Judul Tesis

: STUDI EKPERIMEN DAN NUMERIKAL KUAT LEKAT TARIK TULANGAN POLOS DENGAN BETON

Nama Mahasiswa	:	Armeyn
Nomor Pokok	:	087 016 003
Program Studi	:	Teknik Sipil

Menyetujui, Komisi Pembimbing:

mm

(Prof. Dr. Ir. Bachrian Lubis, M.Sc) Ketua

(Ir. Daniel Rumbi Teruna, MT) Anggota

Dekan,

Ketua Program Studi,

(Prof. Dr. Ir. Roesyanto, MSCE)

Tanggal lulus : 11 Pebruari 2012

Bustami Syam, MSME)



Silinder Beton dan Pull Out Test



Penyetelan Alat Pull Out Test



Foto Setelah Dilakukan Percobaan Pull Out Test

Simulasi Beton Pull-Out Test

Percobaan Pull Out Test dengan program, ANSYS. Dapat terlihat sebagai berikut Dengan kondisi *symmetric test*, model yang riil dapat ditransfer ke FEM model dengan menerapkan unsur axisymmetric. PLANE82 unsur didalam ANSYS paket dipilih dalam kaitan dengan kemampuan khusus nya pada menghitung kondisi yang axisymmetric.

File Masukan: Pullouttest_Case_Axi.Inp

PullOutTest_Case_Axi - Notepad	
Eile Edit Format View Help	
!	^
! Numerical Simulation of the Pull-Out Test	
Disclaimer: User can modify this listing.	
Created on September 2nd, 2011 by Adriyan	
FINISH /CLEAR /FILNAME,Pull-Out Test Axi,1 /TITLE,Numerical simulation of the Pull-Out Test (Axisymmetric Analysis)	=
== PREPROCESSOR STAGE ==	
/PREP7 /UNITS,SI ! SI units ET.1,PLANE82 ! 2-D 8-hode Element KEYOPT,13,1 ! PLANE82 Element behaviour: Axisymmetric MP,EX,1,2.0298E+11 ! steel: Modulus of Elasticity MP,NUXY,1,0.29 ! Poisson's Ratio MP,EX,2,2.7337E+10 ! Concrete: Modulus of Elasticity MP,EX,2,2.045 ! Coefficient of friction	
! ! Modelling	
RECTNG,0,0.008,0,0.26 ! Area 1 RECTNG,0.008,0.075,0,0.16 ! Area 2 RECTNG,0.008,0.075,0,-0.14 ! Area 3 RECTNG,0,0.008,0,-0.14 ! Area 4 AGLUE,2,3,4 ! Glue Area 2,3,4	
NUMCMP,LINE ! Compress line numbering NUMCMP,AREA ! Compress area numbering /PNUM,LINE,1 ! Numbering the line LPLOT ! Plot the line	
! Line Division	
LESIZE, 1,0.002 ! Line 1, divided by element length 0.002 mm LESIZE, 2,0.004 ! Line 2, divided by element length 0.004 mm LESIZE, 3,0.004 ! Line 2, divided by element length 0.004 mm LESIZE, 3,0.004 ! Line 4, divided by 10 elements, and its space 1s 2 LESIZE, 6,0.004 ! Line 6, divided by 10 elements, and its space 1s 0.5 LESIZE, 5,0.004 ! Line 7, divided by 10 elements, and its space 1s 0.5 LESIZE, 5,0.004 ! Line 8, divided by 10 elements, and its space 1s 0.5 LESIZE, 8,0.004 ! Line 8, divided by 10 elements, and its space 1s 2 LESIZE, 5,0.004 ! Line 8, divided by 10 elements, and its space 1s 2 LESIZE, 10,,16,2 ! Line 9, divided by 10 elements, and its space 1s 2 LESIZE, 11,,16,2 ! Line 10, divided by 16 elements, and its space 1s 2 LESIZE, 12,,16,2 ! Line 11, divided by 16 elements, and its space 1s 2 LESIZE, 13,0.002 ! Line 13, divided by 16 elements, and its space 1s 2 LESIZE, 14,0.002 ! Line 13, divided by element length 0.002 mm	
! Generate Mesh	_
TYPE,1 ! Assign the element type 1: PLANE82 MAT,1 ! Assign the material type 1: Steel AMESH,1 ! Mesh the area 1 MAT,2 ! Assign the material type 2: Concrete AMESH,2,4 ! Mesh the area 2,3,4 End of File	=
!	~
	>

