308 - - - Q200008310006
Scientific Notebook # 283

308 - - - Q200008310006
Scientific Notebook # 283

21
300

R

Simon Hsiang 210.522.5209
James O'Daniel
Joonhong Lim
Gwang Jin Oh

The Boorum & Pease® Quality Guarantee

The materials and craftsmanship that went into this product are of the finest quality. The pages are thread sewn, meaning they're bound to stay bound. The inks are moisture resistant and will not smear. And the uniform quality of the paper assures consistent rulings, excellent writing surface and erasability. If, at any time during normal use, this product does not perform to your expectations, we will replace it free of charge. Simply write to us:

Boorum & Pease Company
71 Clinton Road, Garden City, NY 11530
Attn: Marketing Services

Any correspondence should include the code number printed at the bottom of this page as well as the book title stamped at the bottom of the spine.

CNWRA
CONTROLLED
COPY 283

One Good Book Deserves Many Others.

Look for the complete line of Boorum & Pease® Columnar, Journal, and Record books. Custom-designed books also available by special order. For more information about our Customized Book Program, contact your office products dealer. See back cover for other books in this series.

Made in U.S.A.

Technical Approach:
- thermal analysis will be performed with ABAQUS/Standard version 5.7 - implicit numerical code

- impact analysis performed with ABAQUS/Explicit: 5.7 explicit numerical code

JSO

/ Theodor Shanthammer 8-28-2000

Wed 7-29-98 TK 8-28-2000

- developing matrix of plan for project being performed here at Penn State.

/TK/ Theodor Shanthammer 8-28-2000

- have material from ~~NE~~ CNWRA including
  1. Vol. III of IV: Appendix B: drawings of waste packages
  2. Vol. III of IV: partial info, waiting for the rest from CNWRA
  3. other less significant reports

- initial tasks
  1. finding relevant material properties: structural & thermal
  2. discretization of mesh: preliminary development of FE mesh in accordance with Appendix B drawings

ABAQUS, version 5.7 is being used for all simulations performed in this study. It is a commercial Finite element code developed by HKS Inc. Both the Standard (implicit) and Explicit parts will be used.

Preprocessing (development of the mesh) will be primarily performed with PATRAN. Addition development may be done with LS-INGRID due to my own familiarity with that ~~code~~ program.

TK/ 8-28-2000

JSO

/ Theodor Shanthammer 8-28-2000

Thurs 7-30-98 TK 8-28-2000

Initial Plan for Project: emailed to A. Chowdhury

FINITE ELEMENT MODELING OF WASTE PACKAGES
PSU SUPPORT EFFORTS

Objective:

To perform numerical simulations of various waste package designs with the finite element code ABAQUS. These packages will be subjected to thermal and structural loading environments over their entire design life. Additionally, rock-fall impacts will be simulated at specific times during that life span. The objective is to evaluate the designs adequacy for the specified storage environments, to estimate the possible failure conditions for these systems, and to determine if such failure could be expected under the specified storage environments. Additional loading environments, such as ground motion, may be modeled once the initial activity has been completed.

Activity 1:

This activity involves the Finite Element simulations, using ABAQUS, of the 21 PWR Uncannistered Waste Package. The effort will include the following simulations:

Phase 1: Prepare the FE model for the system under consideration, and collect all the required material information for the analysis.

- need report with more information from CNWRA.

JSO

Phase 2: Perform thermal analysis (implicit).

- performed to 1000 years.
- include the effects of gravity.
- assess the system's behavior.

AHC 8/30/00

Phase 3: Rock-Fall Impact Analysis (explicit).

- use the information obtained from Phase 2, above, and superimpose impact events.
- perform impact simulations at selected time intervals (e.g., 100 years) over the 1000 years.
- perform simulations for sufficient durations of the impact events.
- analyze the system's behavior and assess it for possible failure (both global and local).
- determine the combinations of type, size, shape and impact characteristics of rock blocks, and the quasi-static environments that can causes damage and/or possible failure of the system.
- initially use simplifications listed on page 6.3-59 (Vol. III of IV).
- initially use material properties listed on page 6.3-58 (Vol. III of IV).

JSO

/ Theodor Shanthammer 8-28-2000

Cont. →

Phase 4: Residual Stresses within WP

- need specifications and amounts of residual stresses from manufacturer of WP

CLARIFICATIONS AND COMMENTARY ON THE THERMAL ANALYSIS

- long time: 1000 years.
- implicit analysis: ABAQUS/Standard, version 5.7.

Simplifications:

- each tube has the rods modeled as smeared material.
- the FE mesh will cover the entire simulation volume, including empty spaces.

Needs:

- volume heat generation of smeared SNF.
- thermal properties of all the materials in the WP.
- conduction and convection phenomena within the air pockets surrounding the basket.
- any other factors that contribute to thermal behavior (e.g., thermal loss to the outside, etc.).

CLARIFICATIONS AND COMMENTARY ON THE STRUCTURAL ANALYSIS

- short duration events at various times (100 years?) over the 1000 year lifetime. The times at which this is performed need to be specified.
- use output from implicit structural/thermal analysis at the specific times.

Variables to be adjusted in parametric studies:

- consideration of the impactor/rock , includes:

  1. Geometry: spherical and octahedral. This implies that the impact can be: point (sphere, and corner of cube), flat, line, corner
  2. Mass.
  3. Material model.
  4. Impact velocity.
  5. Relative rock-WP position - what location is worst case?

- assess how the possible failure conditions vary over time.

Needs:

- mechanical properties of all materials, including the (Yucca Mtn?) Rock.
- boundary conditions (position, geometry, and composition of supports).

Possible Failure Modes:

- tensile, compressive, shear, etc., for both global and local responses.
- buckling of the various structural components.
- criticality: need to define what affects this phenomenon (e.g., equation of state? How does this relate to changes in the thermal/mechanical conditions?

Simplifications:

- listed in Vol. III of IV, p. 6.3-59.

1 Theodore Sheutherman 8-28-2000

7-30 cont. -98 TK 8-28-2000

Discretization of F.E. Mesh

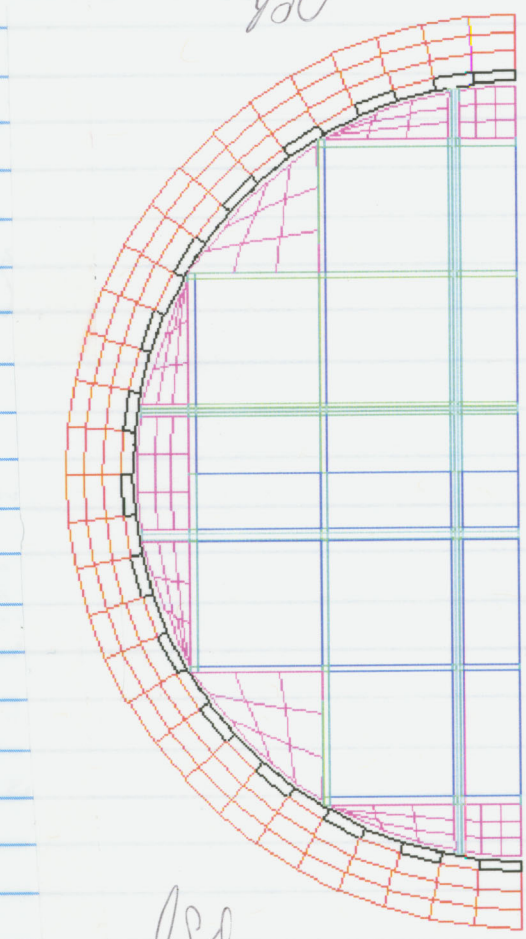
1. 3-D Solid continuum elements for all parts of WP

TK 8-28-2000

2. Interior Basket Assembly completely filled in with 3-D Solid elements
   - will set material properties to needed conditions, either air or empty materials, as necessary.

Preliminary LS-INGRID mesh

JSD



JSD

JSD.

Theodor Thurthammer 8-28-2000

8-1-98

- continuing development of geometry with LS-Ingrid

LS-Ingrid
- geometry (nodes & elements)
- what elements are what materials

Patran:
- boundary conditions
- contact
- everything else.

6 materials
1. A516
2. Alloy 825
3. 316L6A
4. 1063AL
5. SNF /TK 8-28-2000
6. whatever is in spaces around the basket assembly (air, probably)

/Theodor Trauthampner 8-28-2000

8-3-98

- split part 1 outer & inner barrier into two parts for ease of transition into Patran part
- changed definition of lids (outer & inner) to using "a" command rather than "~~cp~~ sf" in order to match up nodes with rest of the barriers /TK 8-28-2000

LS-Ingrid file
- for 21-PWR WP.
(not final version)

```
21 PWR Waste Package - Uncannistered
plane 2 0 0 0 1 0 0 .001 symm
      0 0 0 0 0 10 .001 symm
sd 1 cyli 0 0 0 0 0 1 0.68515
sd 2 cyli 0 0 0 0 0 1 0.7145
sd 3 cyli 0 0 0 0 0 1 0.7645
start
1 3 5; 1 21 41 61; 1 2 3 4 5 6 7 8 9;
0.7145 0.7645 0.8145
0 90 270 360
0 0.3 1.12 1.6875 2.24 2.3075 2.3325
2.4425 2.6675
cyli
mate 1
d 0 1 0 0 2 0
d 0 3 0 0 4 0
d 1 0 7 2 0 9
d 1 0 6 2 0 7
end
start
1 2; 1 21 41 61; 1 2 3 4 5 6;
0.6945 0.7145
0 90 270 360
0 0.3 1.12 1.6875 2.24 2.3075
cyli
d 0 1 0 0 2 0
d 0 3 0 0 4 0
mate 2
end
start
1 6 11 16 21; 1 6 11 16 21; 1 2;
-0.4 -0.4 0.0 0.4 0.4
-0.4 -0.4 0.0 0.4 0.4
2.3325 2.4425
di 1 2 0 4 5;1 2 0 4 5;
a 1 1 0 5 5 0 k 0.7645
d 3 0 0 5 0 0
mate 1
end
start
1 6 11 16 21; 1 6 11 16 21; 1 2;
-0.3 -0.3 0.0 0.3 0.3
-0.3 -0.3 0.0 0.3 0.3
2.3075 2.3325
di 1 2 0 4 5;1 2 0 4 5;
a 1 1 0 5 5 0 k 0.7145
d 3 0 0 5 0 0
mate 2
end
start
1 2 3 4 5 6 7 8 9 10 11 12 13 14 15
16; 1 2 3 4 5 6 7 8 9 10 11 12 13 14
15
16 17 18 19 20 21 22 23 24 25 26 27
28 29 30 31;
1 2 3 4 5 6 7 8 9 10;
0.0 0.29525 0.30475 1.11525 1.12475
1.68275 1.69225 2.23525 2.24475 2.298
2.3075
sf 0 2 0 0 2 0 sd 1
mate 7
mti ;4 5 0 8 9; 4
end
start
1 4;1 4; 1 2 3 4 5 6 7 8 9 10 11;
-0.34525 -0.12175 0.59375 0.68015
0.0 0.29525 0.30475 1.11525 1.12475
1.68275 1.69225 2.23525 2.24475 2.298
2.3075
sf 0 2 0 0 2 0 sd 1
mate 7
mti ;2 3 0 6 7 0 10 11; 4
end
start
1 4;1 4; 1 2 3 4 5 6 7 8 9 10 11;
-0.34525 -0.12175 -0.59375 -0.68015
0.0 0.29525 0.30475 1.11525 1.12475
1.68275 1.69225 2.23525 2.24475 2.298
2.3075
sf 0 2 0 0 2 0 sd 1
mate 7
mti ;2 3 0 6 7 0 10 11; 4
end
start
1 4;1 4; 1 2 3 4 5 6 7 8 9 10 11;
-0.68015 -0.59375 0.12175 0.34525
0.0 0.29525 0.30475 1.11525 1.12475
1.68275 1.69225 2.23525 2.24475 2.298
2.3075
sf 0 2 0 0 2 0 sd 1
mate 7
mti ;2 3 0 6 7 0 10 11; 4
end
start
1 4;1 4; 1 2 3 4 5 6 7 8 9 10 11;
-0.68015 -0.59375 -0.12175 -0.34525
0.0 0.29525 0.30475 1.11525 1.12475
1.68275 1.69225 2.23525 2.24475 2.298
2.3075
sf 0 2 0 0 2 0 sd 1
mate 7
mti ;2 3 0 6 7 0 10 11; 4
end
start
1 4;1 4; 1 2 3 4 5 6 7 8 9 10 11;
-0.68015 -0.59375 -0.10175 0.10175
0.0 0.29525 0.30475 1.11525 1.12475
1.68275 1.69225 2.23525 2.24475 2.298
2.3075
sf 1 0 0 1 0 0 sd 1
mate 7
mti ;2 3 0 6 7 0 10 11; 4
end
start
1 4;1 4; 1 2 3 4 5 6 7 8 9 10 11;
-0.68015 0.0 0.59375 0.68015
0.0 0.29525 0.30475 1.11525 1.12475
1.68275 1.69225 2.23525 2.24475 2.298
2.3075
sf 0 2 0 0 2 0 sd 1
mate 7
mti ;2 3 0 6 7 0 10 11; 4
end
start
1 4;1 4; 1 2 3 4 5 6 7 8 9 10 11;
-0.10175 0.0 -0.59375 -0.68015
0.0 0.29525 0.30475 1.11525 1.12475
1.68275 1.69225 2.23525 2.24475 2.298
2.3075
sf 0 2 0 0 2 0 sd 1
mate 7
mti ;2 3 0 6 7 0 10 11; 4
end
end
```

```
-0.68015 -0.59375 -0.58875 -0.58375 -
0.57875 -0.35975 -0.35525 -0.35025 -
0.34525
-0.13125 -0.12625 -0.12175 -0.11675 -
0.10675 -0.10225 0.0
-0.68015 -0.59375 -0.58875 -0.58375 -
0.57875 -0.35975 -0.35525 -0.35025 -
0.34525
-0.13125 -0.12625 -0.12175 -0.11675 -
0.10675 -0.10175 0.0
0.10175 0.10675 0.11675 0.12175
0.12625 0.13125 0.34525 0.35025
0.35525 0.35975
0.57875 0.58375 0.58875 0.59375
0.68015
0.0 0.29525 0.30475 1.11525 1.12475
1.68275 1.69225 2.23525 2.24475
2.3075
di 1 10 0 15 16; 1 2 0 30 31; 1 10;
di 1 6; 2 6 0 26 30; 1 10;
di 1 2; 6 10 0 15 17 0 22 26; 1 10;
mti 12 13; 1 31; 1 10; 3
mti 1 16; 12 13 0 19 20; 1 10; 3
mti 10 12 0 13 15; 1 3 0 29 31;1 10;
4
mti 15 16; 2 4 0 28 30; 1 10; 4
mti 6 10; 2 4 0 28 30; 1 10; 4
mti 6 8; 2 8 0 24 30; 1 10; 4
mti 2 8; 6 8 0 24 26; 1 10; 4
mti 2 4; 6 10 0 15 17 0 22 26; 1 10;
4
mti 1 4; 10 12 0 13 15 0 17 19 0 20
22; 1 10; 4
mti 4 8; 8 12 0 13 19 0 20 24; 1 10;
5
mti 8 12 0 13 16; 3 12 0 13 19 0 20
29; 1 10; 5
mti 5 7 0 9 11 0 14 16; 9 11 0 14 18
0 21 23; 1 10; 6
mti 9 11 0 14 16; 5 7 0 25 27; 1 10;
6
end
start
1 4;1 4; 1 2 3 4 5 6 7 8 9 10 11;
-0.59375 -0.35525 0.35525 0.57875
0.0 0.29525 0.30475 1.11525 1.12475
1.68275 1.69225 2.23525 2.24475 2.298
2.3075
sf 0 2 0 0 2 0 sd 1
mate 7
mti ;4 5 0 8 9; 4
end
start
1 4;1 4; 1 2 3 4 5 6 7 8 9 10 11;
-0.59375 -0.35525 -0.57875
```

/Theodor Trauthampner 8-28-2000

8-8-98 TK 8-28-2000

- spent time developing mesh in grid part basically complete

Patran to do:
1. mat props
2. contact
   - inside
   - outside
3. boundary conditions
4. initial velocity for the rock

Need to know
- initial rock size

/TK 8-28-2000

/Theodor Stauffhammer 8-28-2000

8-10-98 TK 8-28-2000

- attempting first abaqus file from Patran

Question that came up during discussion: How to explicitly model the SNF rods? Won't do this initially, but eventually will have to.

Material List

1	outer barrier	A516
2	inner barrier	Alloy 825
3	alum. shunt	6063AL
4	basket framework	Alloy 825
5	tubes	316316A
6	SNF	SNF
7	filler	filler
8	Rock	rock

/Theodor Stauffhammer 8-28-2000

8-11-98 TK 8-28-2000 TK 8-28-2000

Material List:

1	outer barrier	A516
2	inner barrier	Alloy 825
3	AL shunt	6063AL
4	basket frame/support	Alloy 825
5	tubes	316316A
6	SNF	SNF
7	filler	filler (?)
8	rock	rock

boundary conditions:
- symmetry planes added in obvious places X & Z directions
- fixing (retraining) nodes at approximate positions where the supports are.

* Need to know more about supports
1. where are they located?
2. geometry of them
3. material (eventually)

initial velocities
- imported onto the rock
- calculated from drop height

Contact
1. inner barrier – basket assembly
2. rock – outer barrier

/Theodor Stauffhammer 8/28/2000

8-11 cont. - 98 TK 8-28-2000

Separate model into 4 parts (groups in Patran)
1. outer barrier
2. inner barrier
3. basket assembly (everything inside)
4. rock

This simplifies the making of the contact surfaces
- just show 1 group on the screen and define its entire surface as one side of the interface.

/Theodor Krauthammer 8-28-2000

8-12 - 98 TK 8-28-2000
still working on getting geometry of contact surfaces correct

/Theodor Krauthammer 8-28-2000

8-13 - 98 TK 8-28-2000
meeting with CAC people (computer guys at Penn State) to get ABAQUS Explicit manuals

- attempted first run: problem with license, wait for CAC to fix it.

/Theodor Krauthammer 8-28-2000

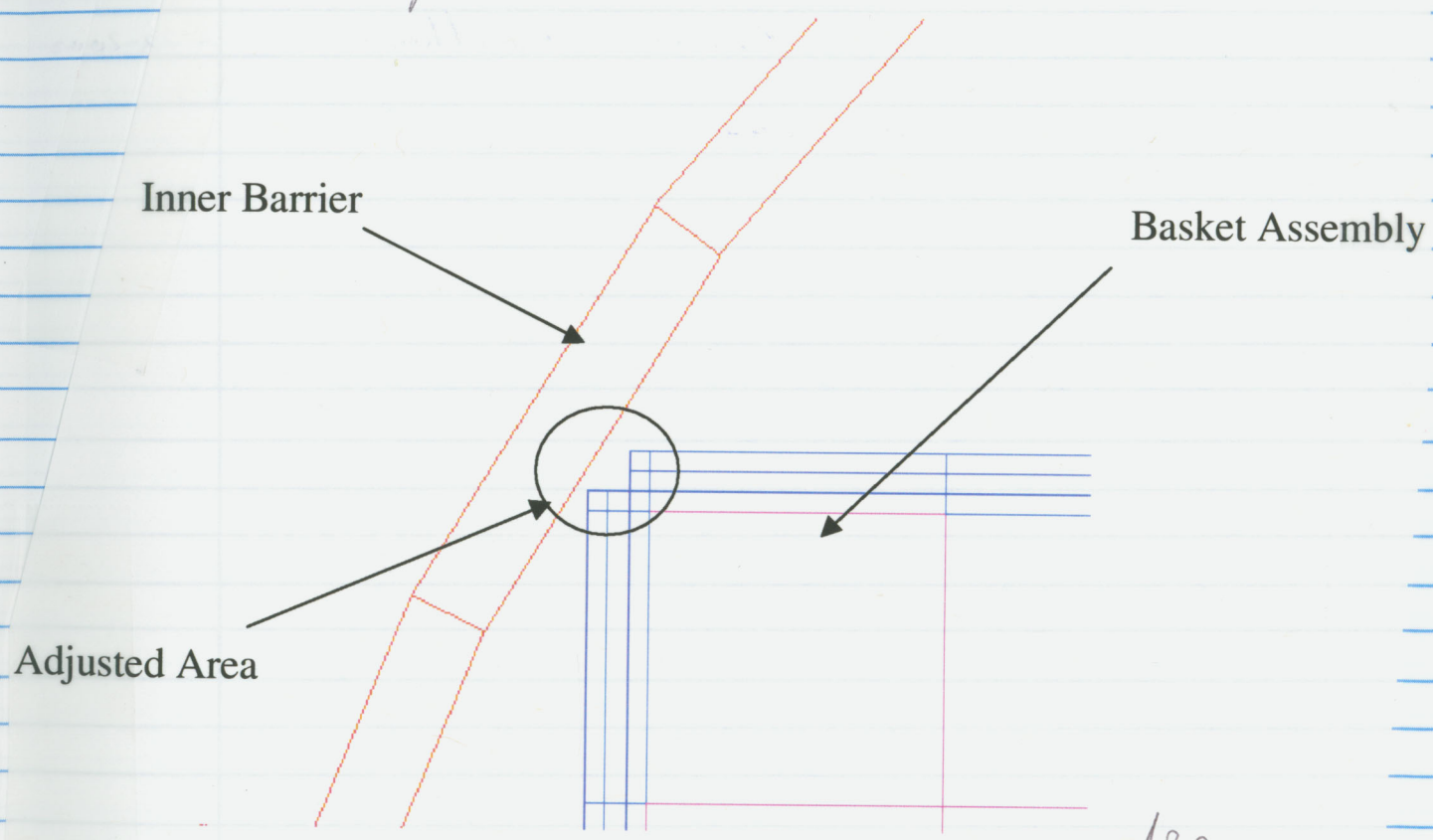
8-14-98 TK 8-28-2000

revision of mesh

- problem that basket assembly and inner barrier are overlapping, adjusted corners accordingly, show new figure of it.

