SOUTHWEST RESEARCH INSTITUTE

Div 20 Work

Chris Waldhart
(210) 522 4796

CNWRA
CONTROLLED
COPY 601

TABLE OF CONTENTS	PAGE No.
Title of Analysis: Validation of LS-Dyna and ABAQUS/Explicit	
Names of Individuals Performing Activity	
PA Cox	
Erick Sagebiel	
AH Chowdhury	
Chris Waldhart	
Objectives: The objective is to validate features of LS-Dyna and ABAQUS/Explicit used in previous simulations by comparing the results of the test cases outlined in the initial validation document	
Special Personnel Training or Qualification: Knowledge of ABAQUS/Explicit and LS-Dyna and how they were used in previous simulations.	
Equipment: Personal computers	
Chris Waldhart ; June 24, 2002	

Work continued from Page

Division 20 analyses - (June 24)

odd behavior/differences noted in case 1.12 (triangular pressure, gravity case)

want to see if plasticity component is to blame

→ abaq_cl.12elastic.inp run to verify this

June 25, 2002 (SW

Division 20 -

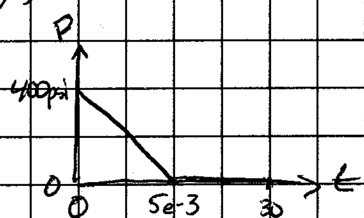
discovered error in model abaq_cl.11 → missing clamped BC

updated BC + rerun as abaq_cl.11b.inp

loads track reasonably well initially but vary slightly after significant yielding occurred

top 0° disp tracks w/in 5% or so

modifying cl.11b so that load curve looks like



saved model as abaq_cl.11c.inp

Work continued to Page

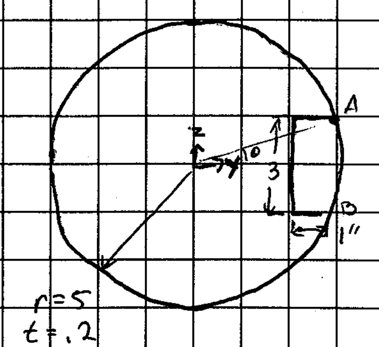
Work continued from Page

June 29 2002 CJW

need to develop sample channel-cylinder interaction model for case 2 geom

Goal → 1) determine how to model contact between surface + edge of surface
2) determine clearance issues at interface

model: c:\projects\cnwra_abaq_dyna_vr\case_2\channel_cyl_ex



pt A define θ $\theta = \sin^{-1}\left(\frac{1.5}{r}\right)$

let $r = 5$ (center of cylinder)

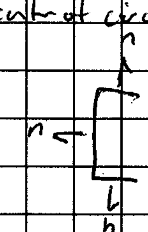
$\Rightarrow \theta = \sin^{-1}\left(\frac{1.5}{5}\right) = 17.4576°$

$A(y,z) = (r\cos\theta, r\sin\theta)$ (CJW 6-29-02)

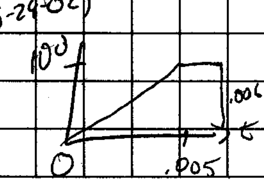
$= (4.769696, 1.5)$ (CJW 6-29-02)

$B = (4.769696, -1.5)$

normals: cylinder → outward radially
channel → out towards center of circle



BC: clamp base (x=0) of both cyl + channel (CJW 6-29-02)
~~move top of channel by 1" in +y direction~~
apply 100 psi pressure to channel long fun



Work continued from Page

[redacted]

Result → BIG Problem

apparently Abaqus tries to adjust surfaces on Every contact surface
- in this case my mesh gets extremely distorted

put in a call/~~to~~ email to Abaqus to see if they can help out here

July 25/26 2002 CJW

appears that for case 1, the primary variation in Dyna sol'ns are largest wrt integration scheme - maybe not so much if # of pts ≥5

~~want to see effect of the dyn~~ (CJW 6-25-02)

convergence study on case 2.3b
- examine effect of # of int pts + int scheme on solution
- files located at D:\cnwra_abaqus_dyna_validation\case_2\c2-3\

model	int scheme	section pts	
c2.3b.inp	Simpson	5	← elset missing, didn't run
c2.3c.inp	Simpson	5	disp are pretty much ~~constant~~ converged
c2.3c.9.inp	Simpson	9	
c2.3c.5.gauss.inp ~~c2.3c.?.inp~~ (CJW 6-15-02)	Gauss	5	

re-ran case c1.11b w/ Gauss integration
c1.11b → simpson w/ 5 section pts

abaq.c1.11b.gauss.inp → gauss integration w/ 5 pts

⇒ results agree much better w/ Dyna fully-integrated shell results

Work continued from Page

Abaqus Shell Element

S4R is a general purpose shell that can approx thick shells that are shear flexible elements (Mindlin) or thin shells (Kirkchoff)