Semua modeling axisymmetric yang menghubungkan file di atas. Kemudian, kondisi kontak dibuat dengan menerapkan manajer kontak. Penting diingat, kontak dapat dilakukan setelah .

1. Membuka ANSYS File



Pilih tujuan nama file yang dimasukan, dan pilih jenis file ke ANSYS Perintah(*.*)



Pilih nama file yang dimaksud, klik Tombol Open dibawah ini

Open ANSYS F	iles	? 🔀
Look jn:	My Documents 💽 🗢 🛅 📰 🗸	
My Recent Documents Desktop My Documents My Computer	Downloads My Music My Pictures Numerical Simulation of the Concrete Reinforcement Pull PullOutTest PullOutTest_Case PullOutTest_Case_Auriter PullOutTest_Case_Avi Type: INP File Date Modified: 9/9/2011 10:05 AM Size: 3.23 KB	
My Network Places	File name: PullOutTest_Case_Axi	en
3000	Files of type: ANSYS Commands (*.*)	icei

Setelah membaca masukan file, di Program ANSYS workspace akan kelihatan seperti sebagai berikut

\Lambda ANSYS Multiphysics/LS-DYNA Utility Menu (Pull-OutTestAxi)						
<u>File S</u> elect List <u>P</u> lot Plot <u>C</u> trls <u>W</u> orkPlane	Parameters	<u>M</u> acro	Me <u>n</u> uCtrls	Help		
D 🗳 🔒 🔊 🎒 🖉 🗮					I 🛃	
ANSYS Main Menu 🛞					ANSYS	
 Preprocessor Solution 					SEP 9 2011 10:24:50	
General Postproc TimeHist Postpro Drop Test						
Topological Opt ROM Tool						
DesignXplorer ¥T Design Opt Design Design						
 Prob Design 						
🔜 Session Editor				Y		
Numerica	l Simulatio	on of t	he Pull-(Out Test (Axisy	mmetric Analysis)	

2. Untuk menghubungkan Kontak antara baja dan permukaan beton. Klik manajer

kontak



Klik contact wizard pada sisi kiri dari contact manager dialog



Kontak wizard Dialog tampak, tombol pada Permukaan Bentuk dan pada Target ke Fleksibel.

Contact Wizard		
	A contact pair consists of a target surface first define the target surface.	e and contact surface. You will
	< Back Next >	Pick Target Cancel Help

Klik Pick Target... Tombol dan Memilih Bentuk untuk Target Dialog nampak. Memilih Garis 8 Area 2 dan Garis 13 Area 4 dengan pengetikan mengedit kotak di bawah Daftar Materi Klik tombol



Klik Berikutnya > tombol pada Contact Wizard, Kontak Dialog. Kemudian, memilih baris pilihan di bawah Bidang-Kontak, dan Surface-To-Surface Pilihan di bawah Jenis Unsur Kontak.



Setelah meng-klik Pick Kontak... Tombol, Memilih Bentuk untuk Kontak akan nampak. Masuk baris 1 dan Garis 2 mengedit kotak di bawah Daftar Materi yang telah mengecek. Klik tombol

Select Lines for Contact
• Pick C Unpick
📀 Single 🔘 Box
C Polygon C Circle
C Loop
Count = 0
Maximum = 14
Minimum = 1
Line No. =
• List of Items
C Min, Max, Inc
OK Apply
Reset Cancel
Pick All Help

Klik Berikutnya> tombol pada Contact Wizard, Kontak Dialog. Pastikan bahwa pilihan Meliputi Awal Penetrasi telah dicek.