- basket corners smaller by approximately 18.0mm

J-Theodor Stauthammer 8-28-2000



J-Theodor Stauthammer
8-28-2000

8-17-98 TK 8-28-2000

- fixed basket assembly mesh size to match filler parts.

- fixed 3 filler ~~parts~~ (mesh) in the end guide parts
/TK 8-28-2000

Latest mesh: very coarse

12788 nodes : 755 toleranced out / coincident nodes
9320 elems

JSO
/Theodor Stauthammer 8-28-2000

8-18-98 TK 8-28-2000

12:30 pm: meeting, received latest reports
3 on ANSYS rock falls
1 on ANSYS thermal

read and summarize: may assumptions in these reports.

Need to make list of our assumptions.

JSO
/Theodor Stauthammer 8-28-2000

8-20-98 TK 8-28-2000

- meeting / discussion

keep working on Patran part: waiting for Explicit to be ready to go

kept working on CNWRA reports (reading them)
JSO
/Theodor Stauthammer 8-28-2000

8-21-98 TK 8-28-2000

Bookkeeping stuff: gathering material together

Assumption:
all parts inside run the length of the inner barrier means end guides are too long. caps are in the right place.
/TK 8-28-2000

Assumption: Inner barrier lid is attached the same at both ends. Means 30mm extra on one end is ignored for now.

Assumption: aluminum shunt extends length of inner barrier and is all one parts (not 2 sets of 4 shunts)

Try run on implicit version to find bugs.

JSO
/Theodor Stauthammer 8-28-2000

8-21-98 TK 8-28-2000

mesh refinement: how will affect the results?
-will have to be considered

/Theodor Stouthamer 8-28-2000

8-31-98 - TK 8-28-2000

Can use combination of standard and explicit data/results in ABAQUS
information flows 2-way

Consider multiple impacts?

Timeline for run

Run standard ~150 years → do explicit impact → run standard for remainder of lifetime

Problem: toleranced too much in LS-Ingrid: node 1472 ended up in both the inner barrier & the basket assembly

Need to fix:
1. output controls: tailor to explicit requirements
2. what's in and out of STEP command

/Theodor Stouthamer 8-28-2000

8-31 cont. -98 TK 8-28-2000

- paperwork: generated invoice for 8/98
- Need to include GRAVITY: how in ABAQUS

/Theodor Stouthamer 8-28-2000

9/2-98 TK 8-28-2000

fixing node 1472 (& any others), correcting tolerancing

didn't work the first time

/Theodor Stouthamer 8-28-2000

9/3/98 TK 8-28-2000
Still working on tolerancing problem. May have to change units to mm rather than meters.

/Theodor Stouthamer 8-28-2000

TK 8-28-2000
9/7/98 rep3.db

redoing all the necessary things for Patran database

9-7, cont. /98 TK 8-28-2000

1 materials / properties ✓
2 groups ✓
3 symm. planes ✓
4 b.c.'s ✓
5 init. velocity ✓
6. contact ✓
7 bad case

/ Theodor Stauthammer 8-28-2000

9/24/98 TK 8-28-2000

Starting DYNA3D run: expecting errors

1. explicit solid element type: C3D8R (C3D8 in implicit code)

2. all contact information needs to be moved inside the STEP

3. node 1508 belongs to two sides of contact pair

* must have toleranced out too large: thereby eliminating some nodes that should not have been.

re-toleranced: 691 nodes deleted

~~redone~~ / TK 8-28-2000

9/24 cont. /98 TK 8-28-2000

Contact Surface problems

1. ambiguous connection at certain nodes

2. no connection between nodes 1721 & 1802
(shouldn't be a connection here, they're close but not on the same element)

ABAQUS changes S2 → S5 for the surface numbering why!?

emailing support@hks.com for answer, and patran support email.

/ Theodor Stauthammer 8-28-2000

9/25/98 TK 8-28-2000

Working on correcting contact surfaces
- redoing surfaces within PATRAN
- redoing toleranceing in LS-ingrid to get correct node configuration

/ Theodor Stauthammer 8-28-2000

9-28-98 TK 8-28-2000

Working on contact surfaces
- have to redo PATRAN part after each change of the mesh.

/Theodor Krauthammer 8-28-2000

9-29-98 TK 8-28-2000

PATRAN reply: ABAQUS needs to change surface numbers (SD → SS) to match its configuration. Wont affect the run.

Still have ambiguous error : not connected error.

/Theodor Krauthammer 8-28-2000

10/12/98 TK 8-28-2000

Have spent time trying to decipher contact surface problem.

reply from HKS helpful, but doesn't completely fix the problems.

ABAQUS Post took some fiddling to get it running.

10/12 cont/98 TK 8-28-2000

Redefining interior contact between inside of inner barrier and the basket assembly to be a surface (inner barrier) and a contact node set (the entire basket assembly)
- mesh is set up so this is easy

Just using the command CONTACT NODE PAIR and generating from node 1646 to node 11460

Job runs!! to 2.0 msec w/arbitrary material properties

Contact surface on wrong side of rack : fix it.

/Theodor Krauthammer 8-28-2000

10/13/98 TK 8-28-2000

altered interval for recording output

½ hour to run 2.0 msec

Results:
Some interpenetration : need much better mesh density.

re run w/ corrected rack slideline.

/Theodor Krauthammer 8-28-2000

10/14/98 TK 8-28-2000

Changed material properties
A516
Alloy 825
6063AL
B316L
used these

Rock from previous report

filler, SNF used arbitrary numbers
(don't have mechanical props. for these, yet)

- typo error: restarted

JSD
/ Theodor Shanthammer 8-28-2000

10/15/98 TK 8-28-2000

Error: deformation speed greater than wave speed

changed filler $\rho = 1000$ kg/m³ $E = 3 \times 10^7$ Pa
/ TK 8-28-2000

Still the error

Filler: $\rho = 1000$ $E = 3 \times 10^9$ Pa

Still speed problems

mesh needs to be more dense

JSD
/ Theodor Shanthammer 8-28-2000

10/16/98 TK 8-28-2000

redoing mesh in LS-ingrid w/ approximately 15000 elements
- added elements to outer + inner barriers.

trying various material properties to try and get the original run to work

Still have problem w/ deformation speed.

JSD
/ Theodor Shanthammer 8-28-2000

10/17/98 TK 8-28-2000

Ran to 0.5 msec: works. (Not to 2.0 msec)
- not enough time to capture entire response

JSD
/ Theodor Shanthammer 8-28-2000

10/20/98 TK 8-28-2000

finished denser mesh: ran thru PATRAN

added contact, bc's, mat. props.,

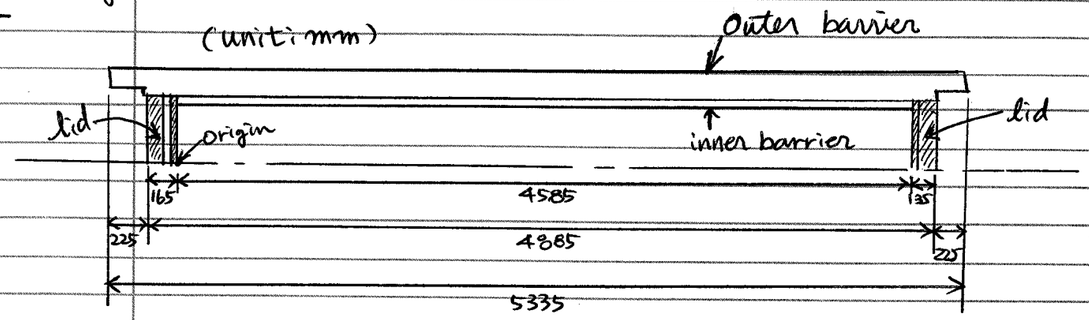
Still: same problem w/ wave speed. larger differences now.

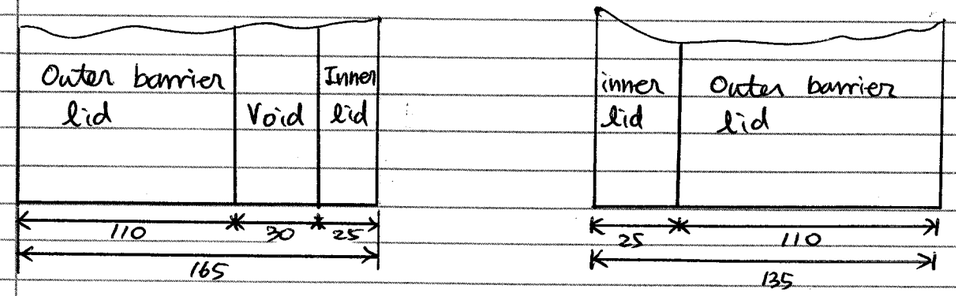
JSD
/ Theodor Shanthammer 8-28-2000

Pages 22 Through 49 Are Intentionally
Left Blank

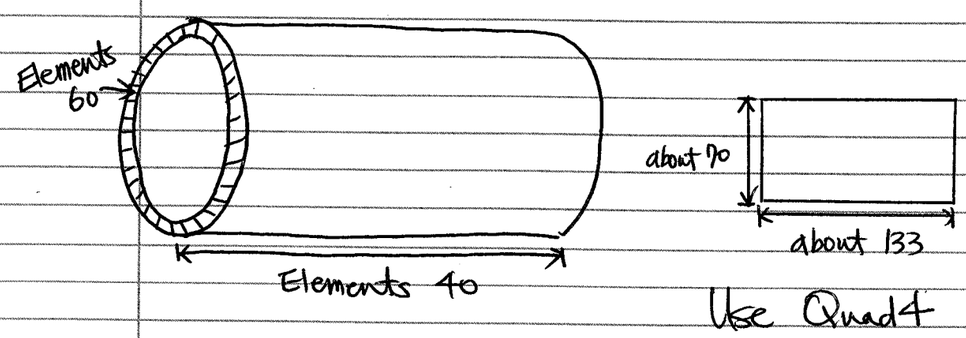
Jan-20-99 21:30 ~ 00:20

- Build the geometric model of outer and inner barrier by using Patran.
-



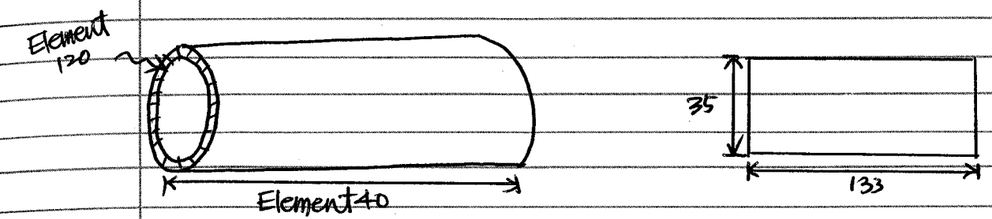


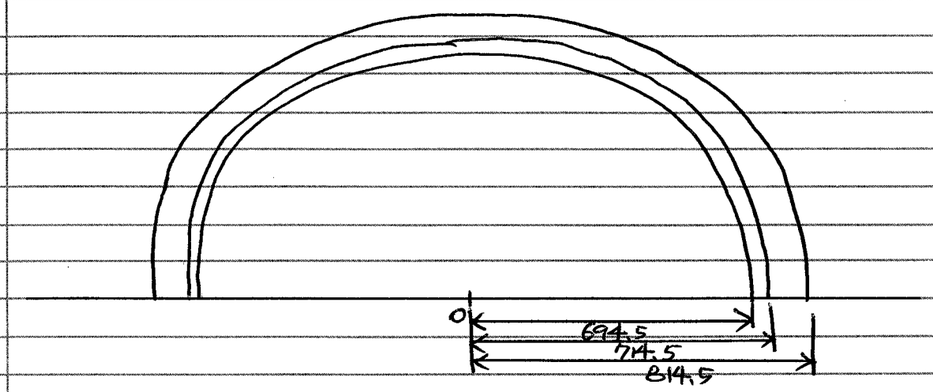
- Mesh
  - Outer Barrier.



Use Quad4

- Inner Barrier.

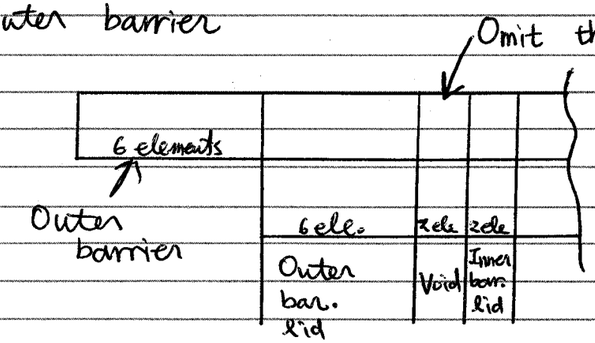




Jan-20-99

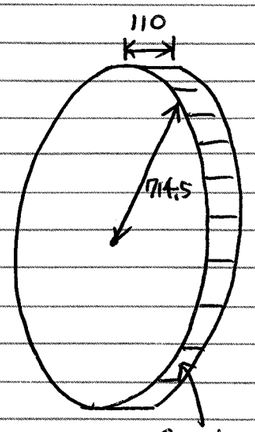
Jan-21-99 20:20 ~ 00:20

- Rebuild the outer barrier. Rebuild the mesh.
- Outer barrier



※ The container is not exact symmetry about the longitudinal axis.

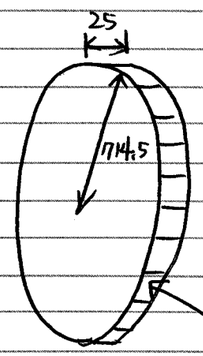
- Outer barrier lid



60 elements ~~around~~ along the perimeter. Use Quad 4

AHC 8/20/2020
Jan-21-1999

- Inner Barrier lid



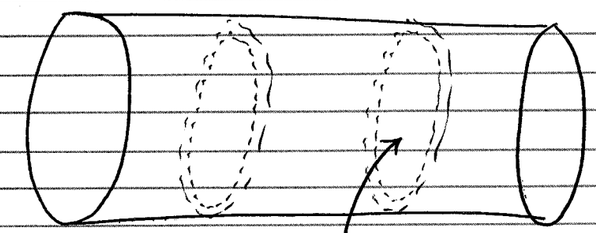
60 elements along the perimeter. Use Quad 4

Jan-21-99

Jan-22-99 11:00 ~ 12:00
- Discussion with Mr. Oh
- About work flow

Jan-22-99

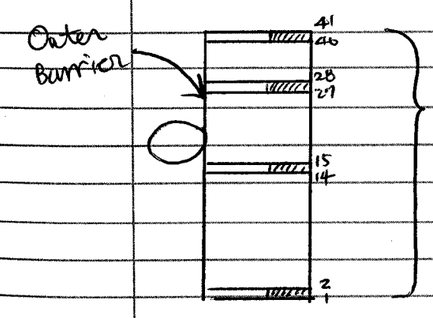
14:30 ~ 15:45
- Meeting with Dr. Krauthammer.
- • Model number: 21 PWR
  • Maintain the element aspect ratio below 2:1
  • The aspect ratio at both sides of outer barrier is not good.



Adjust the density to increase the weight.

Jan-22-99

15:45 ~ 18:00
- Set up material properties. Use equivalance.
- Set up boundary conditions



Element 40 in the middle of the outer barrier.

Jan-22-99

Element numbers.

Inner lid : 16691 ~ 17990
Outer lid : 14741 ~ 16690, 17991 ~ 19940
Inner Bar. : 121 ~ 4920
Outer Bar. : 5641 ~ 12600, 12721 ~ 13440
Rock : 20011 ~ 22110

Material Properties.

Outer barrier (lid) E : 206 GPa
Density : 8131 kg/m³
Poisson's ratio : 0.3
Yield strength : 205 MPa
} A516

Inner Barrier (lid) E : 206 GPa
Density : 8140 kg/m³
Poisson's ratio : 0.42
Yield Strength : 338 MPa
} Alloy 825

Jan-22-99

Jan-25-99 21:30 ~ 23:35

- Input the parameters for the impact of the rock. Try analysis.
- The initial velocity : $v_0 = \sqrt{2gh}$

where g : Gravity Acceleration : 9.81 m/s²
h : Height (m)

At 6 m drop height $v_0 = \sqrt{2 \times 9.81 \times 6}$
$= 12.6$ m/s

Jan-25-99

Analysis

Method
→ Analysis Deck

Restart Parameter
→ Write

Step Creation
→ Job step name
→ Job step parameters
→ Direct Steady State Dynamic
→ Select load case

Step selection
→ Job

Jan-25-99

Jan-26-99 21:30 ~ 00:30

- Try analysis at ABAQUS. Get some problems.
- To solve memory problem : Run on the mainframe.

cat sub
llsubmit sub ← script file
llq (instead of 'ps')
llcancel (instead of 'kill'. Splogin.cac.psu.edu)

Jan-26-99

Jan-27-99 21:30 ~ 00:50

- Try analysis
- Success

abaqus job=lt3 memory=30000000

Failure

abaqus job=lt3 convert=all interactive

Jan-27-99

Jan-28-99 19:00 ~ 01:00
- Abaqus works
- Change the job step to 0.2ms
  abaqus post job=lt3 memory= 30000000
  set, fill= on
  view, view = (x, y, z)
  zoom, factor = ...
  contour, var = S11
  draw, displaced ...
  initial velocity from -12.6 m/s to 1000 m/s

Jan-28-99

Feb-1-99 14:30 ~ 15:15
- Meeting with Dr. Krauthammer.
- Discuss the warning messages about the boundary conditions.

Feb-1-99

21:30 ~ 00:30
- Analyse the warning messages.
- Remove warnings with "ENCASTRE" key word.

Feb-1-99

Feb-3-99 21:00 ~ 00:00
- Check the boundary conditions of the model and fix some.

Feb-3-99

Feb-4-99 21:00 ~ 23:00
- Check the cross section of the model.
  * set, clip normal = (x, y, z)
  * set, clip point = (x, y, z)
  * set, clipping on.