- bending strain approximate Koiter-Sanders shell theory
- thickness change as a fn of in-plane loading
- do not suffer from transverse shear locking or hourglassing

- thickness change def

$$t/t_0 = (A/A_0)^{-\frac{\nu}{1-\nu}}$$

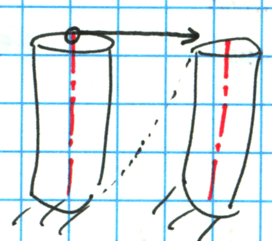
where $\nu = 0.5$ for incompressible and $\nu = 0$ then thickness doesn't chge

- transverse shear treatment
  problem of shear locking goes away as $t \to 0$
  use a Hu-Washizu

-1st order transverse shear flexible theory
- transverse shear strain is constant through the thickness
- correction of 5/6 used for isotropic plates

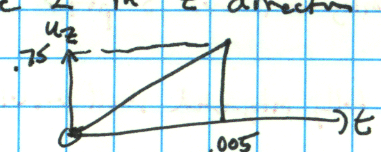
Aug 7, 2002 (JW

Pre-Case 3 work
doing contact w/ 2 beams



beam properties
~~.25" diameter~~ ((JW 8-7-02)
.25" diameter (outer)
.10" diameter (inner)

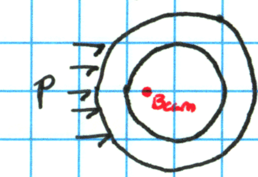
disp on node 2 in z direction



Work continued from Page

Beam to Beam contact in Abaqus
beams can be used in contact analyses **BUT** they must be the **SLAVE** surface (insufficient info to define normals)
as a result of this, there **CAN NOT** be beam to beam contact

Options:
1) modify case 3 to have 1 beam inside a cylinder



[redacted]

2) goto case 4
3) goto case 5

My opinion is that case 5 is easier and makes more sense as it is all solid elements (except for rigid block of course) and doesn't have contact

actually, we probably should have started w/ solid elements as they don't involve all the assumptions that shell elements have
- this would/may have made to solver comparison component "cleaner"

⇒ wait on Erick to review his stuff and he'll get back w/ me once it is finished

Work continued from Page

Aug 8 2002 CJW

modification to case 3 recognizing that beam-beam contact isn't possible
Case 3.0b → 1 beam inside 1 cylinder

elset 4 → loading face (pressure loads)
nset 5 → cylinder base
nset 3 → cyl top node

t	p
0	~~500~~ 600 (CJW 8-8-02)
.005	~~250~~ 300
.010	0
1	0

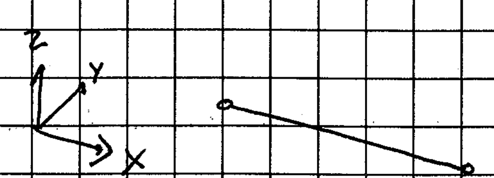
properties:
cylinder $t = 0.20$ in
$\bar{r} = 2.9$ in

beam OD = .25 in
ID = .10 in

$r = .125$ in
$t =$ ~~.0125~~ .075 (CJW 8-8-02)

elastic plastic ($\sigma_y = 45$ ksi)

*beam section, section = pipe, material = ,
elset = ,
r, t



t vector goes from 0 to 1 in x dir
n_1 goes in y dir
n_2 goes in z dir

*surface interaction
*contact pair, interaction = ___, weight = 1.0
s1, s2
master slave

Work continued to Page

SIGNATURE | DATE

DISCLOSED TO AND UNDERSTOOD BY | DATE | WITNESS | DATE

Work continued from Page

called Abaqus South Central [214-513-1600] to ask how to define beam / shell contact in Abaqus/Explicit

spoke w/ ~~Ganesh~~ Sarvesh (CJW 8-8-02) – not much headway

Aug 13, 2002 CJW

modify properties on beam to $r_o = .125$, $r_i = .05$

$r_o = .125$
$t = r_o - r_i = .125 - .05 = .075$

mod. as per pg 6

shorten the spr to .004 [redacted]

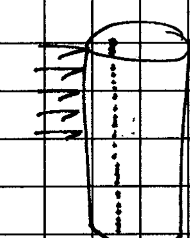
→ filename: case3.0b.v2.inp

Aug 15, 2002 CJW

case5.0b → double check load curve definition from base Excel spreadsheet
looks like I missed some data pts –
comp looks pretty good even w/ messed up loading

LS Dyna damping → use mass weighted damping either by nodes or part
use Rayleigh stiffness damping (values between 0.01 and 0.25 suggested)
$C = \alpha M + \beta K$
use damping relative to rigid body only

Case 3.0b



initial offset on when shell contacts beam

- see if Abaqus can go to mid-plane of shell on contact
- run model w/o beam contact ⇒ $\boxed{\text{case 3.1a}}$
  (Dyna runs don't appear to have diff shell responses even though the contact occurs 0.2" later in one of the models)
  time ends at .003 seconds