Contact Wizard		
	The contact pair is now ready to be created using the following settings:	
	Only Structural DOF has been detected	
	Friction:	
	Material ID 1	
	Coefficient of Friction 0.45	
	Thermal Contact Conductance 0	
	Electric Contact Conductance 0	
	Optional settings	\supset
	< <u>Back</u> <u>Greate</u> <u>Cancel</u> <u>H</u> elp	

Setelah meng-klik Pengaturan opsional... Tombol, Kontak Dialog akan memandu untuk menugaskan kekayaan kontak antar batang-baja dan beton Di basis dasar kekakuan hukuman yang normal antar batang-baja dan beton, ambil: 0.1. Di bawah Perilaku bidang-kontak, memilih item bersamaan dengan analisa pada daftar, berkas: yang terikat.

Contact Properties
Basic Friction Initial Adjustment Misc Rigid target Thermal Electric ID
Normal Penalty Stiffness 0.1 © factor © constant
Penetration tolerance 0.1 • factor C constant
Pinball region Constant
Contact stiffness update Each iteration (PAIR ID based)
Contact algorithm Augmented Lagrange method
Contact Detection On Gauss points
Behavior of contact surface Bonded
Type of constraint Auto assembly detection
OK Cancel Help

Pindah ke tab Friksi. Pilih kekakuan ke Unsymmetric Matriks Kekakuan Unsymmetric adalah dalam kaitan dengan friksi, jika permukaan bebas dari gesekan berubah untuk diabaikan. Kemudian klik OK.

Contact Properties
Basic Friction hitial Adjustment Misc Rigid target Thermal Electric ID
Material ID 1 1 Friction Coefficient 0.45
Tangent penalty Stiffness <auto> 💌 🕫 factor 🔿 constant</auto>
Allowable elastic slip 🛛 <auto> 💽 🕥 factor 🔿 constant</auto>
Contact cohesion 0.0
Maximum friction stress
Stiffness matrix
Static/dynamic friction
Static/dynamic ratio 1.0 Exponential decay coefficient 0.0

Klik Create > pada Contact Wizard

Contact Wizard		
	The contact pair is now ready to be created using the following settings: Only Structural DOF has been detected Create symmetric pair Include initial penetration Friction: Imaterial ID Image:	
	Coefficient of Friction 0.45	
	Thermal Contact Conductance 0	
	Electric Contact Conductance	
	Optional settings	
	< Back Create > Cancel Help]

Seperti itu, ANSYS workspace akan kelihatan seperti sebagai berikut.



Klik Tombol Terakhir pada Contact Wizard Dialog, dan menutup Manajer Kontak Dialog. Di menu Utility, klik Plot - Replot.

ANSYS Multip	hysics/LS-DYNA Utility A	ver
<u>F</u> ile <u>S</u> elect <u>L</u> ist	<u>P</u> lot Plot <u>⊂</u> tris <u>W</u> orkPlane	P
	Replot	
ANSYS Main Menu	Lines	
Preprocessor Solution	Volumes Specified Entities	
⊕ General Postp ⊕ TimeHist Post ⊕ Drop Test □ Topological Or	Nodes Elements Layered Elements	
 ➡ ROM Tool ➡ DesignXplorer ➡ Design Opt 	Materials Data Tables Array Parameters	
 Prob Design Radiation Opt Run-Time Stal Session Editor 	Results Multi-Plots Components	
	Parts	

3. Menetapkan syarat batas (B.C.) pada poros yang symmetric dan fixas bentuk.

Dalam kaitan dengan kondisi axisymmetric kasus percobaan, adalah baik untuk menetapkan B.C Yang Symmetric. dengan segera dan . Prosedur dapat dinyatakan sebagai mengikuti. Pada ANSYS Menu Utama, Klik Solusi- Menggambarkan Beban - Struktural-Penggantian- B.C Symmetric.- Pada Bentuk. Apply SYMM, masuk kan data baris 4 dan 10 Daftar Materi terpilih. Kemudian, klik OK.