Feb-4-99

99 14:30 ~ 15:35

ng with Dr. Krauthammer.

el summary of height (falling)

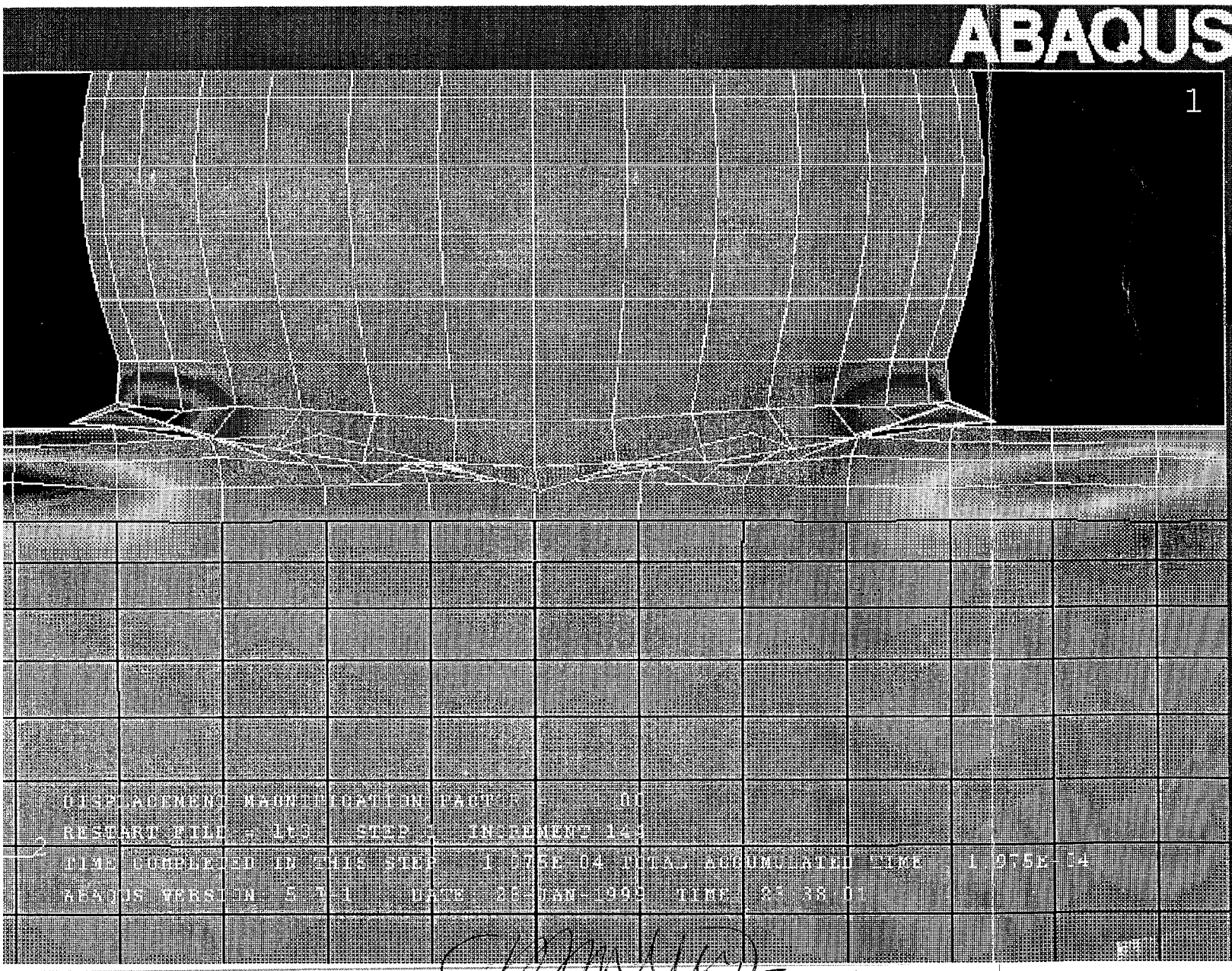
5m : reference, where

6m : " , "

c the number of elements and models.

2 Dr. Krauthammer on Monday morning.

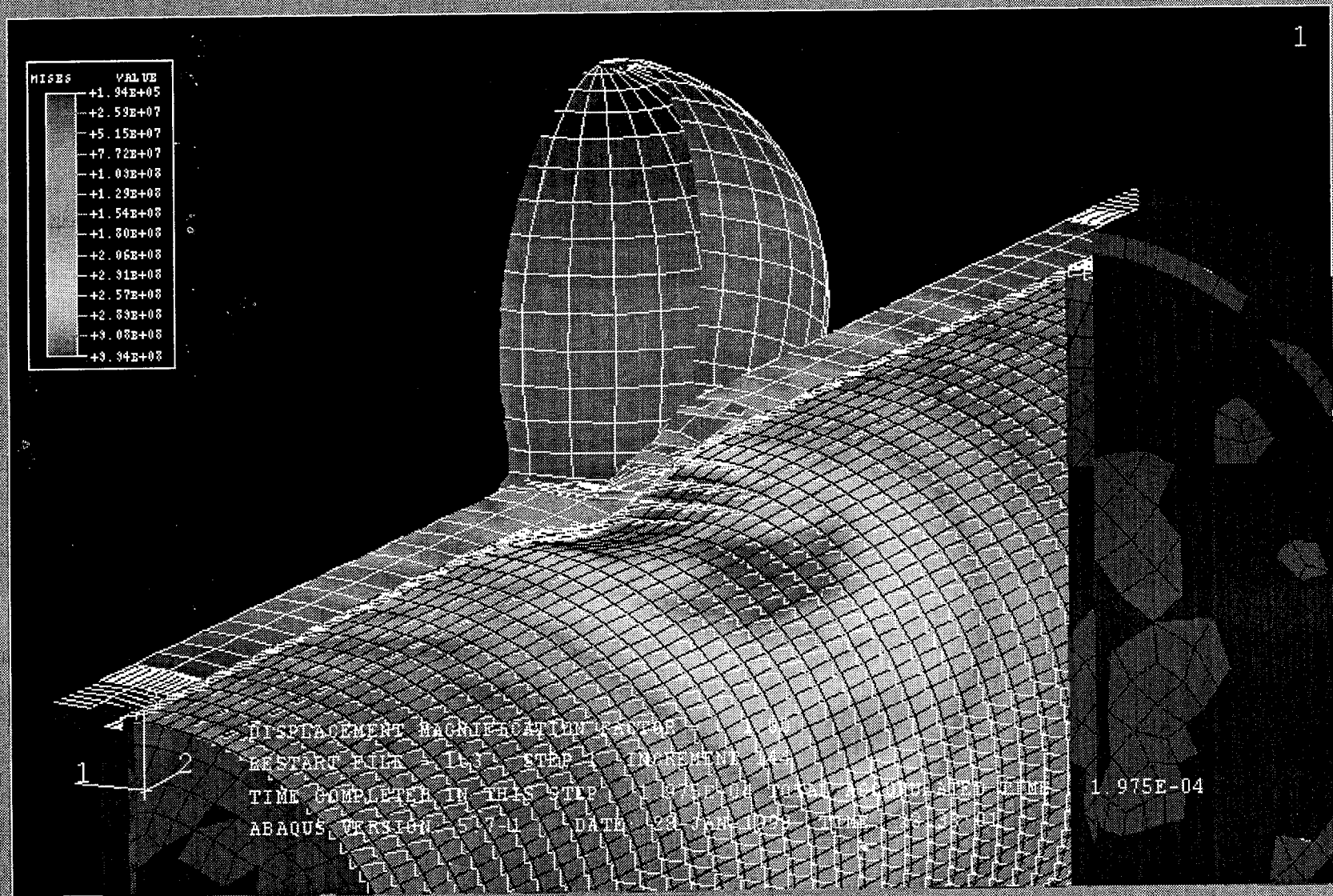
Feb-5-99



Feb-5-99

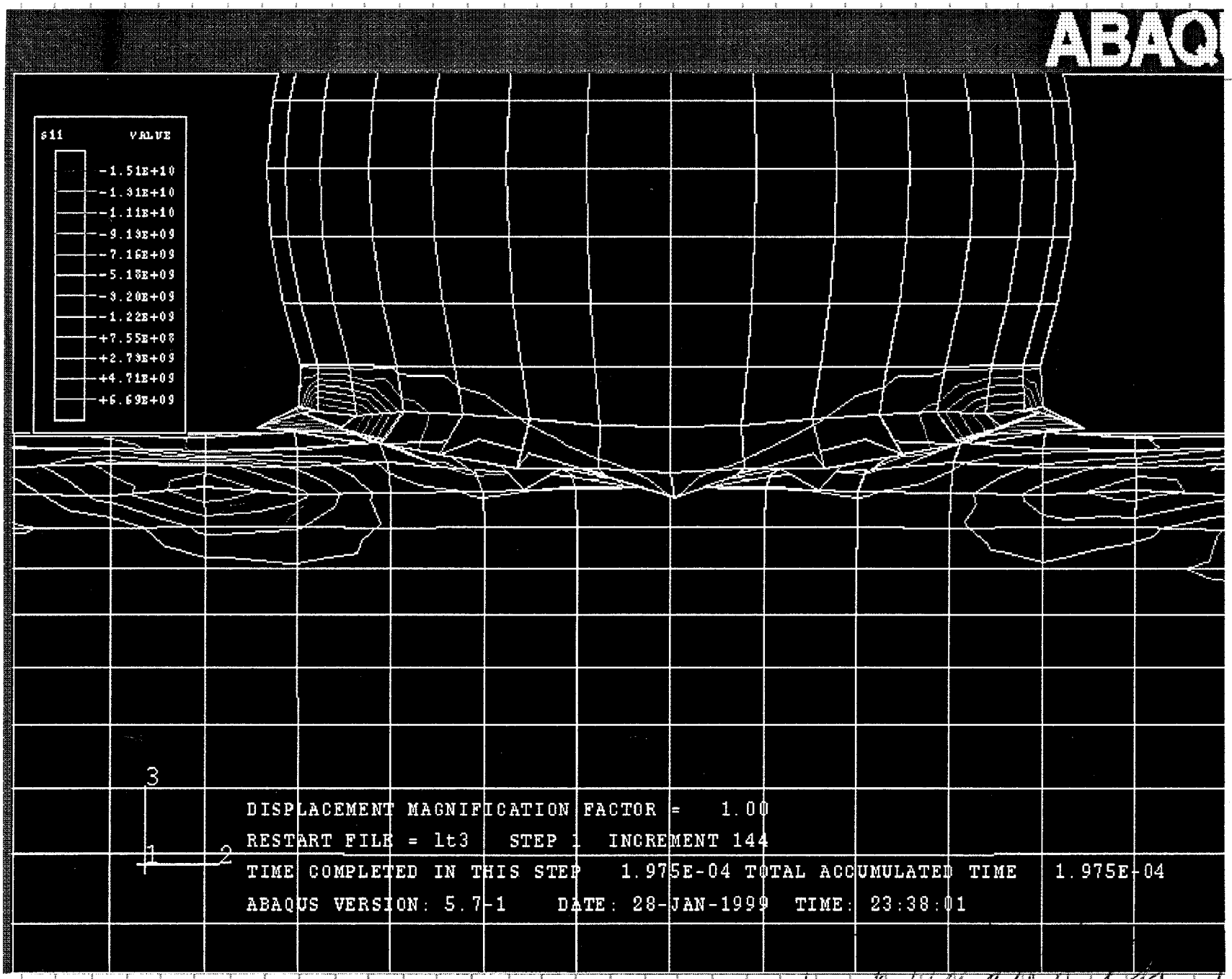
ABAQUS/Post Version 5.7-1

ABAQUS



m, CURSOR
TOUR, VAR=MISES
tour variable not valid for some element types in current DETAIL.
alid elements are plotted in white.

Feb-5-99


ABAQ
S11 VALUE
-1.51E+10
-1.31E+10
-1.11E+10
-9.13E+09
-7.16E+09
-5.18E+09
-3.20E+09
-1.22E+09
+7.55E+08
+2.73E+09
+4.71E+09
+6.69E+09
3
1
2
DISPLACEMENT MAGNIFICATION FACTOR = 1.00
RESTART FILE = 1t3 STEP 1 INCREMENT 144
TIME COMPLETED IN THIS STEP 1.975E-04 TOTAL ACCUMULATED TIME 1.975E-04
ABAQUS VERSION: 5.7-1 DATE: 28-JAN-1999 TIME: 23:38:01

Feb-5-99

Continued...

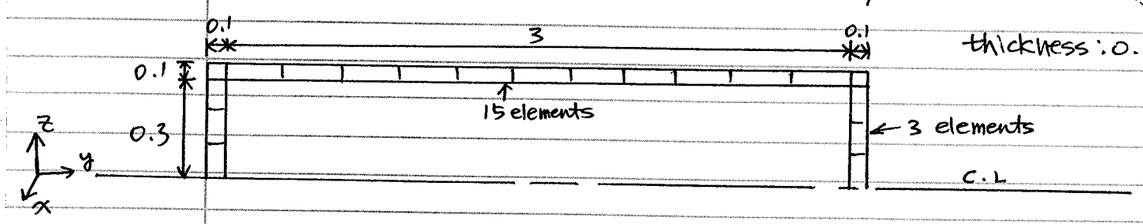
- Gas Pressure inside cylinder is between 26~31 psi
- temperature all component from table
- Warning removed. Rerun
- temperature model (implicit) then,
  Impact model (explicit)
- Mesh sensitivity check
  small model, same size ratio element
  → increase
  → check the differences

Feb-5-99

Feb-6-99 21:00 ~ 01:00

- 2:1 ratio element test model building
- ~~tar -cvf IMP1000.tar~~

dsm -machine = "txk1.archive" - password = "4drnje9"



Feb-6-99

Feb-7-99 21:00 ~ 00:30

- Change stone speed to 50. Fix errors
- Do equivalence
- There is no change of stress at cylinder main body.

Feb-7-99

Feb-8-99 21:00 ~ 00:30

- New model build. Same dimension. Feb-8-99
- Rock radius = 0.2, impact = ~~2000~~ 200 m/sec

- Stress-strain check

* read curve, name = e1, var = e11, elem = (number)
* " " " = S1, " S11, "
> e1
> S1
* DISPLAY CURVE
> se1

element # check

* set, el number = on
* ", n " = "
↑ node

Implicit (TRUSS)

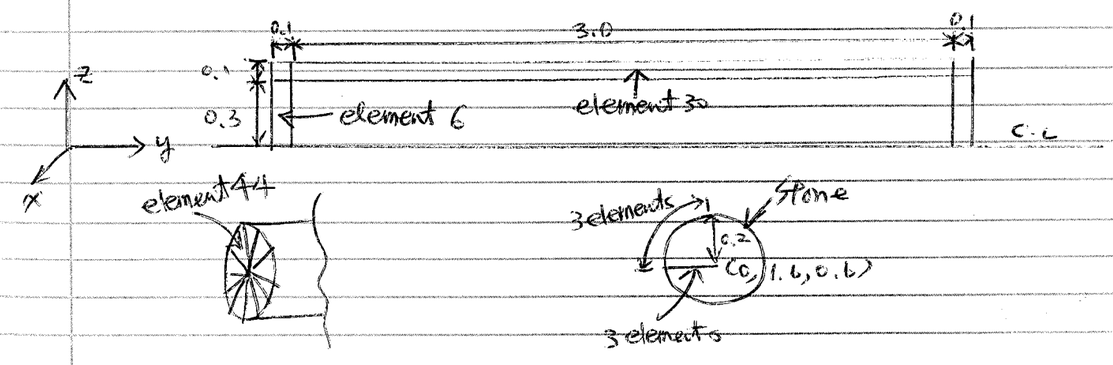
* Step, INC = 1000
* STATIC, DIRECT
1.0, 10.
↑ 1 sec ↑ 10 step

* EL PRINT
* NE PRINT
* END STEP

Feb-8-99

Feb-9-99 21:00 ~ 01:00

- Build refined 2:1 ratio mesh
-



Contact - type : General
Master : Barrier - Solid Face
Slave : Stone - Node

Feb-9-99

Feb-10-99 13:00 ~ 14:30
- Fixing Property errors of the stone.

Feb-10-99

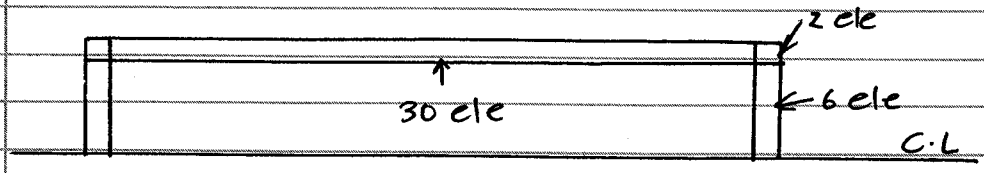
Feb-11-99 22:00 ~ 24:00
- Model complete. Post processing.
- Contact node 50

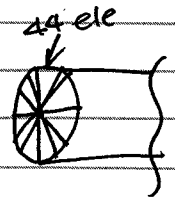
Feb-11-99

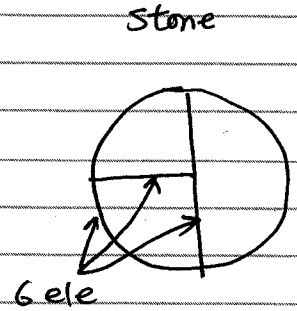
Feb-12-99 22:00 ~ 23:30
- Compare two results. Update model

Feb-12-99

Feb-13-99 21:00 ~ 01:00
- Build new model
: same dimension of the previous page.