Work continued to Page

SIGNATURE | DATE

DISCLOSED TO AND UNDERSTOOD BY | DATE | WITNESS | DATE

Work continued from Page

Abaqus damping methods

provides Rayleigh damping for cases when ~~dissipation~~ energy needs to be dissipated

2 factors are defined: α_R = mass proportional damping

β_R = stiffness proportional damping

α_R → damping forces caused by absolute velocities of model considered

sort of like moving body through viscous "ether"

units 1/time

β_R → damping proportional to strain rate

damping proportional to elastic material stiffness

$\sigma_d = \beta_R [D^{el}] \dot{\varepsilon}$ where D^{el} is current elastic stiffness

generally speaking: mass prop damping damps out low freq ~~vibration~~ response (CJW 8-15-02)

stiffness prop damping damps out high freq response

Aug 16, 2002 CJW

case 5 mass damping

file	α_R $(\frac{1}{sec})$
case 5.1a	.0001
case 5.1b	.001
case 5.1c	.01
case 5.1d	.1
case 5.1e	1.0

case 5 stiffness damping

file	β_R (sec)
case 5.1f	7.5e-7 (critical stable time increment from case 5.1a)
case 5.1g	1.5e-6
case 5.1h	1.5e-5

SCIENTIFIC BINDERY PRODUCTIONS CHICAGO 60605 MADE IN USA

Work continued to Page

SIGNATURE DATE

DISCLOSED TO AND UNDERSTOOD BY DATE WITNESS DATE

Work continued from Page [redacted]

Aug 20, 2002 CJW

case 3.1b → load curve definition

t	P
0.0	20
5e-3	10
10e-3	0
2	0

t_{run} = .038"

time to run to 0.005 seconds

case 5 → run out to 1 sec or so for mass/stiffness damping

still elastic

load case is "C"

t	u_x
.0007	.16
.0014	.32
.0021	↓

← coarse model

file	α_R	β_R	t_{end}
case 5.1i	.01	0	1.00
case 5.1j	0	0	1.00
time step too small due to β_R value → case 5.1k	0	1.5e-6	1.00
case 5.1l	0	7.5e-7	1.00

[redacted]

Aug 21, 2002 CJW

case 3.2a → motion in z direction for base (email is incorrect)

beam nodes → 7777 to 7807

modify z coord on all nodes to be 0.0

make od = .40"

id = .20

SCIENTIFIC BINDERY PRODUCTIONS CHICAGO 60605 MADE IN USA

Work continued to Page

SIGNATURE DATE

DISCLOSED TO AND UNDERSTOOD BY DATE WITNESS DATE

Work continued from Page

modifications to case 5.1

need more data pts to define history response

• reduce step time to 0.25 sec (from 1.00)

• history output every .000025 sec (every 22 inc of time step - should be reasonable)

file	α_0	β_0	t
case 5.1j	0.0	0.0	0.25
case 5.1i	.01	0.0	0.25
case 5.1l	0.0	7.5e-7	0.25

Aug 22 2002

if we could start over, what would I do...

1) element verification

start w/ simple model of a cylinder modeled by (confident that each answer is correct)

i) solids

ii) shell

iii) beam

elastic material

prescribed motion at base or tip going through fairly large motion

2) material validation

~~stage~~ single element verification (shell, solid, beam)

• piecewise linear plasticity model

• kinematic hardening

• other

3) loading conditions

take models in 1 and apply various loading conditions individually

• pressure

• ax load

• gravity

Work continued from Page

4) add plasticity models from (2) to (1)

5) add plasticity models from (2) to (3)

6) contact simulations

solid to solid

solid to shell

solid to beam

shell to beam

shell to shell

CJW

Aug 23, 2002 CJW

[redacted] write up

case 1 - looks like we have large differences in sol'n

case 31.b - when we thin up the cylinder, get excellent solution

⇒ that is critical to our assessment of shell elements

case 2 →

(CJW 8-23-02)

• run case 5.1 w/ refined mesh out to ~ ~~.4~~ .5 seconds

CJW

Aug 26, 2002

resubmittal fine v5.1 mesh (modified case5.06.inp to impulse case)

↑

~~case5.1a~~ (CJW 8-26-02)

case5.1ref

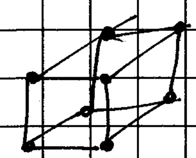
CJW

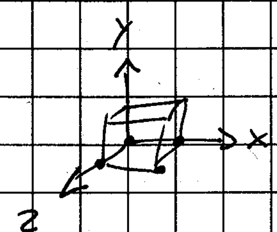
Work continued from Page

Aug 28, 2002 [redacted]

what's left to do

material model exploration





	x	y	z	u_x	u_y	u_z
1	0	0	0	0	0	0
2	1	0	0		0	0
3	1	0	1		0	
4	0	0	1	0	0	
5	0	1	0	0	δ_y	0
6	1	1	0		δ_y	0
7	1	1	1		δ_y	
8	0	1	1	0	δ_y	