Fixas diterapkan baris 9 dan 14. Di Program ANSYS Menu Utama, Klik Solusi-Menggambarkan Beban- Menerapkan- Struktural- Penggantian- Pada Bentuk.



Pastikan bahwa pada U, Bentuk Dialog, DOF yang dibatasi adalah Semua DOF dan nilai 0. Kemudian, klik OK.

Apply U,ROT on Lines	
[DL] Apply Displacements (U,ROT) on Lines	
Lab2 DOFs to be constrained	
	All DOF
Apply as	Constant value
VALUE Displacement value	
	Cancel Help

 Mecahkan masalah untuk memperoleh Penggantian Besar Statis (Langkah Beban 1) Di ANSYS Menu Utama, klik Solusi- Jenis Analisa- Sol'N Kendali.

<u>File Select List Plot PlotStrls</u>	WarkDlapp Devemptore Macro Macro Halp
	Solution Controls
ANSYS Main Menu	Basic Transient Sol'n Options Nonlinear Advanced NL
ANSYS Main Menu Preprocessor Solution Analysis Type New Analysis New Analysis Content of the solution New Analysis New Analysis Solution New Analysis New Analysis New Analysis Solution Solution Solution Solution Solution Solve Manual Rezoning Multi-field Set Up ANAMS Connection Diagnostics Multi-field Set Up ANAMS Connection Diagnostics Multi-field Set Up ANAMS Connection Diagnostics Multi-field Set Up Control Content Diagnostics Multi-field Set Up Control Content Diagnostics Multi-field Set Up Control Content Diagnostics	Analysis Options Small Displacement Static Calculate prestress effects Time Control Time at end of loadstep Automatic time stepping Prog Chosen Mumber of substeps Time increment Number of substeps Max no. of substeps Min no. of substeps O
Prob Design Padiation Opt	l
Run-Time Stats	OK Cancel Help

Pada icon Solusi Control basis dasar, memastikan pilihan sebagai di bawah ini

Analysis Optio	ons	Write	e Items to Results File All solution items	
T Coloral	ate predirece effects	0	Basic quantities	
Time Control	\sim	Noc Noc	User selected al DOF Solution al Reaction Loads	-
Time at end o	of loadstep	Eler Eler	nent Solution nent Nodal Loads	
Automatic tin Number o	of substeps	Eler	nent Nodal Stresses	-
C Time incr	ement	Writ	e every substep	
Number of su	ibsteps	w	nere N = 1	
Max no. of si	ubsteps 0			
Mirrio, or su	o o			

Pada ANSYS Toolbar, klik SAVE_DB untuk menyelamatkan;dan melanjutkan langkah beban selanjutnya



Mecahkan langkah beban ini.



Tunggu proses perhitungan hingga selesai dan klik Tombol ketika solusi dilaksanakan

Note		X
٩	Solution is done!	

dan juga / Status PERINTAH Dialog.

 Mecahkan masalah ketika beban yang diterapkan diperlakukan ke garis Area bagian atas 1 (Garis 3)- 2nd Langkah Beban. Di menu, klik Alur kembali



Workspace kelihatan seperti sebagai berikut



Armeyn

Universitas Sumatera Utara

Memberikan Beban ke 3, dalam hal ini adalah pada Menu Utama, Klik Solusi-Menggambarkan Beban- Struktural- Pada Bentuk.



Pada bentuk Dialog, masuk nilai Tekanan. Tanda tekanan positif menandai yang normal dan sebaliknya. Klik OK untuk menutup dialog

Apply PRES on lines	
[SFL] Apply PRES on lines as a	Constant value
If Constant value then:	
VALUE Load PRES value	-9.77e+7
If Constant value then:	
Optional PRES values at end J of line	
(leave blank for uniform PRES)	
Value	
ОК Арріу	Cancel Help

Di ANSYS Menu Utama, klik Solusi- Jenis Analisa- Sol'N Controls (Kendali).