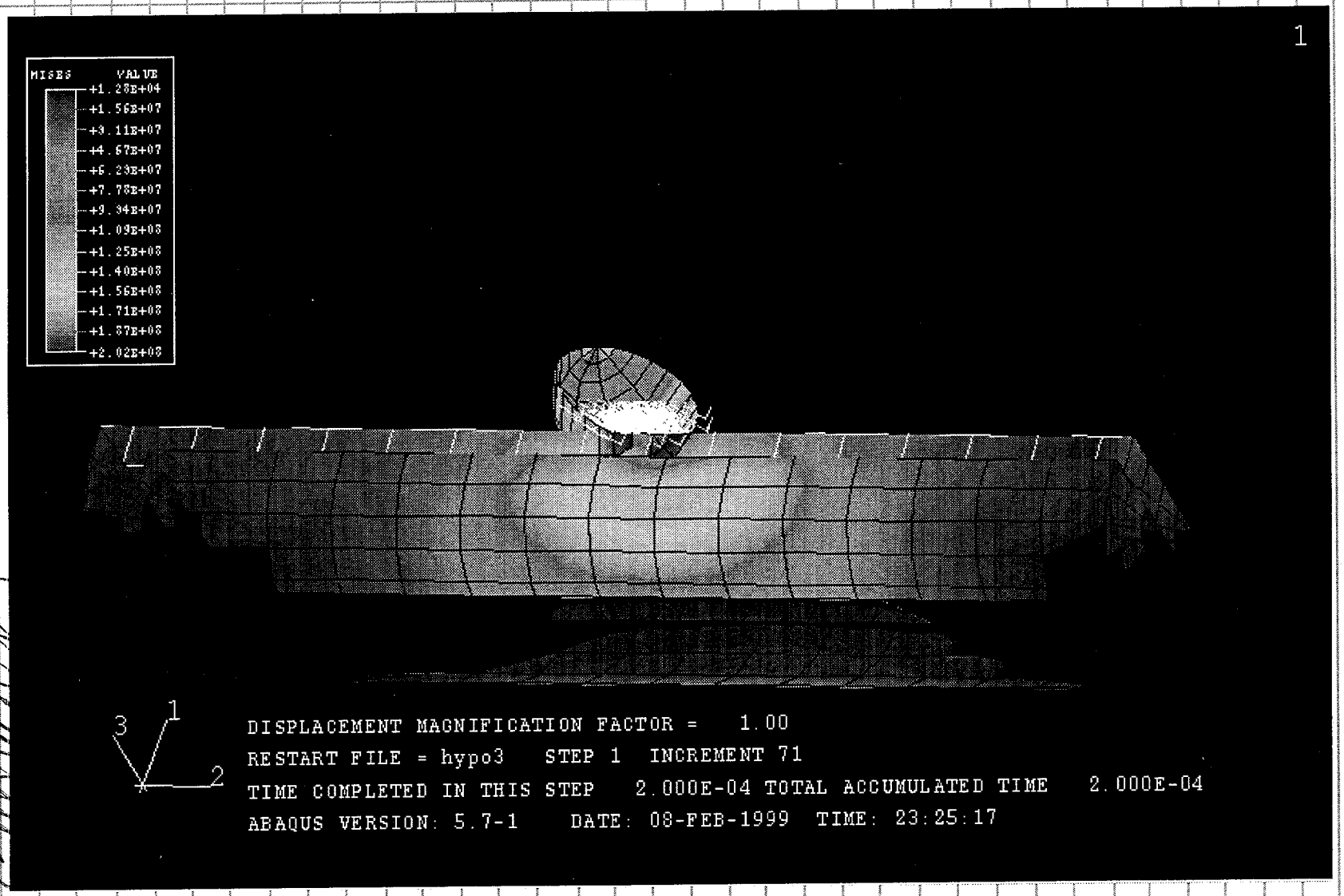




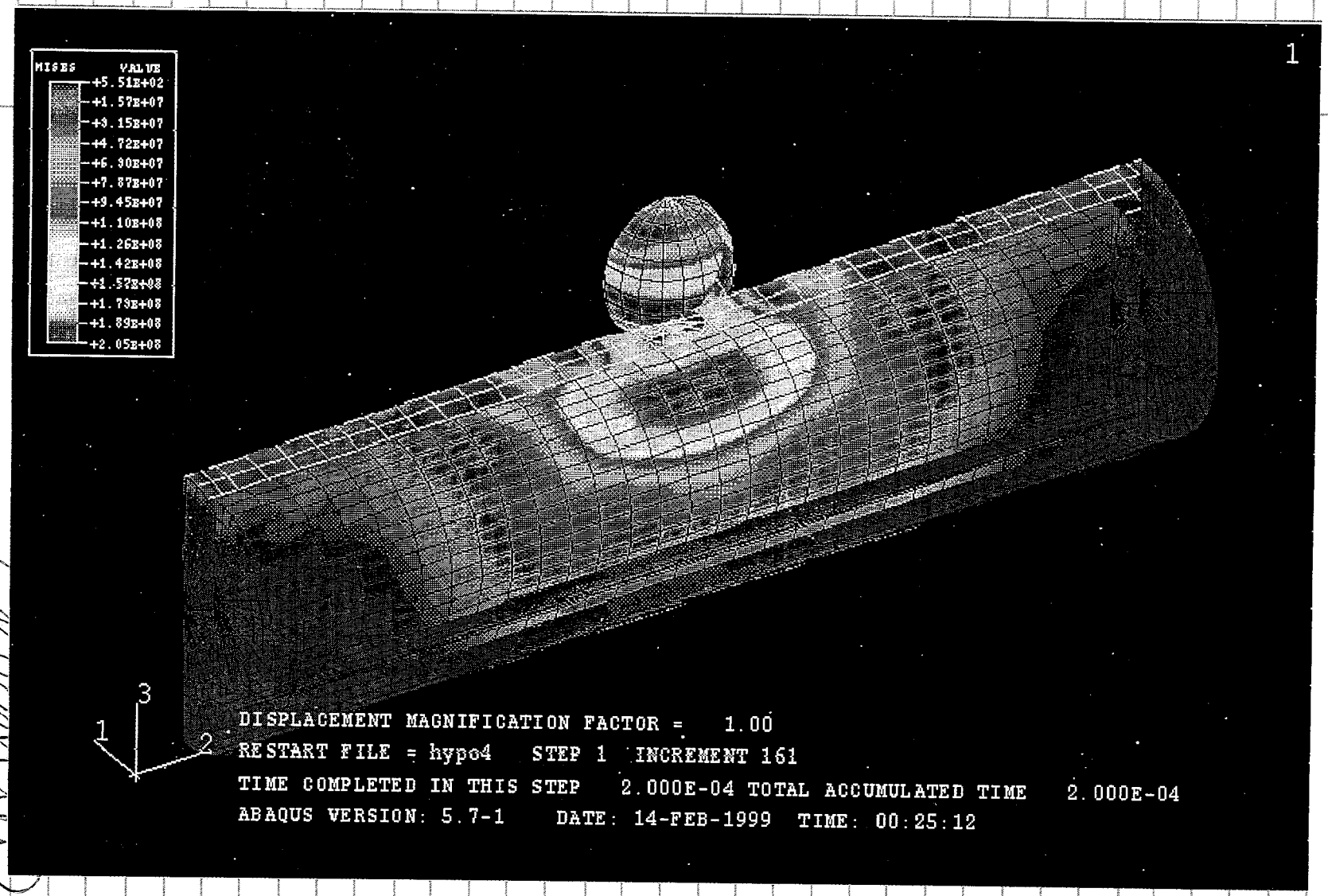
: 20 elements sweep

- It showed less than 10 % result comparing to the model ~~was~~ which was built on Feb-6-99. Good.
  Feb-13-99

Feb-13-99

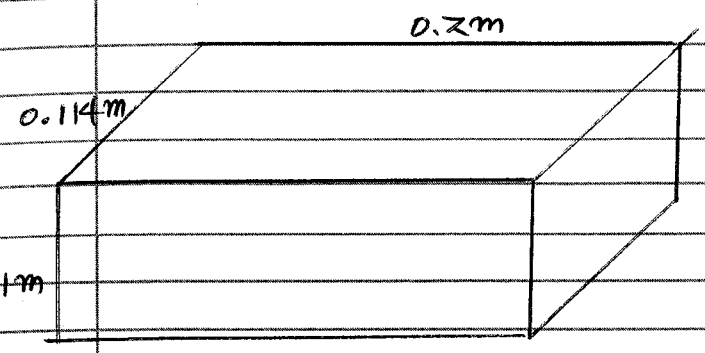

1
MISES VALUE
+1.28E+04
+1.56E+07
+3.11E+07
+4.67E+07
+6.23E+07
+7.78E+07
+9.34E+07
+1.09E+08
+1.25E+08
+1.40E+08
+1.56E+08
+1.71E+08
+1.87E+08
+2.02E+08
3
1
2
DISPLACEMENT MAGNIFICATION FACTOR = 1.00
RESTART FILE = hypo3 STEP 1 INCREMENT 71
TIME COMPLETED IN THIS STEP 2.000E-04 TOTAL ACCUMULATED TIME 2.000E-04
ABAQUS VERSION: 5.7-1 DATE: 08-FEB-1999 TIME: 23:25:17

Feb-13-99

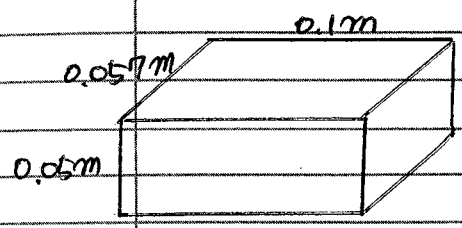

MISES VALUE
+5.51E+02
+1.57E+07
+3.15E+07
+4.72E+07
+6.30E+07
+7.87E+07
+9.45E+07
+1.10E+08
+1.26E+08
+1.42E+08
+1.57E+08
+1.73E+08
+1.89E+08
+2.05E+08
1
3
1
2
DISPLACEMENT MAGNIFICATION FACTOR = 1.00
RESTART FILE = hypo4 STEP 1 INCREMENT 161
TIME COMPLETED IN THIS STEP 2.000E-04 TOTAL ACCUMULATED TIME 2.000E-04
ABAQUS VERSION: 5.7-1 DATE: 14-FEB-1999 TIME: 00:25:12

Feb-13-99

- element of first model

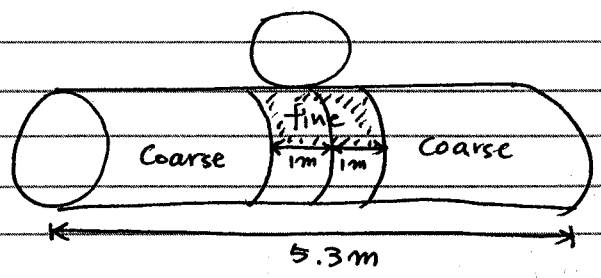


- element of second model



Feb 13 99

Feb-15-99
14:30 ~ 15:35
- Meeting with Dr. Krauthammer

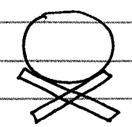


targetting to 15,000 element limit.

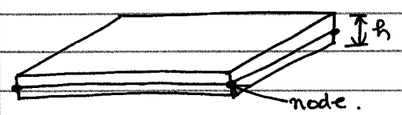
- Report (need 2 or 3 pages)
- Finalize mesh issue
- Inside i need spacers. Solid plate. Assign them mass plate + rods.

$m^* = V \times \rho$ (artificial density)

- How many spacers do we have?



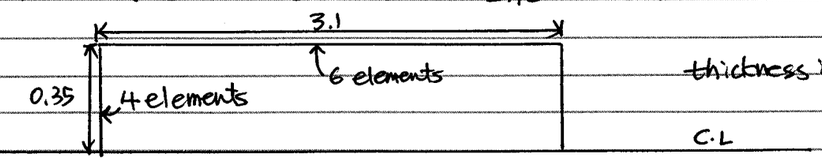
- first make 1~2 ms of coarse mesh
- make our model <u>shell element</u>.

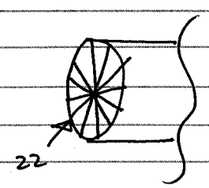


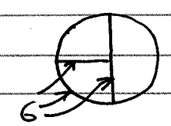
Feb-15-99

21:00 ~ 00:20

- Rebuild the first models with shell element







— 20 sweeps

Feb-15-99

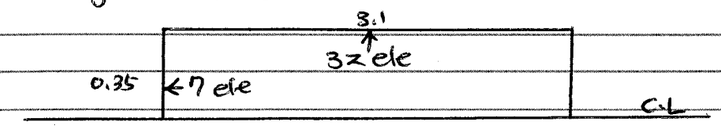
Feb-16-99 21:00 ~ 00:30

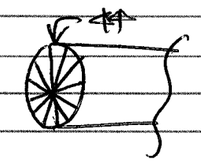
- Material property and B.C and load, contact Fix errors. Finish the first model
- Shell element change.

Feb-16-99

Feb-17-99 14:30 - 18:00

- begin with the second model





The stone is the same as before

Feb-17-99

Feb-18-99 15:00 ~ 18:20

- Material Property, B.C and etc. Fix errors, and post processing.
- There is not much difference between 3D and shell element. The maximum was 7%

Feb-18-99

Feb-19-99 15:00 ~ 18:00

- Post ~~pace~~ processing models. Compare each model
- * set, c report : Max stress, min stress
  * set, d report : Displacement.

Feb-19-99

Feb-20-99 14:00 ~ 15:45

- Rebuild shell element with refined mesh.

Feb-20-99

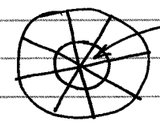
Feb-22-99 15:30 ~ 16:30

- Discussion of mesh sensitivity with Mr. Oh

16:30 ~ 18:40

- Check distorsion warning of shell element.

-



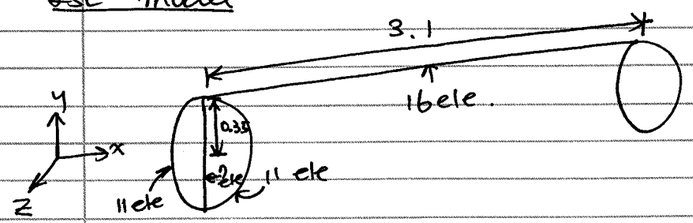
The shell element at this point

Feb-22-99

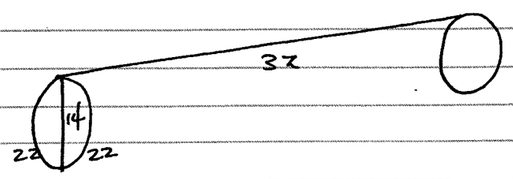
Feb-23-99 18:00 ~ 00:10

- Write report and rebuild shell element.
- Geometry

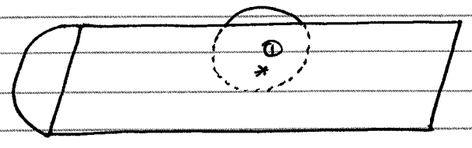
<u>1st model</u>



Sweep 10



① Inside of cylinder.



3D element	model	4.7731E07
"	"	3.1431E07
shell	model	1.0513E08

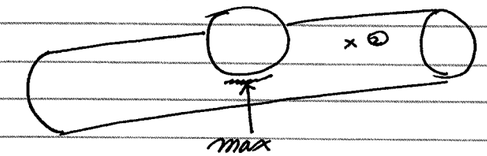
shell model can not be trusted.

Feb-23-99

Feb-24-99 0:10 ~ 1:10

- Write simple report around 2.05E08

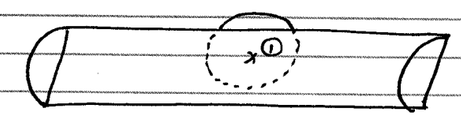
Out side of cylinder, maximums are same.



② 3D element	model	3.968E07
(refined) "	"	4.3266E07
shell	model	2.05E08

The shell model is different. But the color distribution seemed all right. It was strange though.

Inside of cylinder



① 3D element	model	4.7731E07
"	"	3.1431E07
shell	model	1.0513E08

shell model is much different.

Feb-24-99

14:30 ~ 15:35

- Meeting with Dr. Krauthammer.
  - Rock should be checked.
  - Shell might have vibration.
  - Check shell element. Change shell element.

- If find good shell element, then check aspect ratio. Contact problem of shell should be checked.
- thickness issue.
- Typical value : 465 lb for rod.
- 10.97 g/cm³ : Uranium

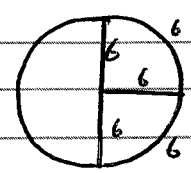
Feb-24-99

Feb-25-99 15:30 ~ 17:40
- Change shell element and compare solid element.
- Stress distribution changes a little as the element changes.

Feb-25-99

Feb-26-99

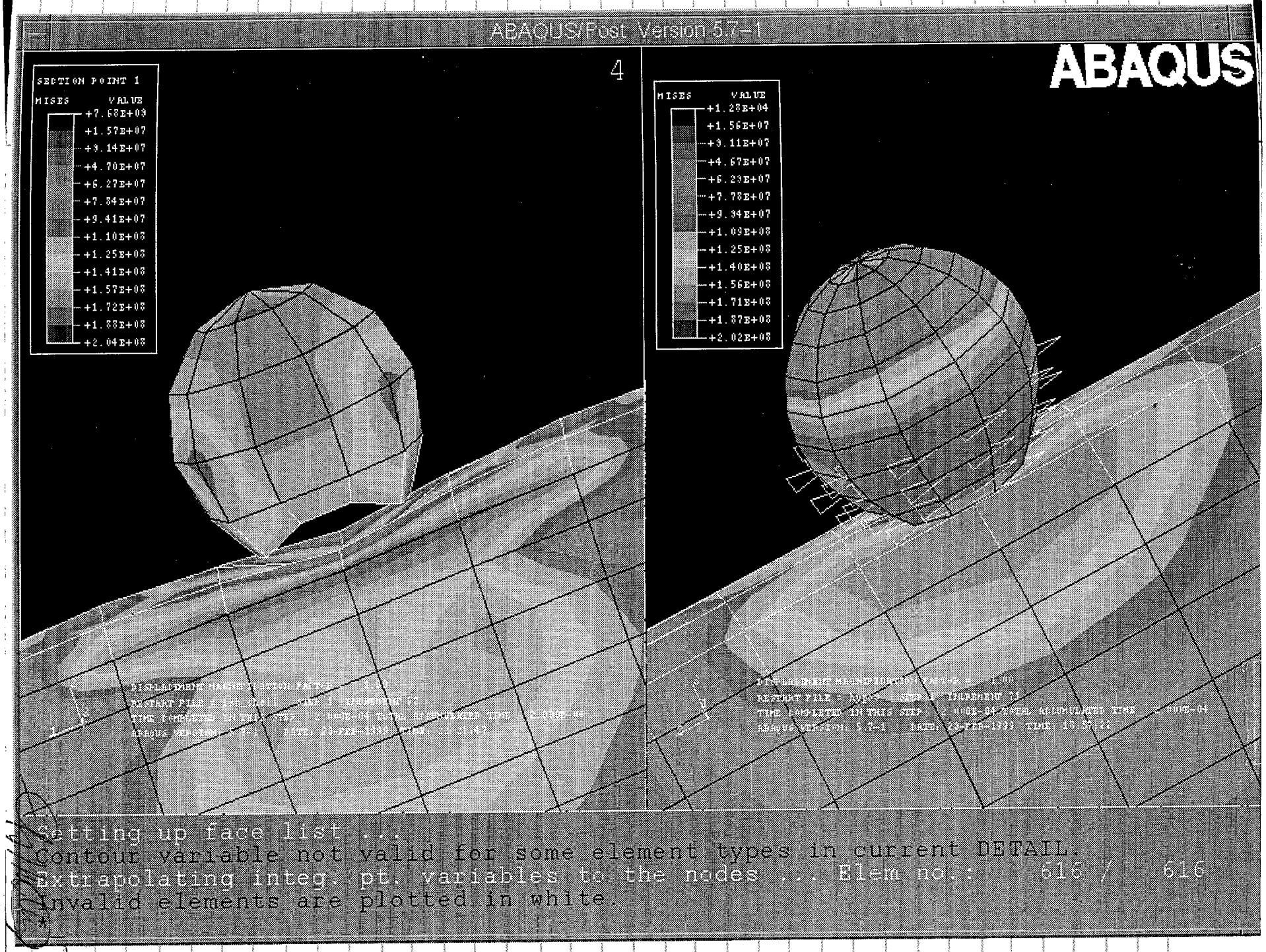
Feb-26-99 15:40 ~ ~~15:50~~ 17:50
- Rebuild the stone mesh and shell model



20 - sweep

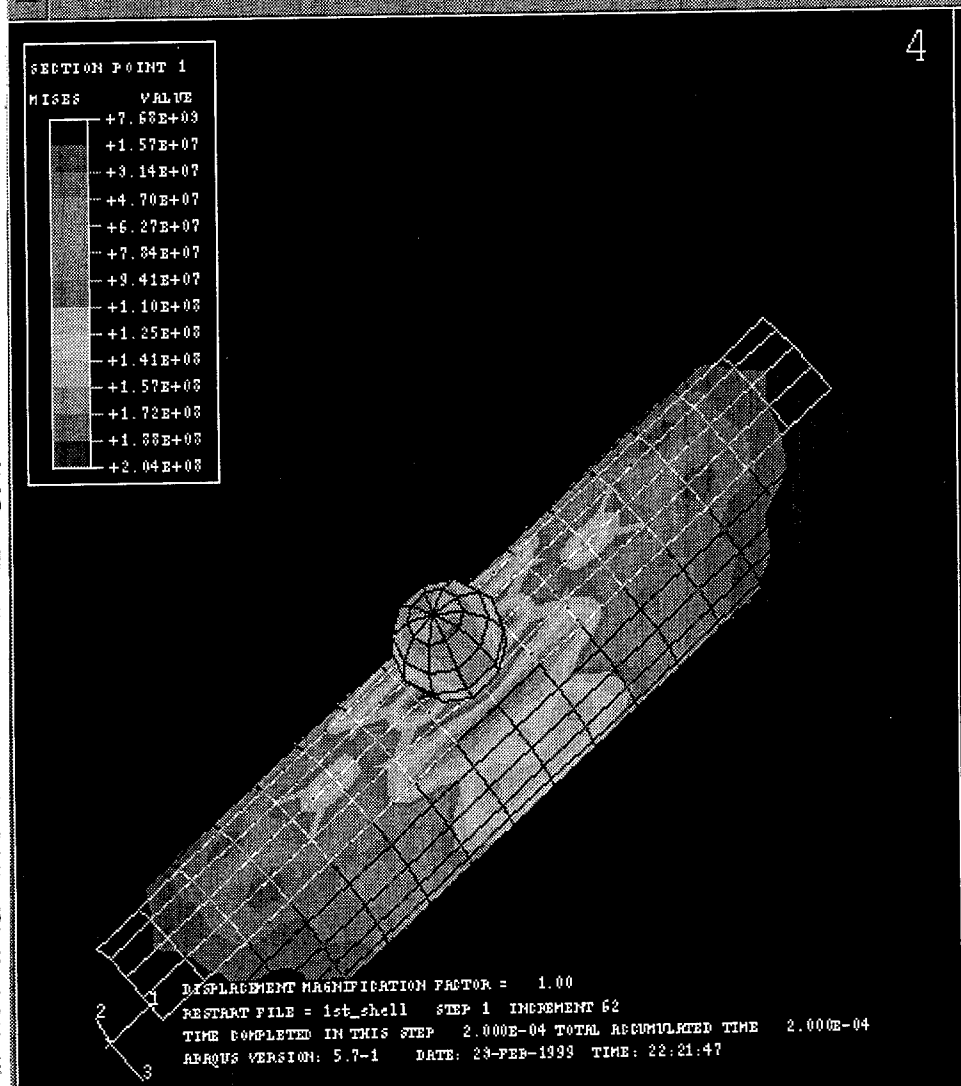
the cylinder is same as the first model.