Use curve from case 5 scaled by 0.3

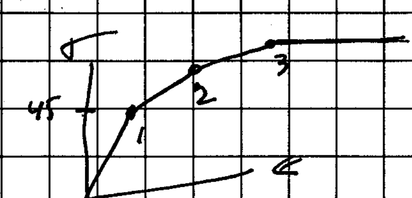
$\delta_y(t) = .15 \sin(500t)$

send single load step

t defined by increments similar to 5.0a (same loading of bar)

material model
I) elastic ~~perfectly plastic~~ (JW 8-28-02) (same as before $E = 30e6$ $\nu = .3333$)
II) elastic perfectly plastic $\sigma_y = 45000$
III) kinematic hardening

$(\sigma_y)_{eng}$	ϵ_{eng}
45000	45000/30e6
50000	.05
55000	.20



Work continued to Page

Work continued from Page

need to convert stress/strain to true stress and logarithmic plastic strain

σ_{nom}	ϵ_{nom}	σ_{true}	ϵ_{ln}^{pl}
45000	$\frac{45000}{30e6}$	45067.5	0.0
50000	.05	52500	.0470
55000	.20	66000	.1801

$$\sigma_{true} = \sigma_{nom}(1 + \epsilon_{nom})$$

$$\epsilon_{ln}^{pl} = \ln(1 + \epsilon_{nom}) - \frac{\sigma_{true}}{E}$$

CJW

Aug 29 2002
case 7 - material model tests on single shell element
case 7.1 - ~~..~~ cyclic axial loading of elastic shell
$t = 0.1$
sine wave form from case 6 ~~...~~
same disp magnitude as in case 6
top-node = node 2

case 7.2 - uniaxial extension w/ elastic, perfectly plastic material
$\sigma_y = 45000$ psi

case 7.3 - modify case 7.1 to apply cyclic rotation on node 2+4
make rotation = 30° (= .5236 rad)

$.5236 = .5 \sin(500t)$

$\frac{.5236}{.5} = 1.0472$ scale applied to bc

case 7.4 - modify 7.2 with ↗ for x rotation on nodes 2+4

Work continued to Page

Work continued from Page

case 7.5 - increase thickness in 7.3 to 0.2"
apply same bending rotation

case 7.6 - increase thickness in 7.4 to 0.2"
apply same bending rotation

case 7.7 - decrease thickness in 7.3 to .05"
apply same bending rotation

case 7.8 - decrease thickness in 7.4 to .05"
apply same bending rotation

CJW

Sept 4, 2002
modified amplitude on 7.4 to be 0.3 to match Erick (who wasn't sure where 1.0472 had come from)
also modified 7.6 + 7.8

CJW

Sept 11, 2002
Case 1 observ. → add time step study

CJW

Work continued to Page

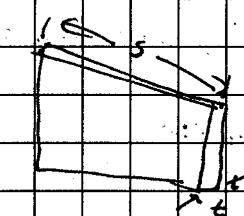
Work continued from Page

Sept 12, 2002
damping - target some % of critical damping w/ our inputs
overview → gives idea of scatter that may appear in between the 2 codes
something we can use in the future when considering design constraints
just keep going w/ case 4 as scoped out
total package will be complete - everything covered

• check on spots welds on Case 2
- give a shot again to see if we can get it going

• case 1 - finished

• case 3 - smoothly running stuff + resubmit
update tutorial problem
might have poor aspect ratio



want $t/s \approx 1/5$ or so

when we went to $t = .038$, you get $\frac{t}{s} = \frac{1}{5}$
$t = .2$, you get $t/s = 1$ } inner cylinder

double check case 1 to see what happens as aspect ratio changes towards $\frac{1}{5}$ or so

• case 5 - calculate damping
filter out high frequency

CJW

SCIENTIFIC BINDERY PRODUCTIONS CHICAGO 60605 MADE IN USA
Work continued to Page
SIGNATURE | DATE
DISCLOSED TO AND UNDERSTOOD BY | DATE | WITNESS | DATE

Work continued from Page

Sept 16, 2002

case 2 – generating spot welds

spot welds:

- only defined at slave nodes for pure master-slave contact pair
- rotational effects not considered/transfered between bodies