Eile Select List Plot Plot⊆tr	
ANSYS Main Menu	Basic Transient Sol'n Options Nonlinear Advanced NL
🔚 Preferences	Analysis Options
Preprocessor	
Solution	Large Displacement Static 🔄 🔍 🐼 All solution items
Analysis Type	Calculate prestress effects G Basic quantities
📰 New Analysis	C Liser selected
Destant	
Sol'n Controls	Nodal DUF Solution
Denne Loaus	Time at end of loadsten 100 Element Solution
Load Step Upts	Element Nodal Loads
SE Management (LMS)	Automatic time stepping Off 🔄 Element Nodal Stresses 🗸
E Solve	Number of substeps Frequency:
Manual Rezoning	C Time increment
H Multi-field Set Up	Write every substep
ADAMS Connection	where N = 1
Diagnostics	Max no. of substeps 0
🔙 Unabridged Menu	Min no. of substeps
🗄 General Postproc	
🗄 TimeHist Postpro	
🕀 Drop Test	
🗄 Topological Opt	
ROM Tool	
🕀 DesignXplorer ¥T	
🗄 Design Opt	
🕀 Prob Design	
H Radiation Upt	OK Cancel Help
1	

Pada Kendali Solusi Dialog, basis dasar, menetapkan pilihan sebagai di bawah ini

Large Displacement S	Static	All solution items Basic quantities	
Fime Control Time at end of loadstep Automatic time stepping C Number of substeps Time increment Number of substeps		User selected Nodal DUF Solution Nodal Duf Footium Nodal Duf Solution Nodal Loads Element Nodal Loads Element Nodal Stresses Frequency: Write every substep	- -
Max no. of substeps Min no. of substeps		witere iv = 1	

Pada ANSYS Toolbar, klik SAVE_DB untuk menyelamatkan; melanjutkan 2nd langkah beban



Mecahkan 2nd langkah beban ini.



Tunggu proses perhitungan hingga selesai dan klik Tombol Close ketika solusi

dilaksanakan

N ote			×
i	Solution is done!		
		Close	

dan juga / Status PERINTAH Dialog.

6. Selanjutnya memproses

Memperluas model dari 2D-Axisymmetric kepada bentuk 3/4 perluasan

ANSYS Multiphysic	s/LS-DYNA Utility Menu (Pull-OutTestAxi)
Eile Select List Plot	Plototi WorkPlane Paran	neters Macro MenuCtris Help
╘╘	Pan Zoon Rotate View Settings	
ANSYS Main Menu	Numbering	
Preferences	Symbols Style	Hidden Line Options
Solution General Postproc	Font Controls	Size and Shape Edge Options
 TimeHist Postpro Drop Test 	Erase Options	Contours
⊞ Topological Opt ⊞ ROM Tool	Animate Annotation	Colors
DesignXplorer ¥T Design Opt Prob Design Radiation Opt	Device Options Redirect Plots	Light Source Translucency Texturing
Run-Time Stats Session Editor Finish	Save Plot Ctrls Restore Plot Ctrls Reset Plot Ctrls	Background ► Multilegend Options ► Floating Point Format
	Capture Image Restore Image	Displacement Scaling Vector Arrow Scaling
	Multi-Plot Controls Multi-Window Layout	Solid Model Facetor Symmetry Expansion Periods Civilia Symmetry
	Best Quality Image	2D Axi-Symmetric
	- Mume	rical Simulation of the User Specified spansion

∧ 2D Axi-Symmetric Expansion	$\overline{\mathbf{X}}$
[/EXPAND] 2D Axi-Symmetric Expansion	
Select expansion amount	
	C 1/4 expansion
	C 1/2 expansion
	© 3/4 expansion
	C Full expansion
	C No Expansion
Also reflect about x-z plane	∏ No
	CancelHelp

Model kelihatan seperti sebagai berikut



Di (dalam) ANSYS Toolbar, klik SAVE_DB



1St yang dibaca [mengisi/memuat] langkah, di (dalam) ANSYS Menu Utama



Rencanakan Y-Component Tekanan (σ yy)