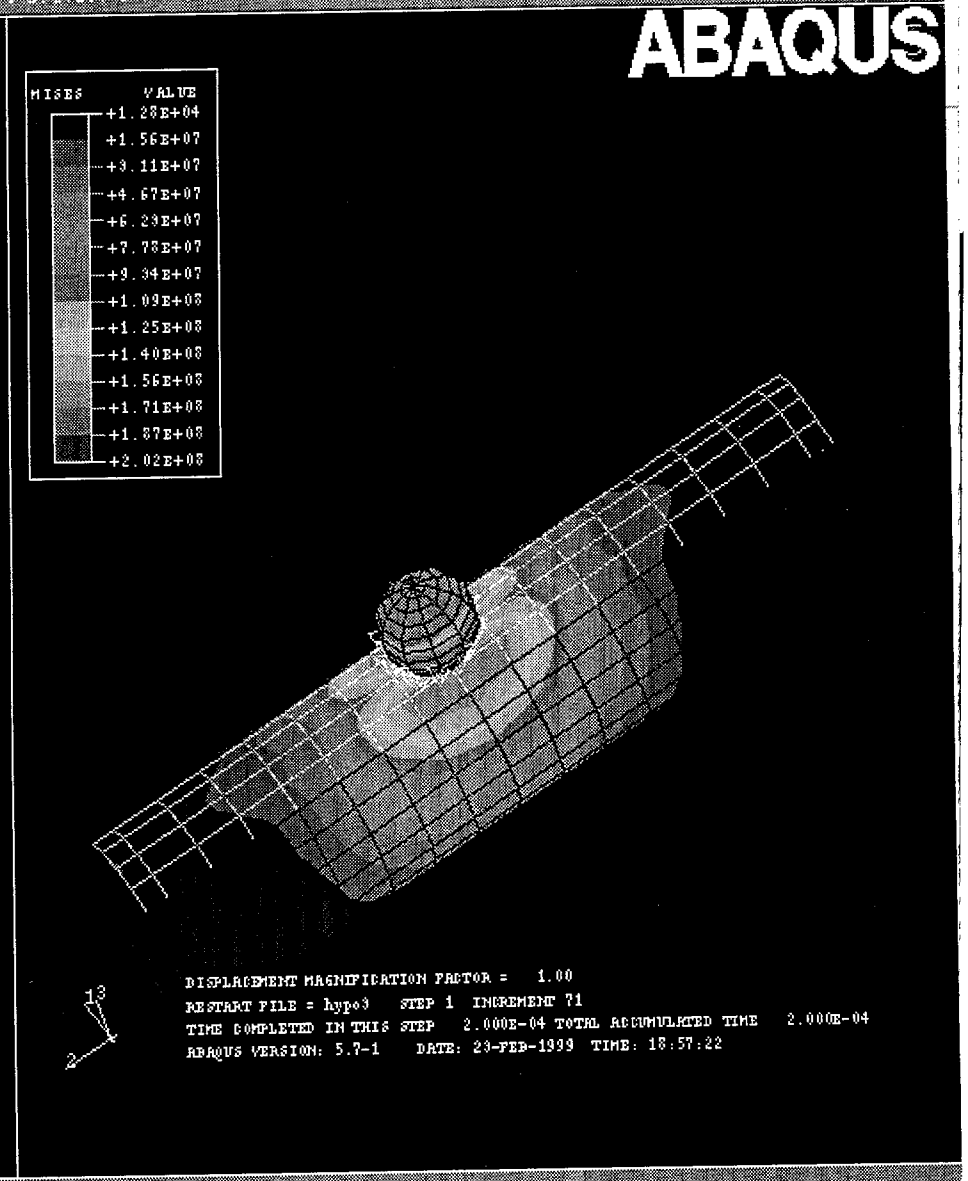
Feb-26-99


ABAQUS/Post Version 5.7-1
ABAQUS
4
SECTION POINT 1
MISES VALUE
+7.68E+03
+1.57E+07
+3.14E+07
+4.70E+07
+6.27E+07
+7.84E+07
+9.41E+07
+1.10E+08
+1.25E+08
+1.41E+08
+1.57E+08
+1.72E+08
+1.88E+08
+2.04E+08
MISES VALUE
+1.28E+04
+1.56E+07
+3.11E+07
+4.67E+07
+6.23E+07
+7.78E+07
+9.34E+07
+1.09E+08
+1.25E+08
+1.40E+08
+1.56E+08
+1.71E+08
+1.87E+08
+2.02E+08
Setting up face list ...
Contour variable not valid for some element types in current DETAIL.
Extrapolating integ. pt. variables to the nodes ... Elem no.: 616 / 616
Invalid elements are plotted in white.

Feb-26-88

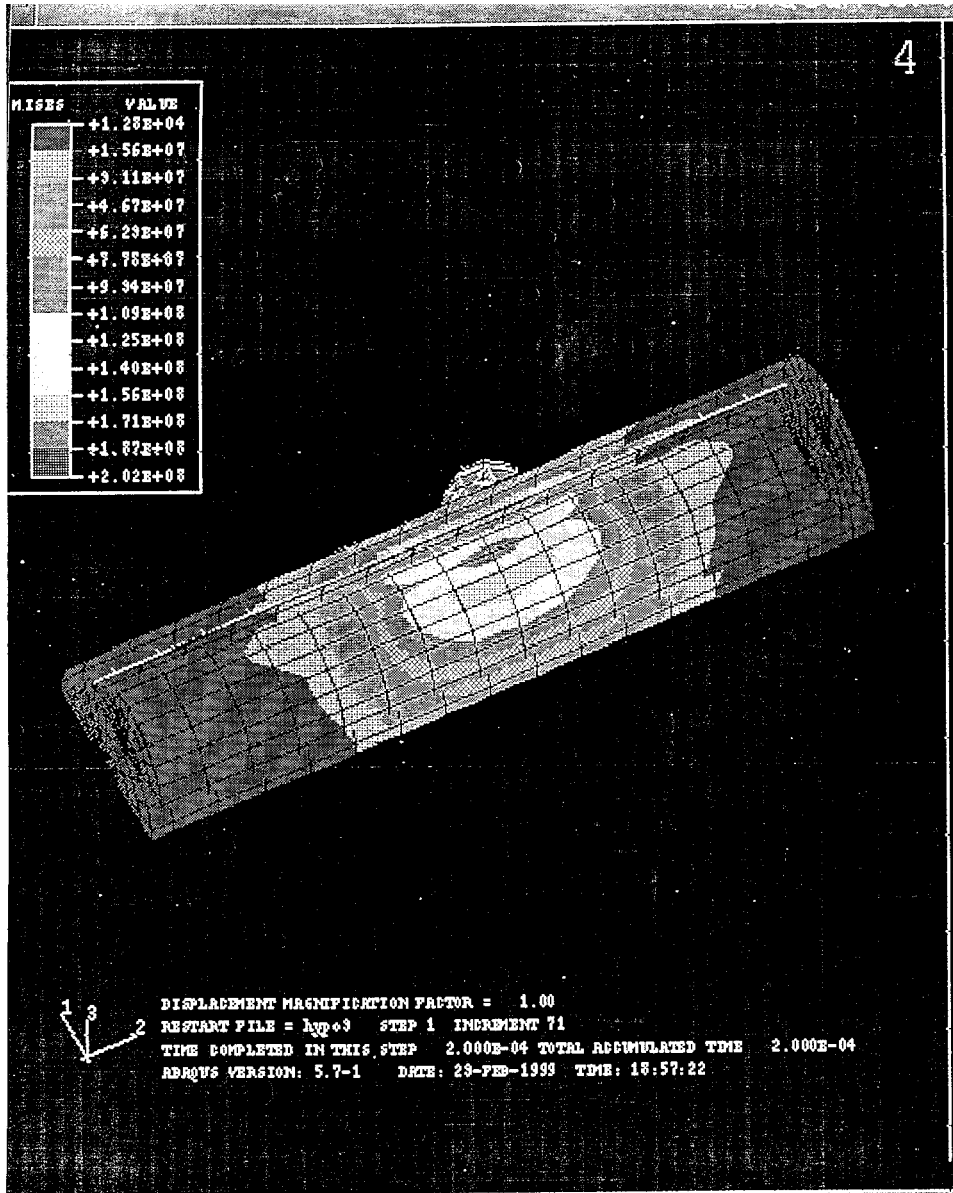
ABAQUS/Post Version 5.7-1

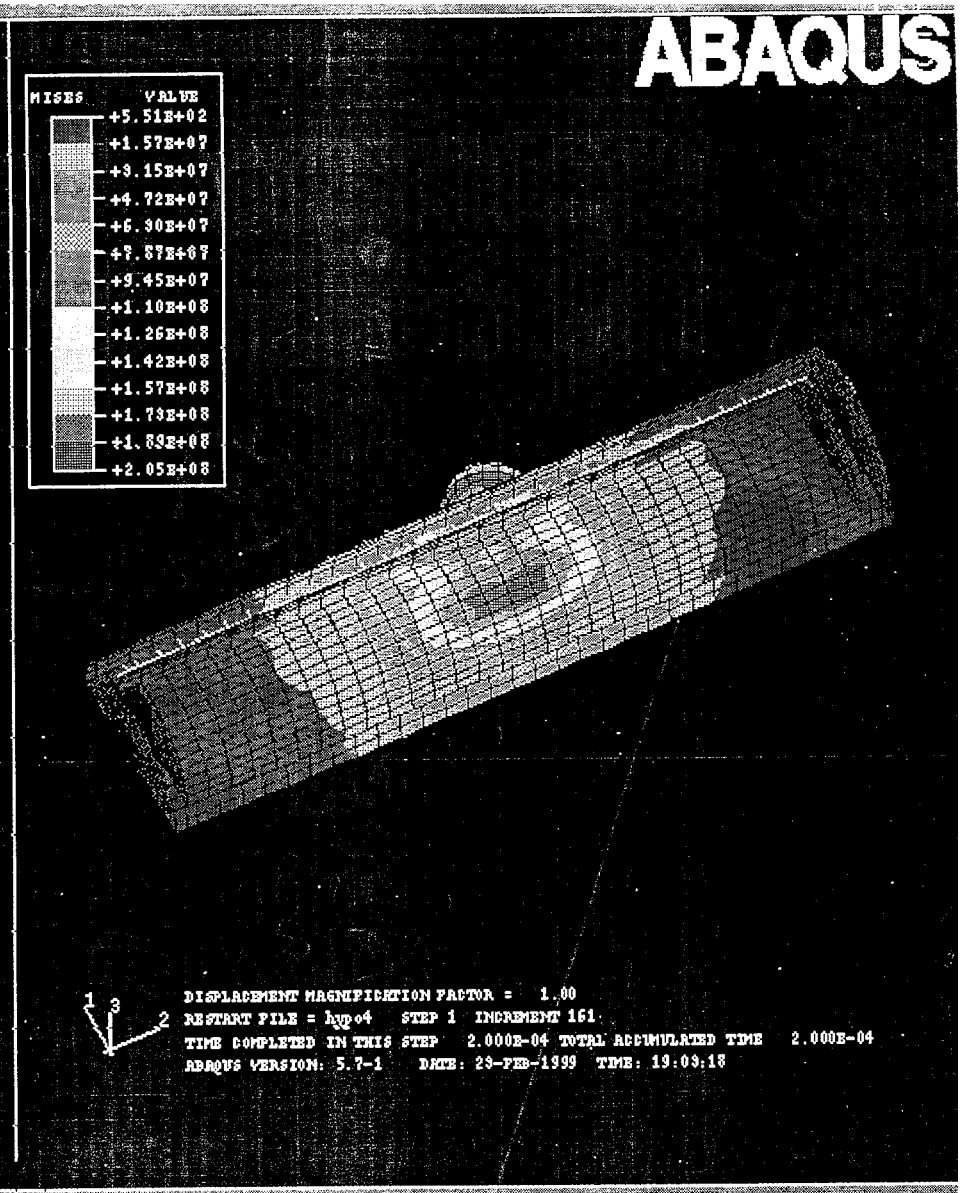





```
Setting up face list ...
Contour variable not valid for some element types in current DETAIL.
Extrapolating integ. pt. variables to the nodes ... Elem no.:   1046 /  1046
Invalid elements are plotted in white.
*
```

Feb-26-99






```
Element : 2076,   MISES   =   4.0970E+06
Element : 2076,   MISES   =   4.0970E+06
Element : 2076,   MISES   =   4.0970E+06
No element found.
```

Feb-26-99

Feb-27-99 20:00 ~ 00:00

- Continuing shell element model. Meet element distorsion error.
- It stopped at small increament of time step. It is because the element distorsion error. It is serious if we have to deal with large deformation. Next time, I should go through a refined mesh to reduce distorsion of element.

Feb-27-99

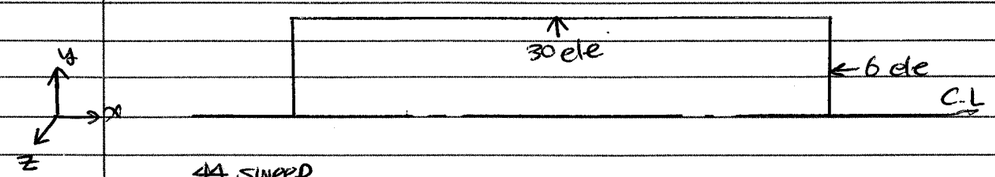
Feb-28-99 15:00 ~ 17:00

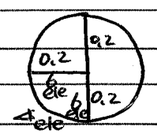
- Refined shell element model. But got same error.

Feb-28-99

20:15 ~ 00:05

- Finish shell element. Test same geometry with solid element.





20 sweep

- The Abaqus license expires at midnight.

Feb-28-99

Mar-1-99 11:30 ~ 12:35

- Meeting with Dr. Krauthammer.
- Report
  - Abaqus license expires.
  - Distorsion error of the shell element.
  - Shell element change didn't make any difference

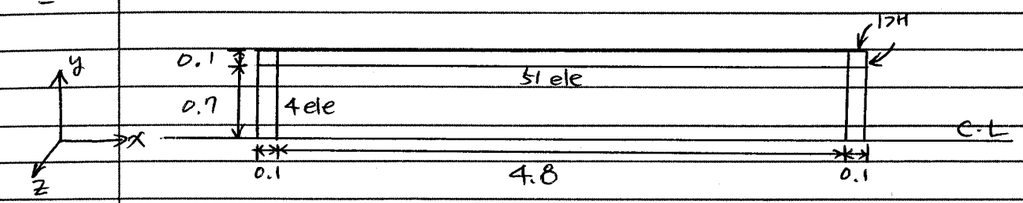
  - except stress distribution
- Decide : make a list for modeling
- 3D Solid : where the impact occurrs.

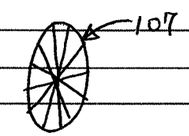
  Check it.
- Make mesh (6,000) outer barrier only. Up to 50 ms. Storage check!

Mar-1-99

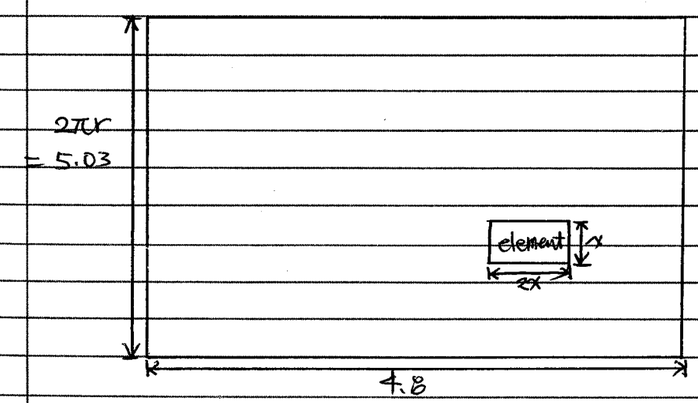
Mar-3-99 21:00 ~ 00:00

- Start 6,000 elements model.
-





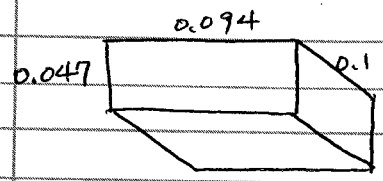
Element calculation.



$2n\,x^2 = 5.03 \times 4.8$ n = Assume 5,500

# of element

$$x = \sqrt{\frac{5.03 \times 4.8}{5500 \times 2}} = 0.047$$

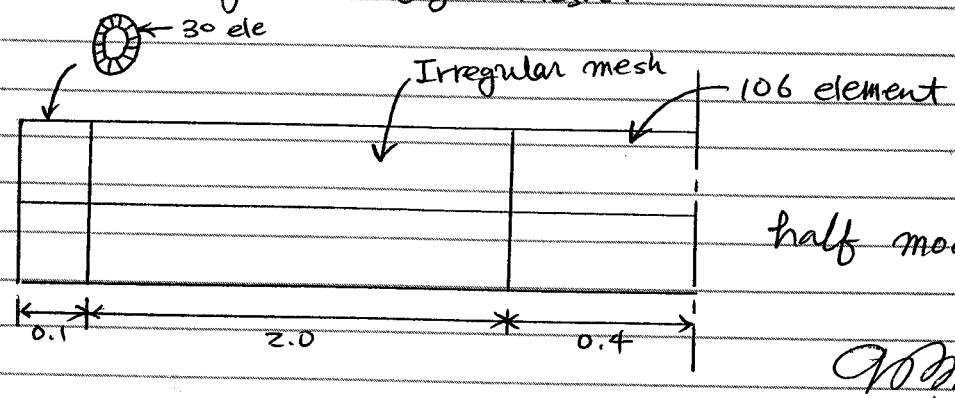


about 1 : 2 : 2.13

Mar-3-99

Mar-4-99 21:00 ~ 01:00

- Make an irregular size mesh.

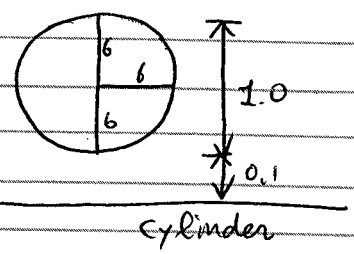


half model

Mar-4-99

Mar-5-99 13:00 ~ 15:00

- Make the stone. Got surface error



20 sweep.

Mar-5-99

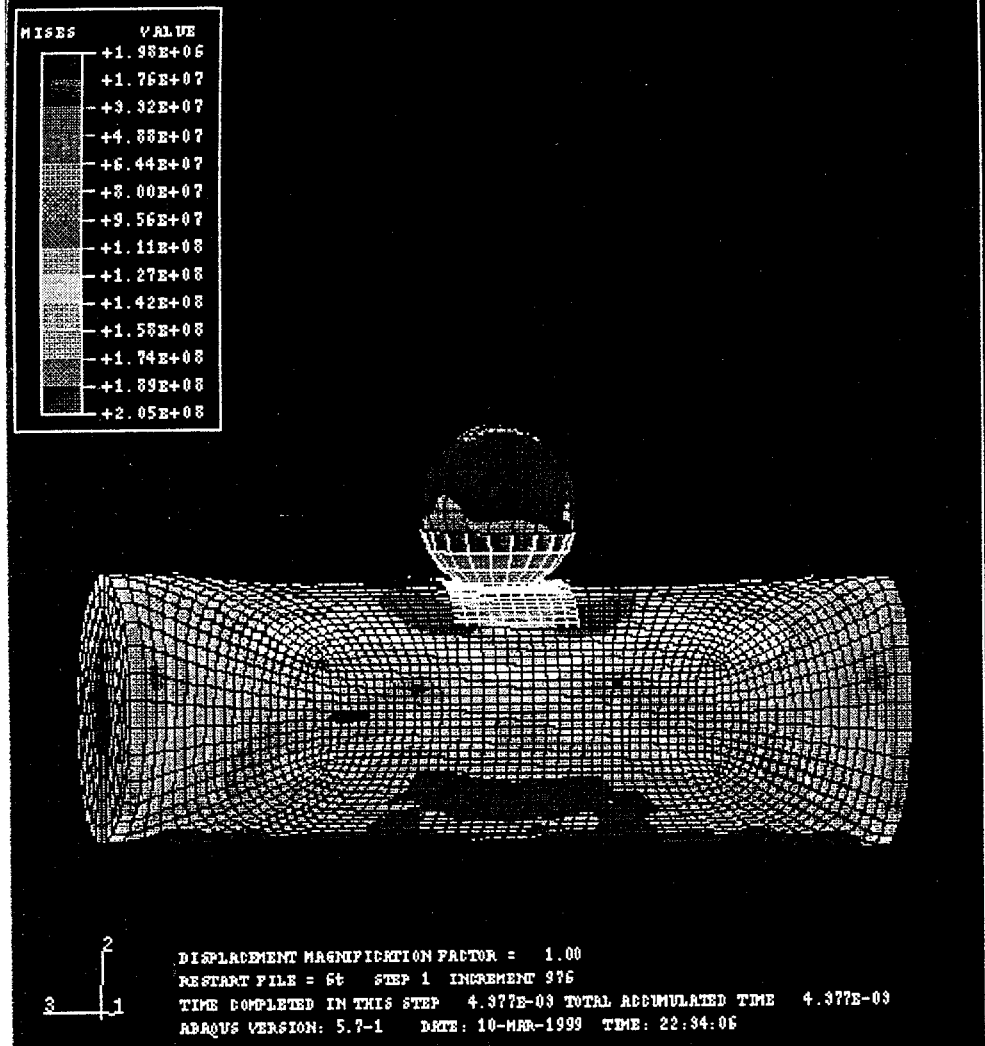
Mar-7-99 20:00 ~ 01:00

- Fix surface error. Test model up to 50 ms.
- Took two hours.