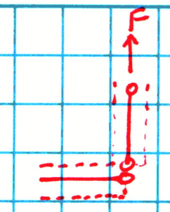
how to do this

- define edge of C shell to be a nodal group: c_node

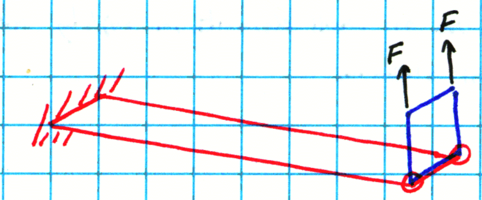
```
*surface, name=cyl-mstr, type=element
cylinder, sneg
*surface, name=~~c-node~~ c-slave, type=node (CJW 9-16-02)
c-node ← this should be entire slave group (channel in this case)

*contact pair, weight=1.0, interaction=spt-weld, mechanical ~~[illegible]~~ constraint = kinematic (CJW 9-16-02)
cyl-mstr, c-slave
*surface interaction, name=spt-weld
*bond
c-node
```

file case2\ channel-cyl-ex-spt-weld.inp

Work continued from Page



Need to check spot weld implementation



- 10" × 1" wide by 0.1" thick (1 elem)
- 1' × 1" wide (1 elem)

spot weld properties

- $F_n = \frac{9}{10} F = 3.375$
- $F_s = \frac{9}{10} F = 3.375$
- d_b = bead size = 0.0
- T_f = 1.0 sec

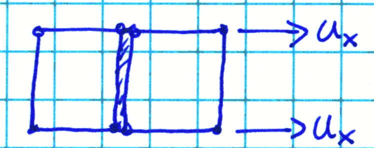
calc F: $\sigma = \frac{(2F \cdot L)(\frac{1}{2})(0.1)}{\frac{1}{12}(1)(.1)^3}$

$\sigma =$ ~~45 ksi~~ 45 ksi

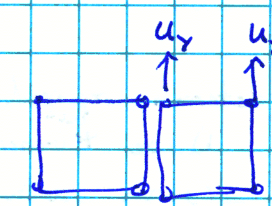
$\Rightarrow F = 3.75$ lbs

possible simplistic models to check out

1)

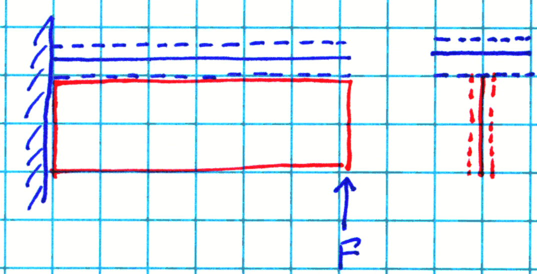


2)



⇐ no, don't like

3)



CJW

Work continued from Page

Oct 1, 2002

mesh refinement –

concern exists over the applicability of the use of shell elements in highly refined meshes → element length ≈ element thickness

does this exceed good engineering approx?

Abaqus / Explicit

↳ $\dfrac{K_{\alpha\alpha} \, l^2}{D_{(\alpha+3)(\alpha+3)}} > 100$ [limit on applicability]

$\alpha = 1, 2$ (not summed)

assumptions: (linear elastic matrix)

$K_{\alpha\alpha}$ = transverse shear stiffness

✱ ⟹ l = characteristic length ≠ element's characteristic length

$D_{\alpha\beta}$ = section stiffness matrix

to determine required input

1) use *Shell General Section, Composite option

2) run datacheck analysis

mesh refinements to be examined

1) 10×20 mesh

2) 80×96 mesh

3) 160×192 mesh

Work continued from Page

new directory created

e:\chwra_abaqus_dyna_validation\case_1\shell_validity_check\

cl_10by20.inp

cl_80by96.inp

cl_160by192.inp

cl_10by20 results

$(K_{ts})_{11}$ = 1.875 e6

$(K_{ts})_{22}$ = 1.875 e6

D_{44} = 22494

D_{55} = 22494

since l is a fn of model geometry only, focus on stiffness fact.

$\dfrac{K_{\alpha\alpha}}{D_{(\alpha+3)(\alpha+3)}} = 83.4 > \dfrac{100}{l^2}$

$l > 1.095$ in ($t = .2$ inches)

this ends up being a very thick cylinder $D/t = 5.5$

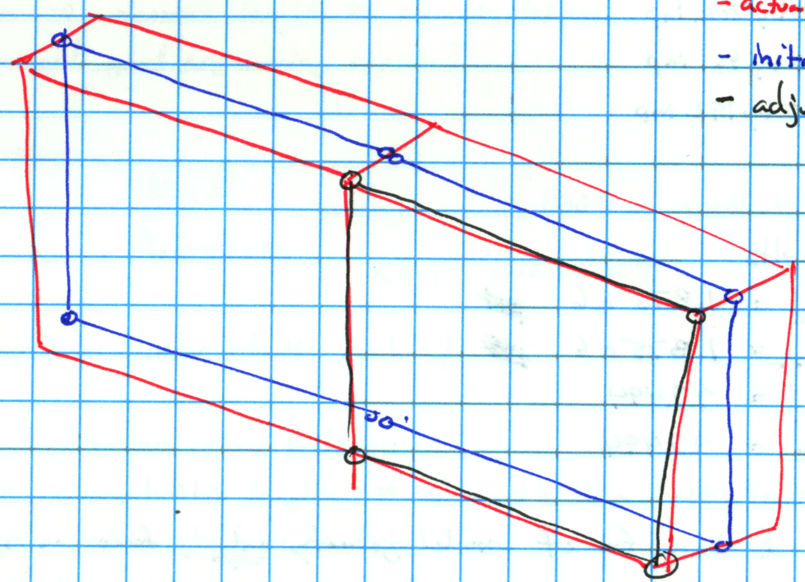
K + D values identical for 160×192 mesh

As anticipated validity of shell formulation is not affected by mesh discretization for elastic material