ANSYS Main Menu	▲ Contour Nodal Solution Data	
 Preferences Preprocessor Solution General Postproc Data & File Opts 	Rem to be contoured	-
 Results Summary Read Results Failure Criteria Plot Results 	Beress M-Bernportenk of stress fr Component of stress fr Component of stress fr Component of stress	_
Deformed Shape Contour Plot Nodal Solu Etement Solu Etement Solu	 W Shear stress Y Shear stress X Shear stress X Shear stress 	
Etem Table	 Ist Principal stress 2nd Principal stress 	•
Loncrete Plot ThinFilm List Results Query Results	Undisplaced shape key Undisplaced shape key Deformed shape only	<u> </u>
Uptions for Outp Results Viewer Write PGR File Nodal Calcs	Scale Factor True Scale	8
Element Table	OK Cancel	Help



Rencanakan X-Component Tekanan (σ xx)

\Lambda Contour Nodal Solution Data				
Item to be contoured				
😪 Favorites 💋 Nodal Solution	-			
M DOF Solution				
Stress				
X-Component of stress				
2-Component of stress				
🔗 XY Shear stress				
😥 YZ Shear stress				
2 Az Shear stress				
Undisplaced shape key Undisplaced shape key Scale Factor True Scale	<u> </u>			
Additional Options	۲			
OK Apply Cancel H	elp			



Rencanakan XY-COMPONENT Tekanan (T xy)

▲ Contour Nodal Solution Data	×
Item to be contoured	
Pavorites Solution Constant of stress Component	-
1st Principal stress	<u> </u>
Undisplaced shape key Undisplaced shape key Scale Factor True Scale	
Additional Options	۲
OK Apply Cancel	Help



yang dibaca 2Nd Langkah Beban Substep no.. 1

ANSYS Main Menu	
📰 Preferences	\Lambda Read Results by Load Step Number 🛛 🛛 🛛
Preprocessor Solution	[SET] [SUBSET] [APPEND]
🖃 General Postproc	Read results for Entire model
🧱 Data & File Opts	Endre moder
🚍 Results Summary	LSTEP Load step number
Read Results	
📰 First Set	SBSTEP Substep number
📰 Next Set	E0/TL Scale Factor
📰 Previous Set	
🔜 Last Set	
By Pick	
🔚 By Load Step	
By Time/Fred	OK Cancel Help
🔤 By Set Number	
FLOTRAN 2.1A	







Substep no. 2

ANSYS Main Menu	
Preferences	\Lambda Read Results by Load Step Number
Preprocessor Solution	[SET] [SUBSET] [APPEND]
🖂 General Postproc	Read results for
📰 Data & File Opts	
🔤 Results Summary	LSTEP Load step number
Read Results	
📅 First Set	SBSTEP Substep number
📰 Next Set	EACT Scale factor
Previous Set	
📰 Last Set	
Py Pick	
📰 By Load Step	
By fime/Freq	OK Cancel Help
🔤 By Set Number	
FLOTRAN 2.1A	

Dan mengikuti prosedur yang sama, seperti halnya, substep dan tegangan.

MOTTO DAN PERSEMBAHAN

Motto:

- Jadikanlah setiap desah nafas dan langkahku dalam kehidupan sebagai Ibadah yang terindah kepada Allah, ingin selalu kuniatkan segalanya karena ALLAH.
- Kumohon Ampunan Dosaku dan Dosa Kedua Orang Tuaku, kuingin dalam setiap kehidupanku, keberadaanku tidak menjadi beban bagi siapapun. Cukuplah beban itu kusandarkan pada ALLAH
- Dan kuingin, Allah ciptakan keberadaanku di muka bumi ini sebagai berkah, manfaat dan sebagai pembawa kebaikan.
- 4. Ya Robbi, kabulkan permintaanku itu.

(Penulis)

Persembahan:

Tesis ini kupersembahkan untuk:

Semua orang yang membutuhkan sebagai Akademisi dan Praktisi