Mar-7-99

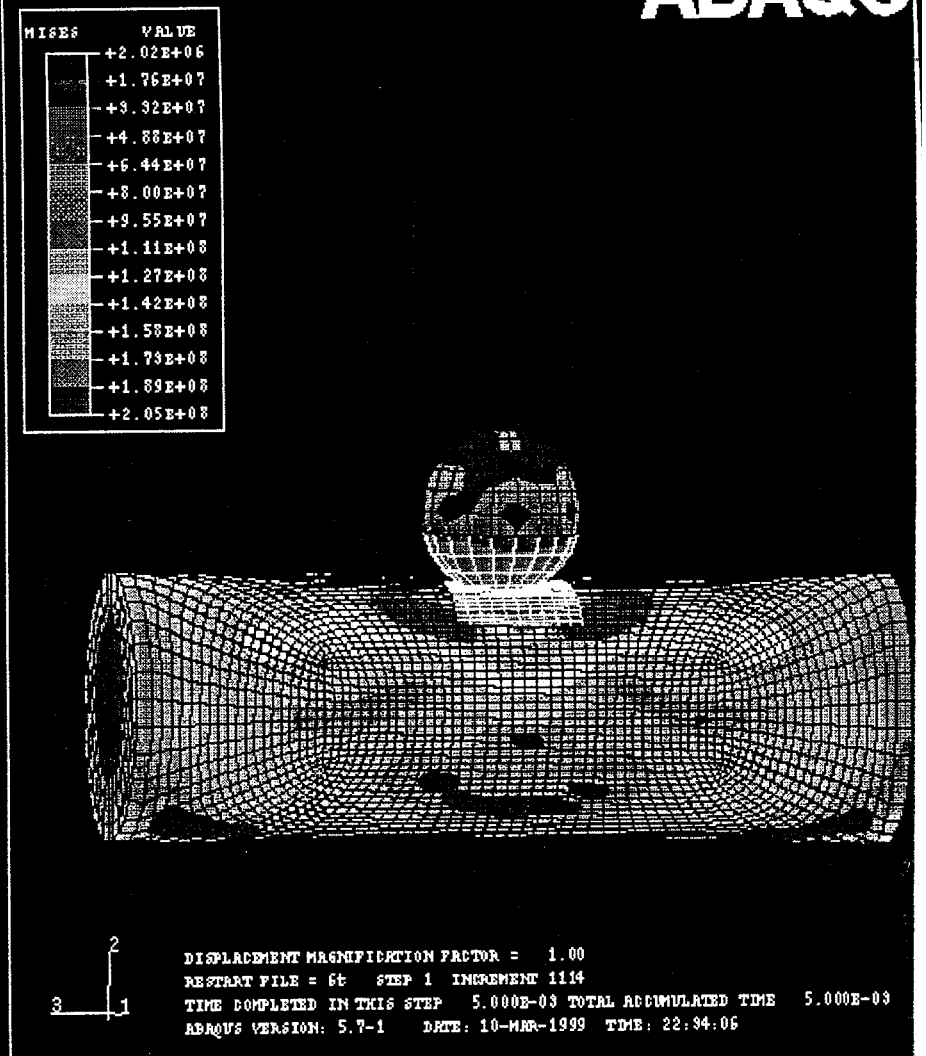
ABAQUS/Post Version 5.7-1

4



0.0043775

ABAQU



0.005 s

Mar-7-99

Mar-8-99 14:30 ~ 16:45

- Test up to 5 ms with 10 restarts.
- 10 restarts require 99% of disk storages.
- The stone speed was 50 m/s (initial)
- # of element : 6756
- # of nodes : 12449

Mar-8-99

Mar-10-99 22:00 ~ 03:05

- Test up to 5 ms with 8 restarts.
- 8 restarts require 98% of capacity.
- Stone speed was 50 m/s (initial)

- Test up to 5 ms with 20 restarts.
- Remains of disk capacity = 33% (I've erased Patran files)
- Remains of disk when calculation was finished. = 91%
- Remain of disk after converting
  → Disk capacity exceed error.

- Begin with 12 restarts up to 5 ms (33%)
- Calculation was finished. (72%)
  begin to convert
- After converting, disk capacity was 100%
- Begin with 12 restarts up to 10 ms (33%)
  → Error! element distorsion error.

- Change stone speed down to 25 m/s
- Restart test with 12 restarts up to 10 ms (33%)
- After calculation, used disk space was 72%
- After converting, " " " 100%

Mar-10-99

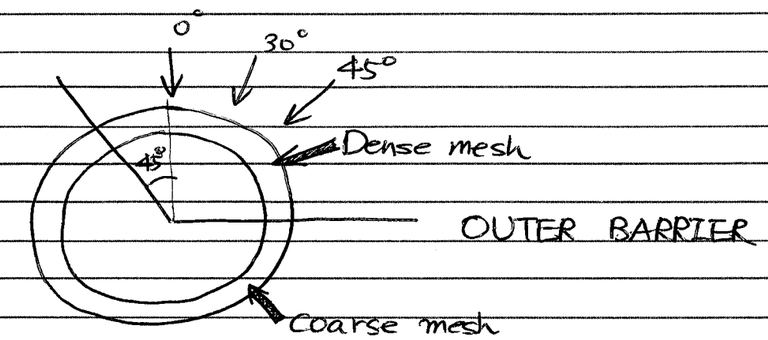
Mar-17-99 15:00 ~ 16:00

- Discussion with Mr. Oh

Mar-17-99

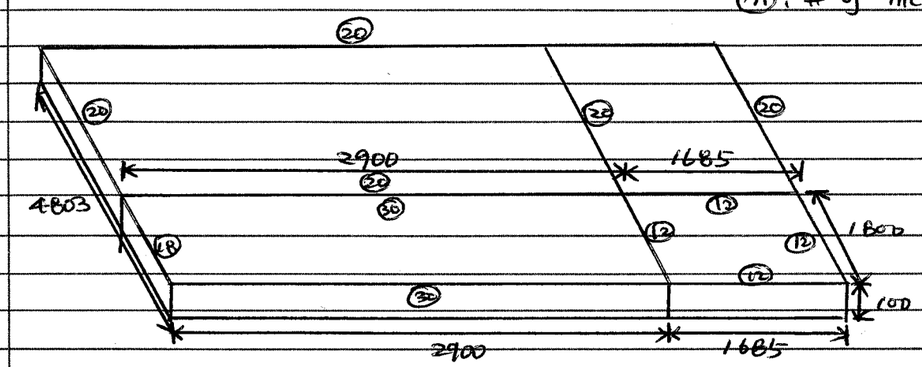
Mar-18-99 21:00 ~ 01:15

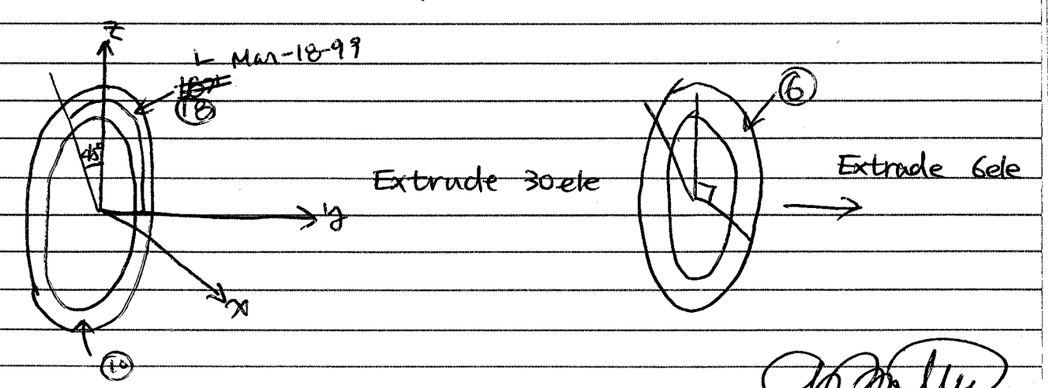
- Calculate the number of elements of the outer barrier.
- " the size of " of the outer barrier



- Outer barrier —

(n) : # of mesh seed.



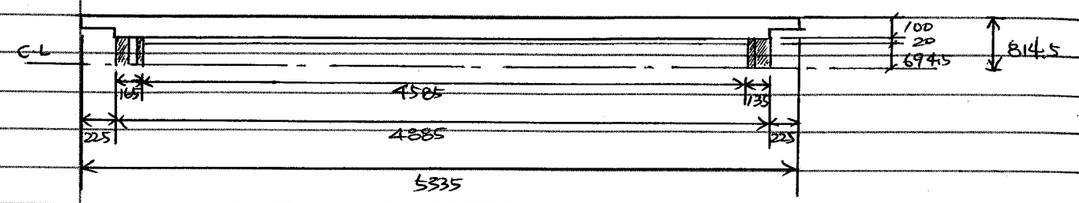


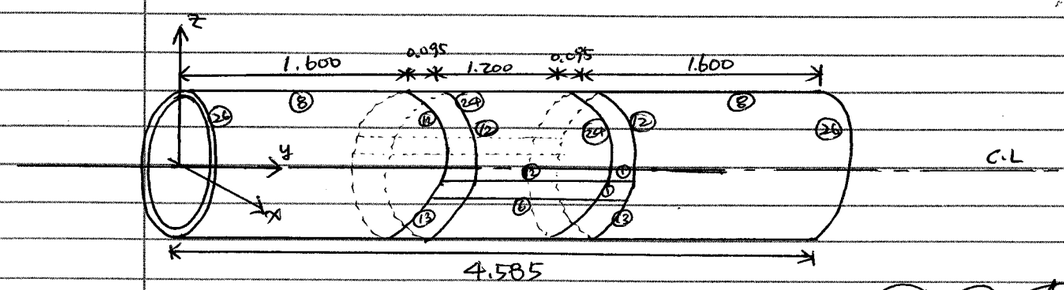
Mar-18-99

Mar-19-99 14:30 ~ 15:35
- Meeting with Dr. Krauthammer.
- Element transition is required.

Mar-19-99

Mar-19-99 22:00 ~ 02:40
- Analyse and design of geometry of the outer barrier.





Mar-19-99

Mar-20-99 21:00 ~ 22:00
- Recalculate the # of element

Mar-20-99

Mar-21-99 14:30 ~ 15:35
- Meeting with Dr. Krauthammer

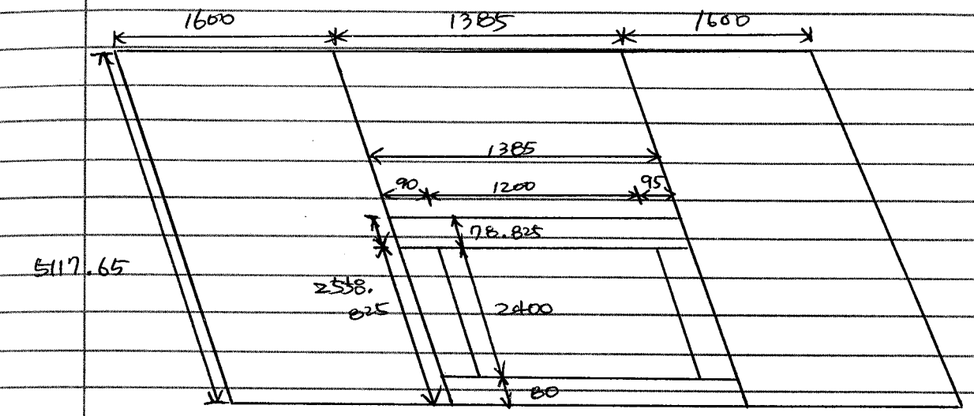
Mar-21-99

Mar-21-99 22:00 ~ 00:15
- Try a geometry of the outer barrier. Failed.

Mar-21-99

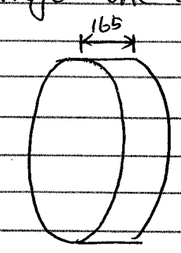
Mar-21-99

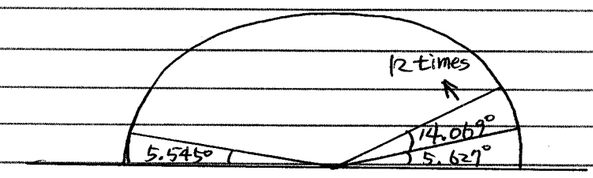
Mar-22-99 15:00 ~ 18:00
- Try a geometry of the outer barrier



Mar-22-99

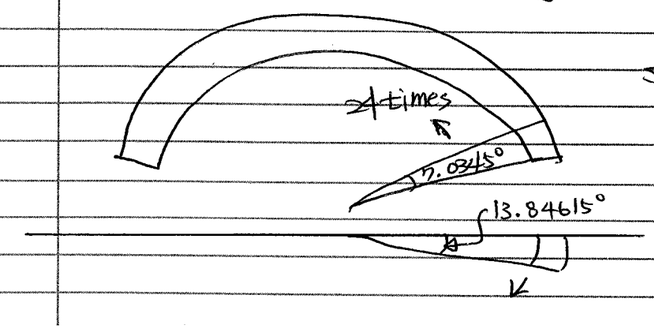
Mar-22-99 20:00 ~ 22:30
- Change the dimension and solid. Start over.



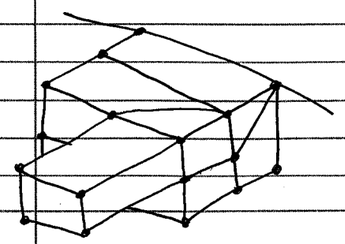


Mar-22-99

Mar-23-99 15:10 ~ 17:55
- Build the fine mesh of the outer barrier.



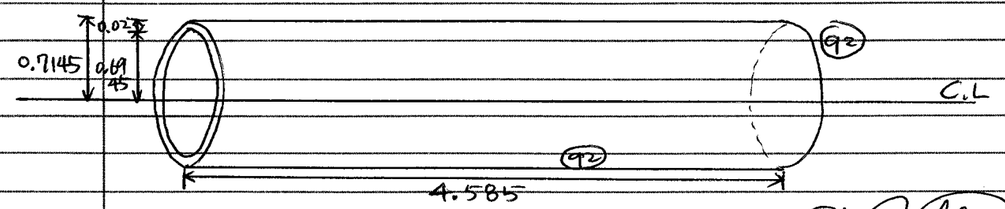
surface of the fine mesh



Mar-23-99

22:00 ~ 01:00

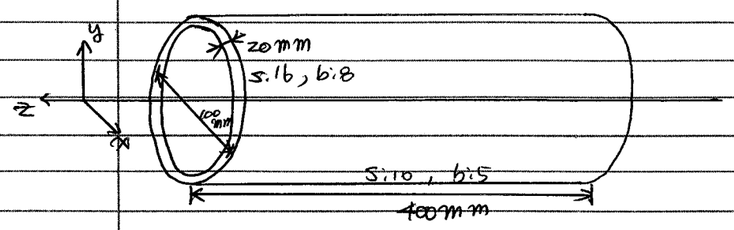
- Start the inner barrier



Mar-23-99

Mar-25-99 15:00 ~ 18:00

- Build 1:2:2 and 1:4:4 aspect ratio test model



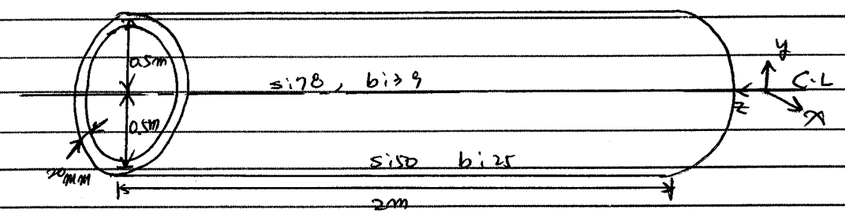
s: small
b: big

I think it is too rough especially for the diameter. Two models are much different with results. I should go to more refined mesh to test again.

Mar-25-99

20:00 ~ 01:00

- Begin the second test of the aspect ratio test models



E: 206E9
ν: 0.3
ρ: 8131
σy: 206E6

Mar-25-99

Mar-26-99 14:50 ~ 16:00

- Meeting with Dr. Krauthammer.
- Thick shell
- half model
- Gravity (effect of gravity), test twice.
  without gravity
  get to steady-state
- Summary

Mar-26-99

Mar-27-99. 14:00 ~ 18:00

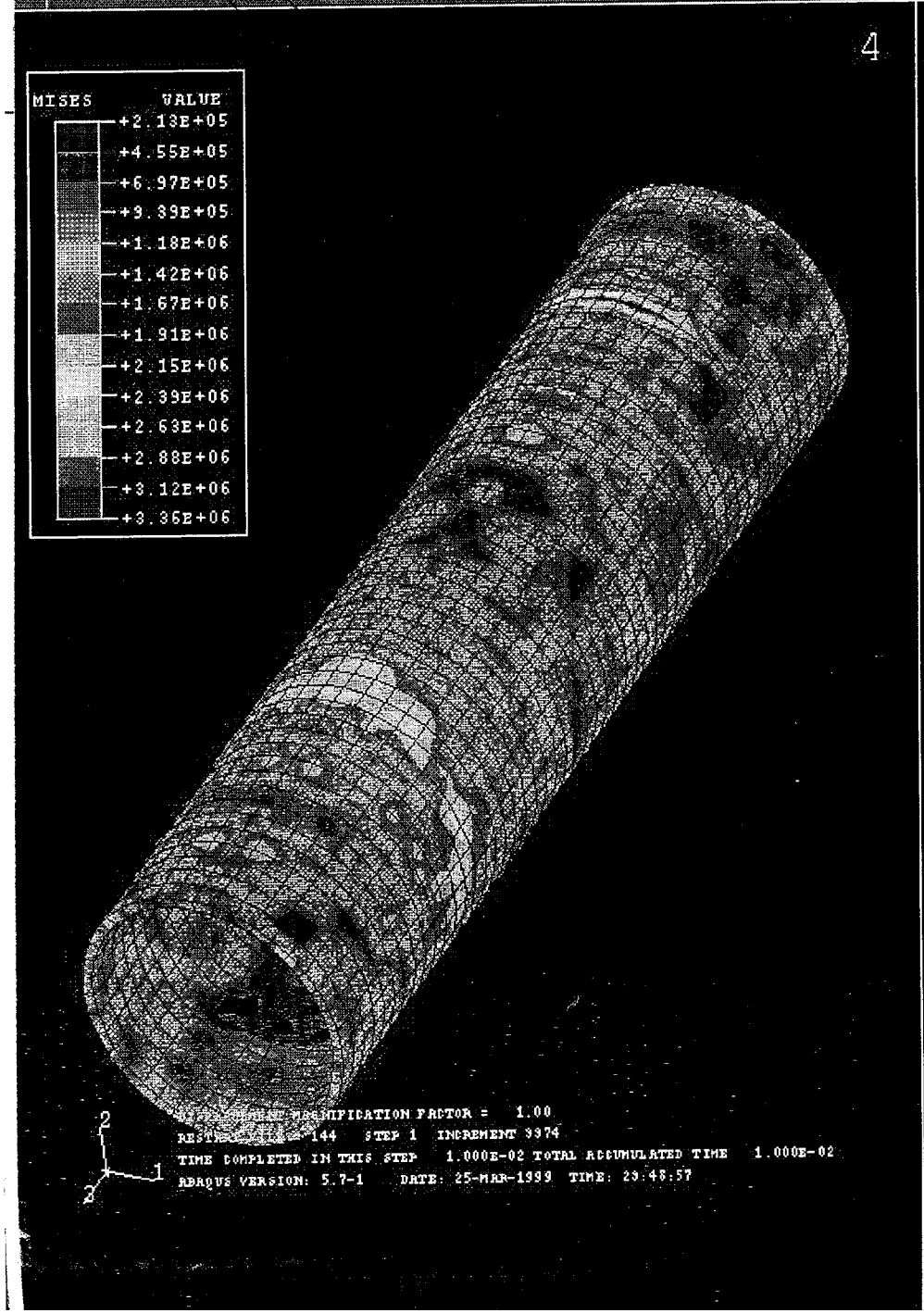
- Check the gravity option of Abaqus.
  Test the second model with gravity optim.
- DLOAD