Work continued from Page

spot weld work

looking at 8.0 (2 elements w/ spot weld joint + axial beam) there is an offset problem



- actual geometry
- initial shells + nodes
- adjusted slave surface (flush w/ spos on master surface

case 8.0b.inp

- offset master element nodes to SPOS surface (hopefully get everything in a line after adjustment)
  - define master nodes to be a SPOS geometrically
  - offset el 1 to SPOS

- aligns 2 spot weld surfaces (sort of) but the loaded nodes are no longer in line

case 8.0c also has issues

Work continued to Page

SIGNATURE DATE

DISCLOSED TO AND UNDERSTOOD BY DATE WITNESS DATE

Work continued from Page

need to call Abaqus

- can I perform a spot weld analysis for the case where I have a shell perpendicular to another

CJW

Oct 2, 2002

add large mass damping factor to case 5

$\alpha_R = 314.0$

⇒ case 5.1m.inp

base time increment = 7.4623e-7

total analysis time = 0.25

10000 pts output ⇒ time interval = .000025

executed w/ double precision solution

email sent to Abaqus regarding spot weld concerns - waiting reply

Work continued to Page

SIGNATURE DATE

DISCLOSED TO AND UNDERSTOOD BY DATE WITNESS DATE

Work continued from Page

Thick vs Thin Shell Solutions

→ elastic : $E = 30e6$, $\nu = .333$

make a ring with ~~mean diameter~~ following geometry

- height = 1.0" t = ~~0.5, 1.0~~, .05, .1, .5, 1.0
- ID = 10.0" (ISO ...)   OD = 10.1, 10.2, 11, 12

apply an ~~external~~ internal pressure in slow manner (quasi-static)

compare solution to hand calc

- focus on stress variation through thickness
- focus on strain variation through thickness

directory = thick_thin_shell

model	t	$\bar{r}$ $(=\frac{1}{2}(ID+t))$	OD/t
tts_1	.05	5.025	202
tts_2	.1	5.05	102
tts_3	.5	5.25	22
tts_4	1.0	5.50	12

need to model closed pipe

- may have issues w/ single element axially if go to equivalent line load for thrust
- put hemiheads on model to "accurately" capture thrust loads
- make model 6 elements axially on cylindrical portion

estimated pressure

$\sigma = 50000$

tts_1 → p = ~~498 psi~~ 500 psi
tts_2 → p = 1000
tts_3 → p = 5000
tts_4 → p = 10000

Work continued from Page

need to redo the model

- make nodes on inner surface
- outward normals
- offset = Sneg

⇒ only need to change t on shell section for 4 models

CJW

Oct 23, 2002

run tts_3 w/ plasticity (no strain hardening)

σ_y = 50000 psi

t_{max} = .015

amp curve = unchanged

tts3_plasticity.inp ← input deck

yield stress too high (forgot ax tension component)

$$\sigma_y = \frac{\sqrt{3}}{2}\sigma_y^{old} \text{ (von mises)} = 43300 \text{ psi}$$

speed up analysis

t_{max} = .0075
end press @ .006 sec

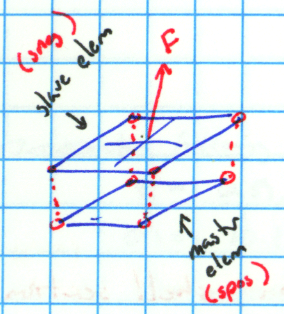
CJW

✱ previous case 8 models moved to initial models - edge - edge - contact directory

Work continued from Page

Oct 4, 2002

case 8.0 model



o spot welds

t = .50"

normal + shear failure force on spot welds = ~~0.5~~ 1.0 lbs

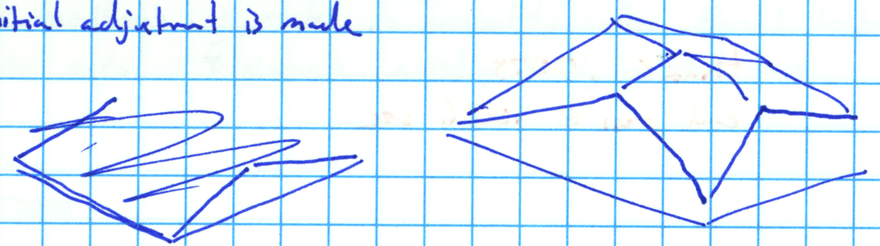
applied force F

t	F
0	0
1	~~4.0~~ 4.10
2	4.10

total duration considered = 1.5 seconds

problem w/ spot welds

the use of ~~fee~~ nodal contact pairs instead of surfaces w/ shell elements results in formation of a tent like structure after initial adjustment is made



