Housing
Linear Analysis V2.95(2005/18	olver	Separameters
Nodes: 3302 Elements:	1560 D.O.F: 962	7
Establishing Geometric Stiffness:	100% Completed	Eestraint Eestraint-1 Eestraint-
Stop Pause	Elapsed Time 00:	00:01
		Plot1 (-Mode Shape 1-)

🤏 😭 😫 🞯



List Result - Mode Shape

COSMOSWorks >> List Result >> Modes



Li	ist Modes		×
1	Study name: Bud	kling	
	Mode No.	Buckling Load Factor	
	1	705	
	Close	Save	Help



Deformation Plot



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

🥵 😭 🈫							
Housing $x_{=?}^{=?}$ Parameters $x_{=?}^{2} O Buckling (-Default-)$							
😑 👒 Solids	What's wrong?						
⊟ - 🧐 Housing Info Boc Load/Restro Info Restrain Info Restrain Pressur	Run Run Design Scenario Update All Components Export						
	Details Properties						
Beport Benefit Plot1 (- Benefit Plot1 (Plot1 (Copy Paste Save all plots as JPEG files Save all plots as eDrawings						

🗲 Linear Analysis V2.9	95(2005/180	0) - C:\	Wousing	💶 🗆 🛛		
Sparse Solver						
Nodes: 6418	Elements:	3282	D.0.F:	18939		
,						
Decomposition of stiffnes	s matrix:					
		2:	2% Completed	1		
Stop Pause		E	lapsed Time	00:00:02		



List Result - Mode Shape

COSMOSWorks >> List Result >> Modes

1	ng.SLDPRT *]					
1	COSMOSWorks	Window	Help			
	Study				2 8 8	0
1	Advisor					11
4	Material			۲	et 🗟 🏘 🕯	
-	Loads/Resti	raint		۲		
-	Drop Test S	etup				
	Result Options					
	Contact/Gaps					
	Shells			×		
	Mesh			۲		
E	Run					
	Plot Results			×		
	List Results			F	Displacement.	
	Result Tools	5		F	Stress	
)	Report Compare Test Data			Strain		
			Modes	le la		
	Design Scenario 🔹 🕨		Thermal	.0		
•	Optimization			Therman		
	Fatigue 🕨			Customize Menu		
Ì	Parameters					
	Select All Fe	eature(s) F	aces			
	Export					
	Import Moti	on Loads				
	Options					
	Help 🕨					
	Research					
	About COSMOSWorks					
	Customize Menu					

List Modes 🛛 🔀					
Study name: Buc	kling				
Mode No.	Buckling Load Factor				
1	695.4				
Close	Save	Help			



Deformation Plot



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