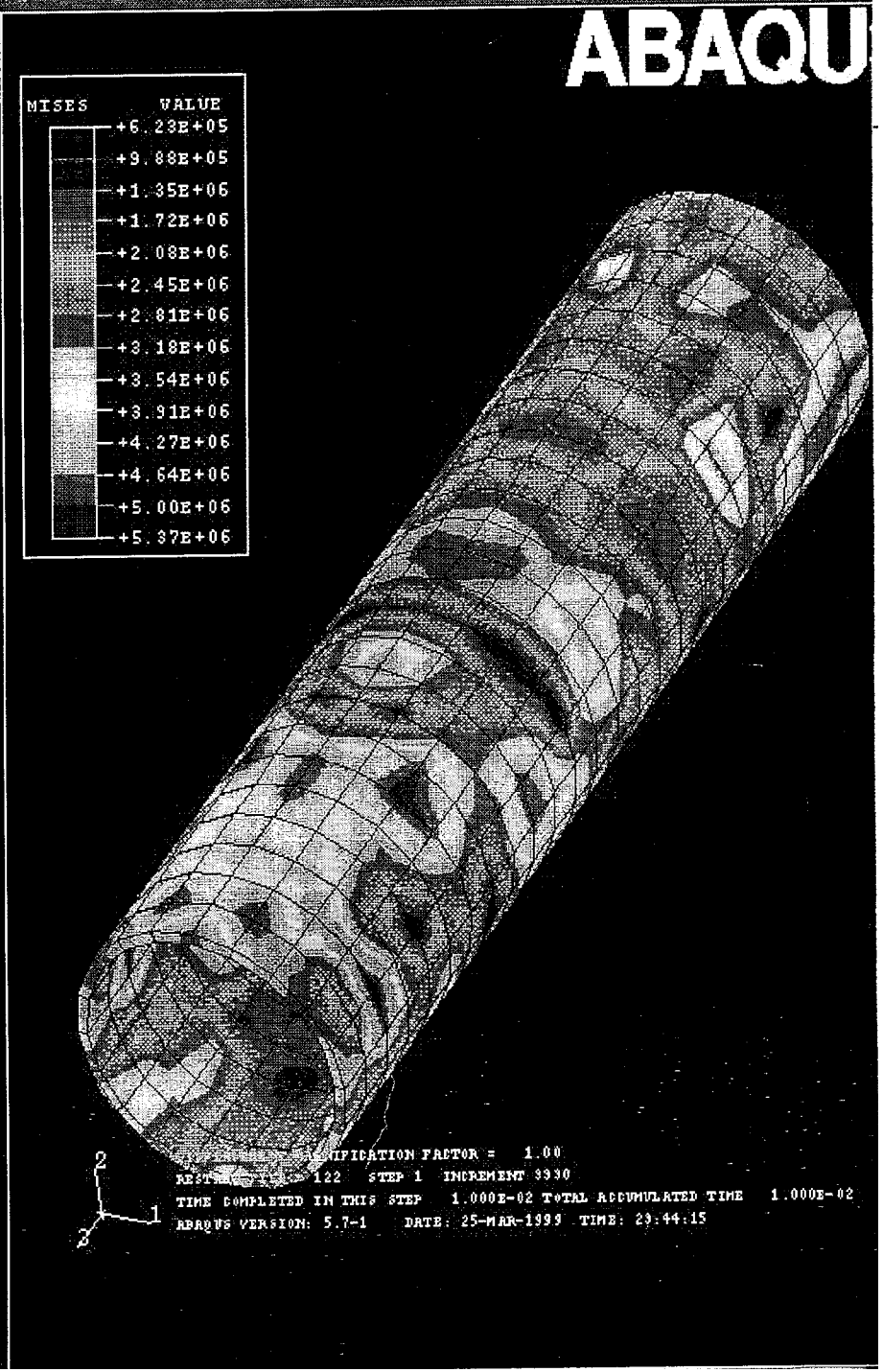
  WHOLE, GRAV, 9.81, 0, −9.81, 0

  Element set; Gravity keyword; Total gravity; x-component; y-component; z-component

Mar-27-99

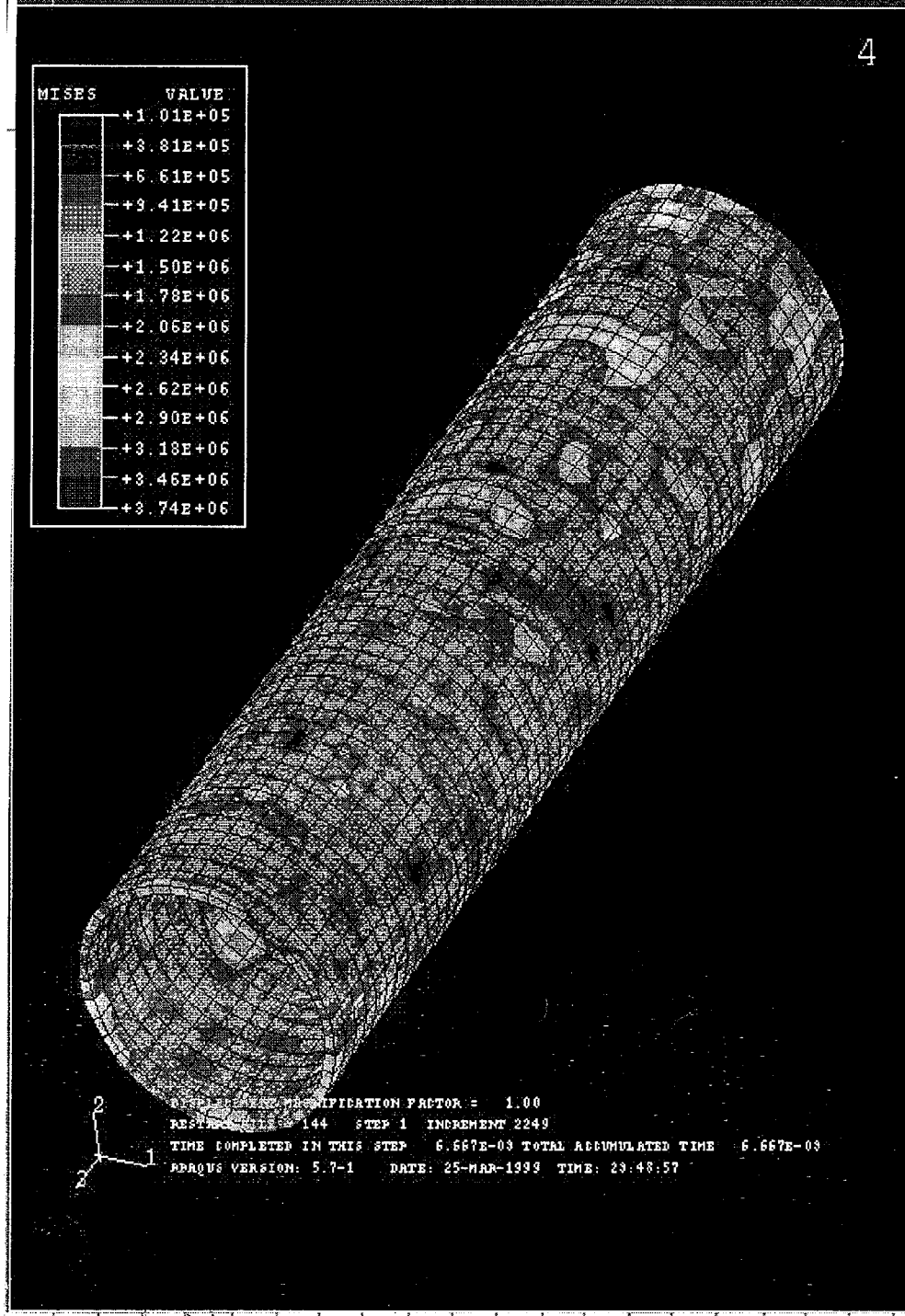
ABAQUS/Post Version 5.7-1

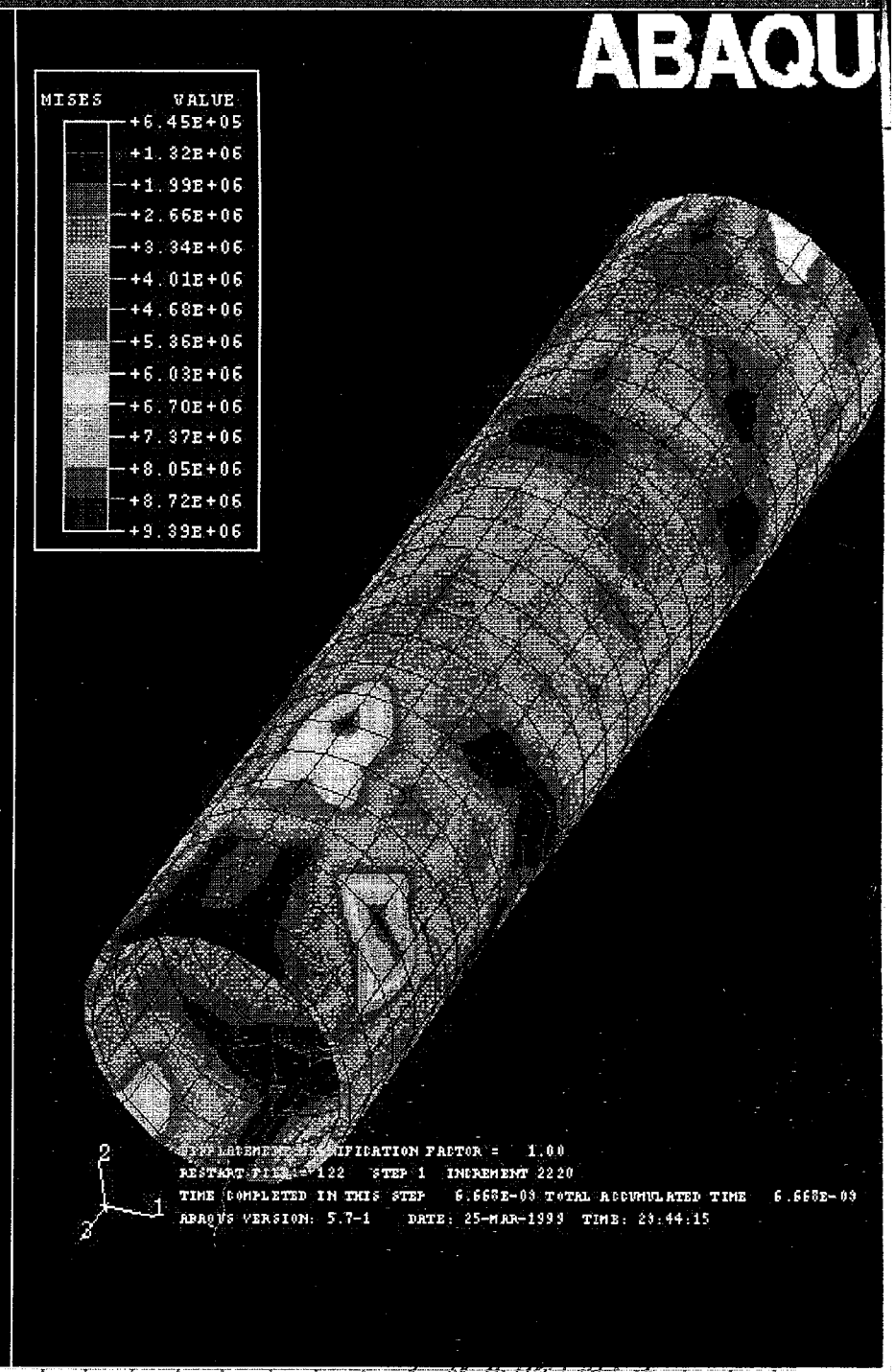




ABAQUS/Post Version 5.7-1

4

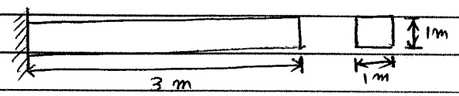




Mar-27-99

Mar-28-99 14:00 ~ 18:30

- Check gravity option with simple case.



$\rho = 1000\,kg/m^3 / 9.81$

$E = 3\times10^8\ kg/m^2$

$I = \frac{bh^3}{12} = \frac{1}{12}$

Impose only gravity acceleration $G = 9.81\ m/s^2$

Theoretical solution

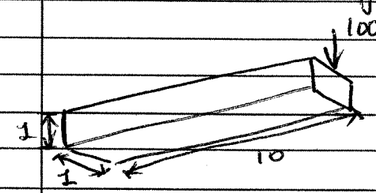
$$\delta = \frac{qL^4}{8EI} = \frac{1000\times3^4}{8\times3\times10^8\times\frac{1}{12}} = 4.05\times10^{-4}\ m$$

48 elements of C3D8R gives 87% of real displacement, 1024 elements model gives 91%. Something's ~~wrong~~ wrong.

Mar-28-99

Mar-30-99 22:00 ~ 01:00

- Keep testing case study.
- C3D8R is not so good.



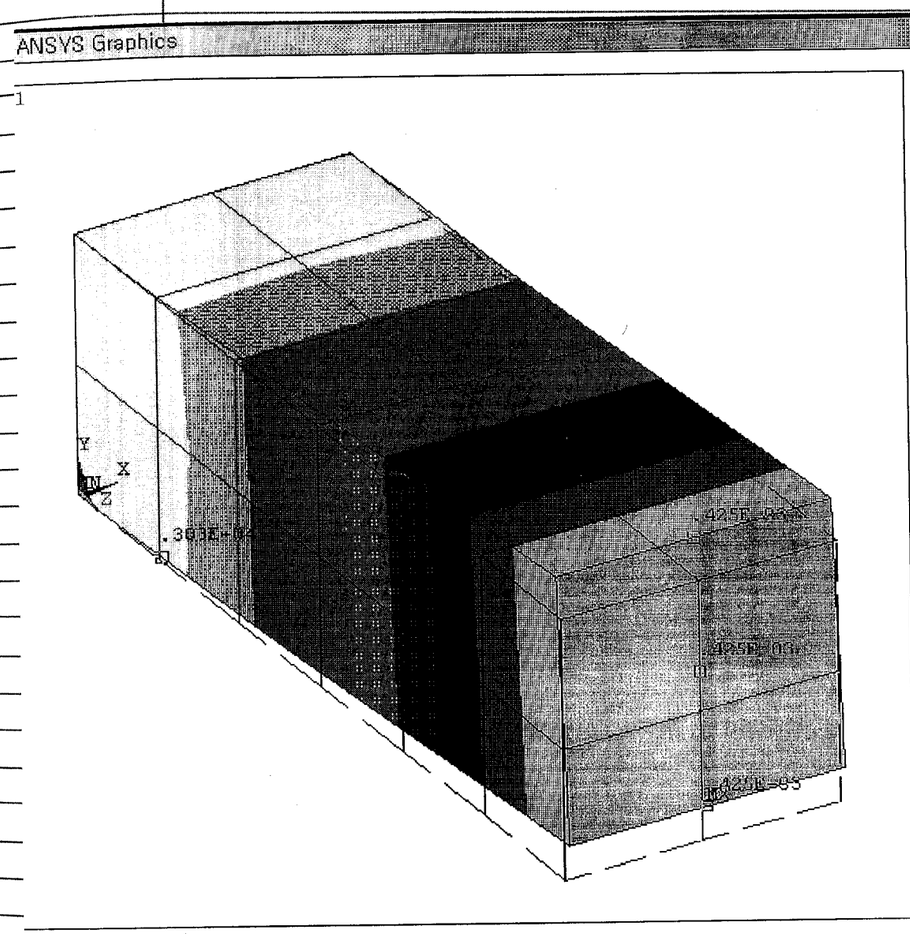
40 element of C3D8R just give 75% of real displacement. It's not good!

Mar-30-99

Mar-31-99 21:00 ~ 01:45

- Compare with the results of Ansys. Check element.

Mar-31-99



Mar-31-99

Simple report for Abaqus about gravity and concentrated load.

Introduction

I have tested simple cantilever cases applied by gravity and concentrated load separately. However, I have got strange results from Abaqus comparing with theoretical solution. Followings are detail.

Model Dimension

$1 \times 1 \times 7m$ Block.
Young's Modulus : 300,000,000 pascal
Moment of Inertia : $\frac{1}{12}m^4$
Density : 101.94 kg/m^3
Gravity Acceleration : 9.81 m/s^2

Theoretical Solution

- Deflection by distributed load

$$\delta = \frac{wl^4}{8EI} = \frac{1000 * 7^4}{8 * 300{,}000{,}000 * \frac{1}{12}} = 0.012005m$$

where,

$$w = 9.81 \times 101.94 \cong 1000$$

Result of gravity

Element Number	Element Type	Element Name	Result	Compare with theoretical Solution
10	Beam	B21	0.012189	1.5% larger
7	3D Solid	C3D8R	0.8491	7,000% larger
56	3D Solid	C3D8R	0.016095	34% larger
448	3D Solid	C3D8R	0.012983	8.1% larger
1512	3D Solid	C3D8R	0.012536	4.4% larger

Mar-31-99

Theoretical Solution

- Deflection by concentrated load to the end of cantilever beam

$$\delta = \frac{Pl^3}{3EI} = \frac{1000 * 7^3}{3 * 300{,}000{,}000 * \frac{1}{12}} = 0.004573m$$

where,

$$P = 1000kg \cdot m/s^2$$

Result of concentrated load

Element Number	Element Type	Element Name	Result	Compare with theoretical Solution
10	Beam	B21	0.0046143	0.9% larger
7	3D Solid	C3D8R	0.3218	7,000% larger
56	3D Solid	C3D8R	0.0059389	29.9% larger
448	3D Solid	C3D8R	0.0049885	9.1% larger
1512	3D Solid	C3D8R	0.0047930	4.8% larger

Comment

1. According to the theory, a finite element models are stiffer than a real model. However, all these finite models showed larger deflection than theoretical solution. I am not sure why this happened.
2. This is a really simple model. However, solution of finite element models are approaching the exact solution too slow. If this is caused by element type, I hope you recommend a good one.

I am sending the input file of 56 elements model also. Thank you.

Mar-31-99

Dear Prof Krauthammer,

I reviewed the input files and the results you sent me.

C3D8R has only 1 integration point located at the centroid of the element. So it suffers from a numerical problem called hourglassing. It is possible for it to distort in such a way that strains calculated at the integration point are all zero, which leads to uncontrolled distortion of the mesh. This is what you are seeing in your results. If reduced integration first order elements are used for bending problems, atleast 4 elements must be used through the thickness of the beam.

A better choice of elements would have been to use incompatible mode elements, like C3D8I. This element has additional degrees of freedom to enhance its deformation gradient. These enhancements are entirely internal to the element and are not associated with the nodes. These elements can produce results that are comparable to quadratic elements at a significantly lower cost. I replaced the C3D8R elements in the two input files you sent me with C3D8I elements, and re-ran the analysis. The resultant maximum deflections are in error by approximately 1%. However, these elements must be used only in problems where the element distortions are small.

An alternate choice of elements for the problem would be to use quadratic reduced integration elements, C3D20R. They should be used for general analysis work unless large strains are involved. They too exhibit some hourglassing, but these modes do not propagate through the mesh. However, if the problem involves large distortions, a fine mesh of linear reduced integration elements, (like C3D8R) is the most appropriate.

Please refer to section 14.1.1 for more information on continuum elements and their selection.

In the future, please provide your academic support ID if you have purchased

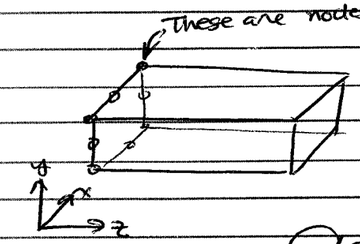
Mar-31-99

Apr-1-99

- Check the symmetry condition of Abagus and apply outer barrier
- Example of symm.

  *Boundary

  Middle, zsymm



Apr-1-99

Apr-3-99 22:00~01:00

- Write a simple report about strange result of Abagus of simple case.