need to modify model some what

Work continued to Page

SIGNATURE DATE

DISCLOSED TO AND UNDERSTOOD BY DATE WITNESS DATE

Work continued from Page

modifications made to 8.0.1.inp

- move slave element nodes flush w/ top of master elements
- offset reference plane using sneg
- increase spot weld strength to ~~1~~0 000 lbs (from 1) ← get larger deformations

remove for the being ✗ ~~repeat analysis~~

~~delete - want to be able to track plate disp after failure~~

~~• add 1 second time to failure~~

~~modify amp curve (0,0) (1, 4.10) (~~

modify the scale to ~~1.0~~ 1.00

modified shear force in spot weld and got "good" answer (separation when applied load = 4 × F_{normal} for spot weld)

~~(8.0.2 → go to shear condition (apply F_y) ∞ F_{normal} 1.0 F_{shear}~~ ((JW 10-4-02)

CJW

Work continued to Page

SIGNATURE DATE

DISCLOSED TO AND UNDERSTOOD BY DATE WITNESS DATE

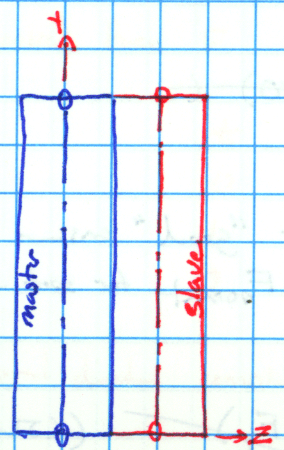
Work continued from Page

Oct 7, 2002

unresolved issues w/ spot weld adjustments in v6.2

need to determine how adjustments are made

case	Z_{master}	adjustment of slave nodes	adjustment ⇓ my theoretical shift
full offset (no overlap)	0.0	.15	.25
no offset (centerlines flush)	0.5	.35	.25
full offset (no overlap, SNEG side)	0.0	.85	.75



Abaqus will automatically move slave nodes to free surface of master element

why doesn't this appear to be the case?

looks like it thinks the master shell is 0.70" thick not the 0.5" input

if I change both shell thicknesses to ~~0.25~~ ~~0.2~~ 0.3", keep the center of master el @ z=0, the adjustment is 0.25" when it should be

0.5"(slave node) − ½(0.3")(master pos surface) = .35

keeping ↑ unchanged, I make SNEG ref surface

adjustment → 0.75"

"should have been" → 0.5 − (−½)(.3) = .65

Work continued to Page

SIGNATURE DATE

DISCLOSED TO AND UNDERSTOOD BY DATE WITNESS DATE

Work continued from Page

for z=0 on master ele

~~t shell | ref surface | ref surface | adj ABQ~~ adjustments → my

~~0.5 pos neg~~

~~0.3~~

t shell	master elem ref surface	adjustments Abaqus	my
0.5	pos	.15	.25
0.5	neg	.85	~~.28~~ .75
0.3	pos	.25	.35
0.3	neg	.75	.65

in each case, the Abaqus adjustment was

0.1" < my prediction on pos side

0.1" > my prediction on neg side

?

send summary off to Abaqus again to see if they have any suggestions

(hotline@abaqus-sc.com @ 7:01 am 10-7-02)

→ the culprit was the beadsize was set to 0.1"

if this was set to 0.0, everything would be fine I believe

other modifications

1) make slave surface the entire shell elements (not just corner nodes)
2) try setting d_g = 0.0 (might adjust the bed)
3) time to failure = 0.0 seconds

} completed summary on next page

Work continued to Page

SIGNATURE DATE CJW (10/7/02)

DISCLOSED TO AND UNDERSTOOD BY DATE WITNESS DATE

Work continued from Page

Oct 8, 2002

case 8.0.inp

elastic (30e6, 0.333)

surface-surface contact w/ slave corner nodes spot welded

$F_n = 1.0$ lbs

$F_s = 1{,}000{,}000$

$d_b = 0.0$

$T_f = 0.0$

z ~~disp~~ force at center of slave elements

t	F
0	0
1.0	4.10
2.0	4.10

analysis run to 1.2 seconds

case 8.1.inp (shear loading)

modify 8.0.inp so that → $F_n = 10{,}000{,}000$ lbs

$F_s = 1.0$ lbs

z force becomes positive y force

case 8.2.inp (normal + shear)

modify 8.1.inp so that $F_n = F_s = 1.0$ lbs

applied force in y + z directions $= 2.828$ $(= \frac{4}{\sqrt{2}})$

CJW

Work continued to Page

SIGNATURE DATE

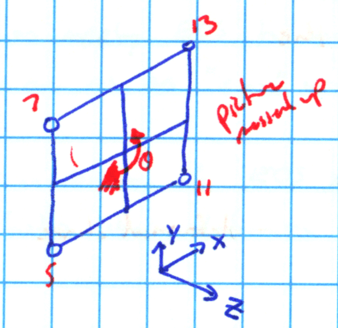
DISCLOSED TO AND UNDERSTOOD BY DATE WITNESS DATE

Work continued from Page

Oct 9th, 2002

want to "unzip" our spot weld model

- current analyses (8.0, 8.1, + 8.2) have all welds fail at same time



o spot welds

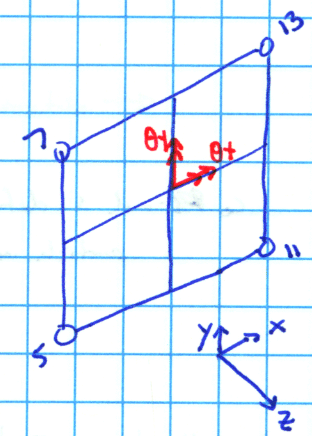
θ net rotation about diagonal

$\theta_x = \theta_y = \theta$

let $\theta_x = \theta_y = .1745$ rad (10 degrees)

save as 8.3 and rerun

[baseline model is 8.2]



slave surface

11-13-5-7

much too high applied rotation:

make it .01745 rad

up normal/shear strength to 10 lbs

final version of 8.3 → .01745 rot about x + y axes

1000 lbs normal + shear spot weld strength

Work continued to Page

SIGNATURE DATE CJW (10/9/02)

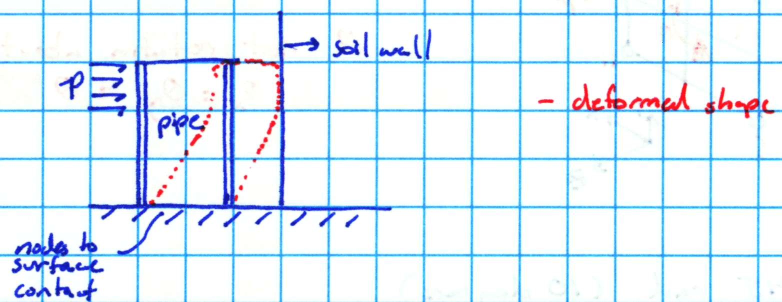
DISCLOSED TO AND UNDERSTOOD BY DATE WITNESS DATE

Work continued from Page

Oct 10, 2002

features in case 41 – thin walled pipe filled w/ concrete impacting soil wall

- concrete material properties of eqn of state tabulated computation and model
- soil properties
- contour surface? Abaqus doesn't have this
- semi-infinite elements



what is an equation of state + how can it be applied to concrete?

Oct 11

EOS

linear $U_s - U_p$ Hugoniot Form

$$P_H = \frac{\rho_0 c_0^2 \eta}{(1-s\eta)^2}$$

c_0 and s define linear shock velocity U_s and particle velocity U_p

$$U_s = c_0 + s U_p$$

$$\rho_0 c_0^2 = \text{elastic bulk modulus @ small strains} = K = \frac{E}{3(1-2\nu)}$$

$$\eta = \cancel{1-\frac{\rho_0}{\rho}}\ 1 - \frac{\rho_0}{\rho} = \text{nominal volumetric strain} = \epsilon_{vol}$$

not clear – start over

SIGNATURE | DATE

DISCLOSED TO AND UNDERSTOOD BY | DATE | WITNESS | DATE

Work continued from Page

possible text that might help
"Cracking in reinforced concrete flexural members" Shock and Vibration Digest vol 31 no 5 (9-1-99)
by Rao
→ no luck

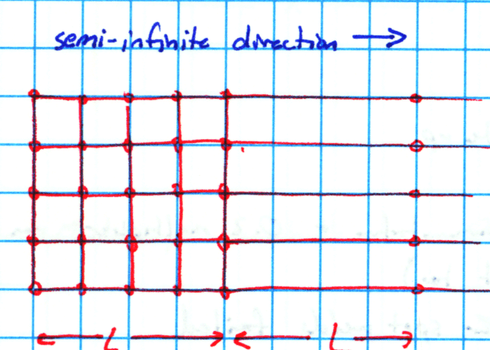
everything appears to focus on apply EOS to materials that change state (solid - liquid - gas)
really doesn't strike me as an "easy" way to model concrete at all

Abaqus does allow for a combination of EOS (defines only hydrostatic pressure component) and elastic shear deformation & a plasticity option (Mises or Johnson-Cook plasticity)

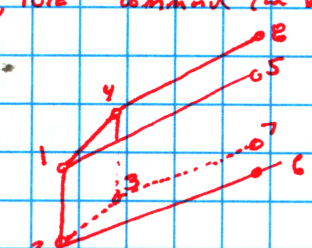
CJW

Oct 11, 2002

semi-infinite elements



if needed the *Ncopy, Pole command can be used (explicit user manual 2.1.1)



might have to use a script to define these elements?

SIGNATURE | DATE

DISCLOSED TO AND UNDERSTOOD BY | DATE | WITNESS | DATE