Apr-3-99

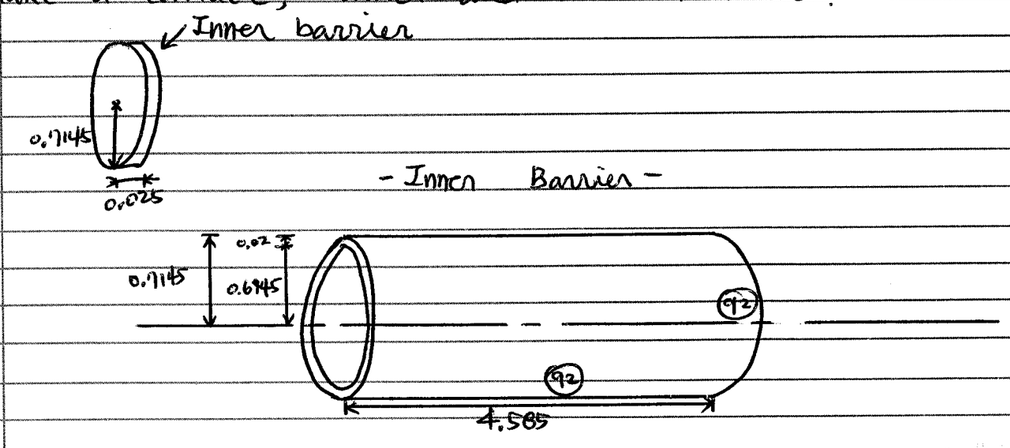
Apr-7-99 14:30~18:30

- Correct a report with gravity and concentrated load case.

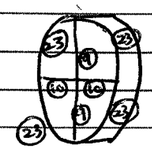
Apr-7-99

Apr-8-99 21:00~01:00

- Make a contact, inner and outer barrier.



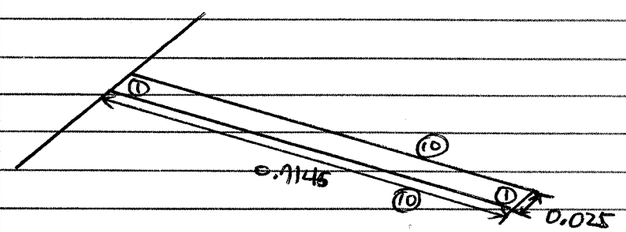
- Inner lid -



Apr-8-99

Apr-11-99 21:00 ~ 01:00

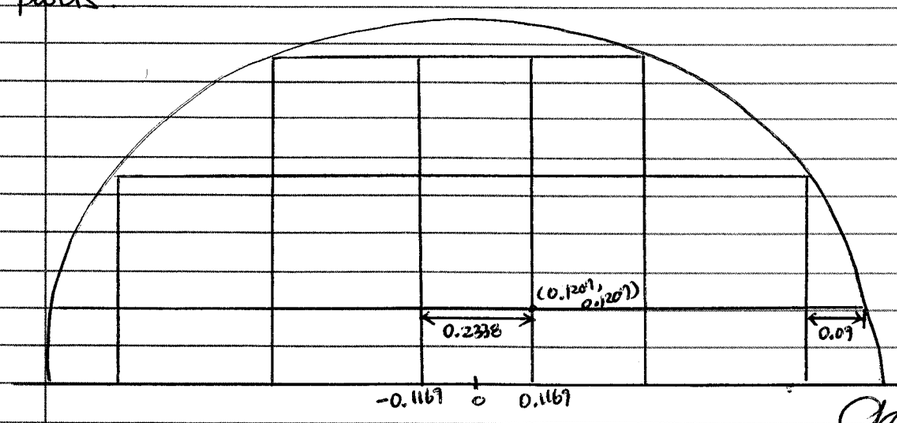
- Rebuild the edge of outer barrier and outer & inner lid.



Apr-11-99

Apr-12-99 21:00 ~ 01:00

- Finish the barriers and lids. Start the inner parts.



Apr-12-99

Apr-13-99 22:00 ~ 02:00

- Build the inner part of the model.

Apr-13-99

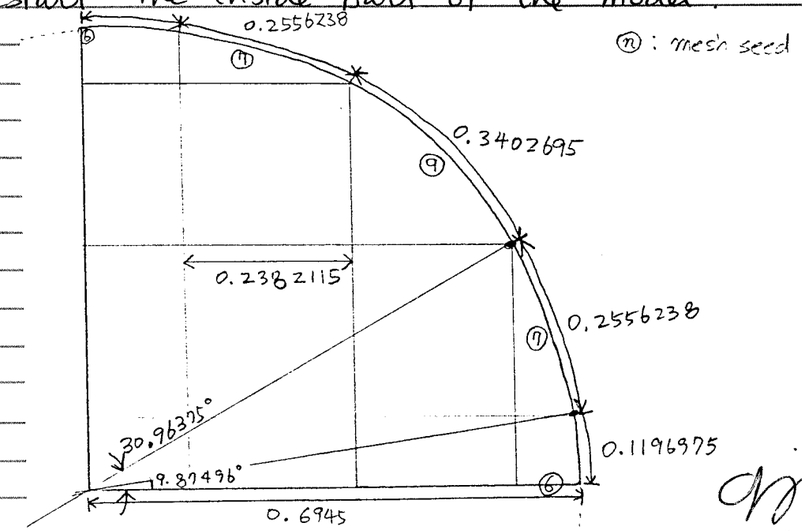
Apr-14-99 22:00 ~ 00:10

- Check the contact condition and redesign the mesh of the inside.

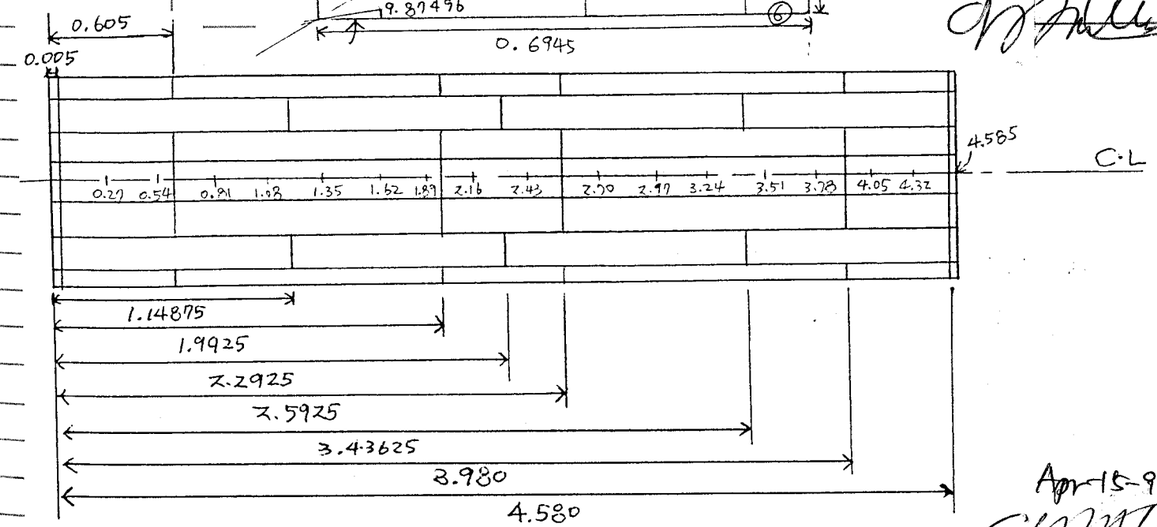
Apr-14-99

Apr-15-99 21:00 ~ 01:00

- Restart the inside part of the model.



Apr-15-99



Apr-15-99

Apr-18-1999 20:00 ~ 00:00
- Inside. Moved shell elements to extrude. Finish the element in the diaphragm.

Apr-18-99

Apr-19-99 20:00~00:00
- Finish the inside of the model. However, contact problem between inner and outer barrier occurs.

Apr-19-99

Apr-20-99 14:30~15:30
- Meeting with Dr. Krauthammer.

Apr-20-99

21:00~ 00:15
- Change the inner barrier to avoid contact problem. And change the diaphragm to fit the inner barrier.

Apr-20-99

Apr-21-99. 20:00~ 00:00
- Continue a diaphragm

Apr-21-99

Apr-22-99. 21:00 ~ 01:30
- Finish a contact problem. Get a property error.

Apr-22-99

Apr-23-99 15:00 ~ 18:00
- Work on a property error.

20:00 ~ 22:15
- Work on a property error.

Apr-23-99

Apr-24-99 22:00 ~ 02:05
- Fix a property problem. Try to fix a distorted element error.

Apr-24-99

Apr-26-99 15:00 ~ 18:00
- First test drive of our model without diaphragm part.

21:00 ~ 00:15
- Remake the diaphragm to fit the inner barrier.

Apr-26-99

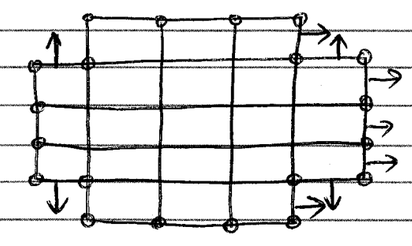
Apr-27-99 15:00 ~ 17:55
- Finish the diaphragm part and equivalencing
- Equivalence Group.
  1. mesh_in_assembly
     mesh_in_spacer
  2. mesh_in_diaphragm
     mesh_in_thermal_shunt
     mesh_inner_bar
  3. mesh_edge_outer_bar
     mesh_outer_bar
     mesh_transi_outer_bar.
     mesh_inner_lid
     mesh_outer_lid.

Apr-27-99

21:00 ~ 00:00
- Begin a half model.

Apr-27-99

Apr-28-99 15:00 ~ 17:10
- Succeed the first half model without diaphragm contact.
- Change contact and equivalence.
- Direction of master face of assembly.



o: equivalenced.

Apr-28-99

21:00 ~ 00:15
- Work on contact errors of diaphragm

Apr-29-99

Apr-29-1999 16:00 ~ 18:00
- Work out contact error of diaphragm

Apr-29-99

22:00 ~ 00:15
- Test the half model and begin the symmetry boundary condition

Apr-29-99

May-1-99 14:00 ~ 15:00
- Meeting with Dr. Krauthammer.
  10^{-5} time step
  30 ms, run up to
  summary of everything.
  Peak of σ, u, v, w. ⇒ Table, comments.

May-1-99

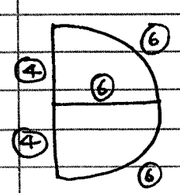
May-3-99 15:00 ~ 17:30
- Runtime and disk space test.

May-3-99

20:00 ~ 23:20
- Finish runtime and disk space test and summarize it.

May-3-99

May-4-99 22:00 ~ 24:00
- Test a model up to 0.01 and begin a stone.



May-4-99.

May-5-99 14:30 ~ 17:30
- Finish a stone and test the impact.
- Get 1 Gb disk from Vijay Agarwala.

May-5-99

22:00 ~ 24:00
- Change diaphragm and test the impact up to 0.015

May-5-99

May-6-99 15:00 ~ 17:30
- Investigate parameters to reduce the result file

May-6-99

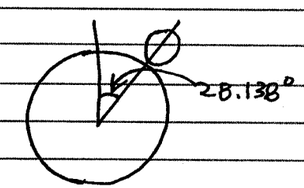
May-8-99 14:00 ~ 18:00
- Build a test model to get specific curves.

May-8-99

May-12-99 14:00 ~ 18:00
- change the contact between diaphragm and assembly

May-12-99

20:00 ~ 24:00
- Rotate the stone and make a new contact definition.



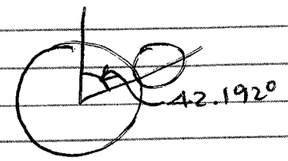
May-12-99

May-13-99 15:00 ~ 18:00
- Finish the second model and begin another model of rotated stone.

May-13-99

21:00 ~ 01:00

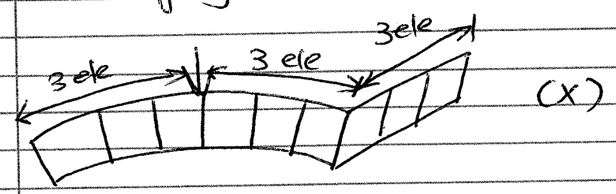
- Finish the third model



May-13-99

May-14-99 14:00 ~ 15:00

- Meeting with Dr. Krauthammer.
  - Summary of detail

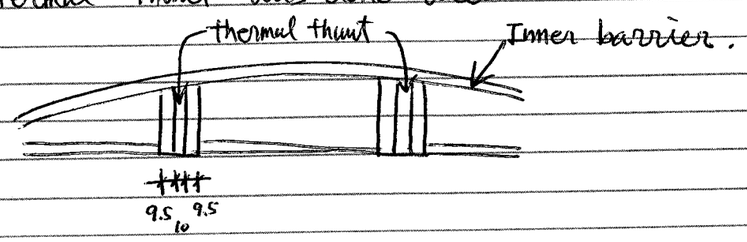


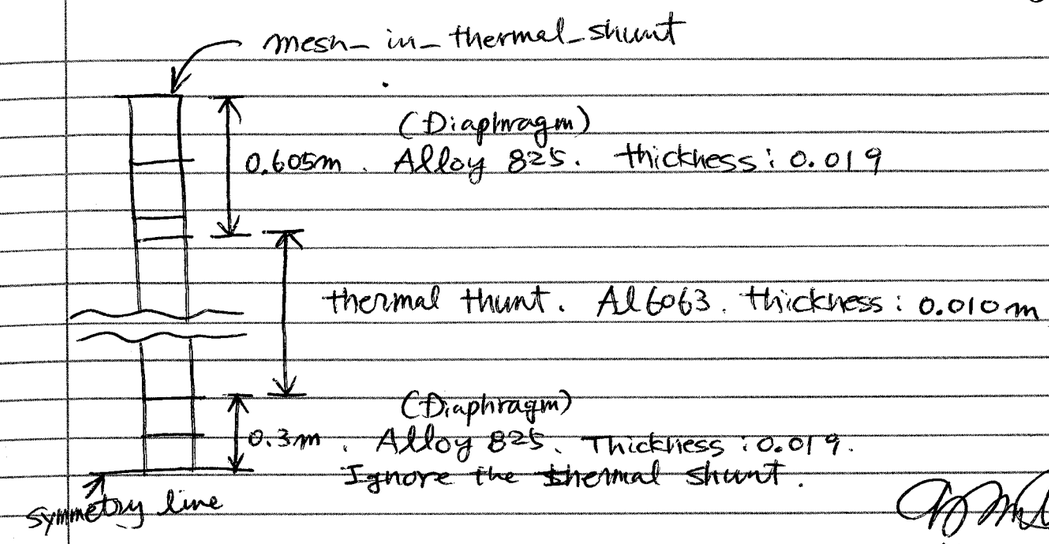
- Check support around 4 elements for each support.
  - $u, \dot{u}, \ddot{u}$ at nodes.
  - E, S at each element.
- How many element, node, properties, components
- Check worst.

May-14-99

May-15-99. 15:00 ~ 18:00

- Back to the first case, begin to input parameters
- The thickness of diaphragm = 0.0095
- Thermal thunt was done well.





May-15-99

21:00 ~ 01:45

- Input parameters, fix the input file Test run.
- Inner pressure of Helium gas : 28psi
  = 28 × 6894.76 pa
  = 1.93053×10^5 Pa
- Element set name.

C3D4 : STONEPRO	S3RS : DIAPHRAGM
C3D8R : INNERBAR	S4RS : DIAPHRAG
C3D6 : INNER BAR	S4RS : SPACERPR
C3D8R : OUTER BAR	S4RS : TUBEASSE
C3D6 : OUTER BAR	S4RS : THERMALS
	S4RS : DIAPHR-I

- Initial velocity of stone : 10.85 m/s
  6m Drop Height : $v_0 = \sqrt{2gh} = \sqrt{2 \times 9.8 \times 6}$
- Elset Added.

MESH_EDG	MESH_I_2	MESH_TRA
MESH_I_4	MESH_OUT	
MESH_INN	MESH_I_1	
MESH_O_1	MESH_IN	
MESH_I_3	MESH_STO	

up to 150 ms
start : 219.0, 210896

May-15-99

May-16-99 15:00 ~ 17:00
- Work on deformation speed errors.

May-16-99

20:00 ~ 23:40
- Change the material property and test model for errors
- up to 5 ms (1 hour expected)

May-16-99

May-17-99 14:00 ~ 15:00
- Meeting with Dr. Krauthammer.

May-17-99

May-18-99 14:40 ~ 17:50
- Test a model with solid diaphragm
- Change material property
  Outer Tube Assembly.
  $\sigma_y = 3061.6\,MPa$
  $E = 3471\,GPa$
  $t = 0.001\,m$
  Inner Tube Assembly
  $\sigma_y = 184.7\,MPa$
  $E = 209.41\,GPa$
  $t = 0.0178\,m$

May-18-99

May-19-99 14:00 ~ 15:00
- Meeting with Dr. Krauthammer.
- Increase E and thickness of diaphragm. It might solve deformation speed problem.

$$c = \sqrt{\frac{E}{\rho}}, \quad v = \frac{\sigma}{\rho c}$$

May-19-99

May-20-99 15:00 ~ 15:45
- Meeting with Dr. Krauthammer.

May-20-99

16:00 ~ 18:20
- Change the contact condition and test

May-20-99

22:00 ~ 24:20
- Release the equivalence between inner barrier and tube assembly. Test it

May-20-99

May-23-99 15:00 ~ 17:00
- Begin to write the first report

May-23-99

21:30 ~ 01:30
- Change the location of the stone.
- Rotate the stone with 28.138°

May-23-99

May-24-99 13:30 ~ 14:20
- Meeting with Dr. Krauthammer.

May-24-99

15:00 ~ 17:00
- Check the numbers of node and element of 30° rotated model.

May-24-99

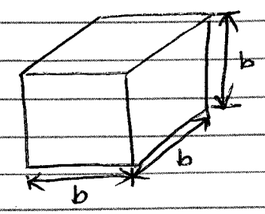
20:30 ~ 23:20
- Build 42° rotated model.

May-24-99

May-25-99 21:00 ~ 24:00
- Start to build the cubic shape rock impact model.
- Size of the cubic rock.

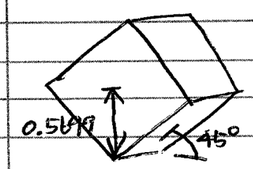
$$\frac{4}{3}\pi r^3 = b^2$$

$$r = 0.5m \Rightarrow b \doteqdot 0.806m$$



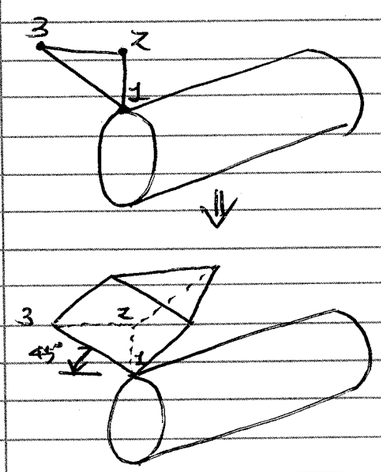
May-25-99

May-26-99 15:30 ~ 19:30
- Rotate the cubic stone to change the impact condition



May-26-99

22:00 ~ 24:20
- Rotate the cubic stone to be point impact.



1: (0, 2.29, 0.8145)
2: (-0.5699, 2.8599, 0.8145)
3: (-1.1399, 2.29, 0.8145)

May-28-99 21:00 ~ 24:00
- Post processing for the report.

May-28-99

Jun-1-99
22:00 ~ 24:00
- Collect the data from ~~resut~~ result file (Jul-1-99)

Jun-1-99

Jun-2-99 16:00 ~ 18:00
- Collect the data from result.

Jun-2-99

22:00 ~ 24:00
- Collect the data from result.

Jun-2-99

Jul-4-99 20:00 ~ 24:30
- check result, change running time and rerun

Jun-4-99

Jun-6-99 20:00 ~ 23:30
- Check result and disk capacity. Keep running.

Jun-6-99

Jun-8-99 14:00 ~ 18:00
- Check result, fix input file and rerun.

Jun-8-99

Jun-10-99 15:00 ~ 18:00
- Write the report

Jun-10-99

Jun-11-99 10:30 ~ 12:00
- Meeting with Dr. Krauthammer. & write the report.

Jun-11-99

14:00 ~ 18:40
- Write the report.

Jun-11-99

Jun-12-99 15:00 ~ 18:00
- Check result, collect data

Jun-12-99

22:00 ~ 24:00
- Fix the input file, rerun.

Jun-12-99

Jun-19-99 14:00~18:00

- Full model build from the stored model.
  : Fix group : mesh-in-assembly, mesh-in-diaphragm, mesh-in-thermal shunt.
- Equivalence.
  diaphragm - thermal shunt - inner barrier.
  → Structural stiffners are all attached to the inner barrier.

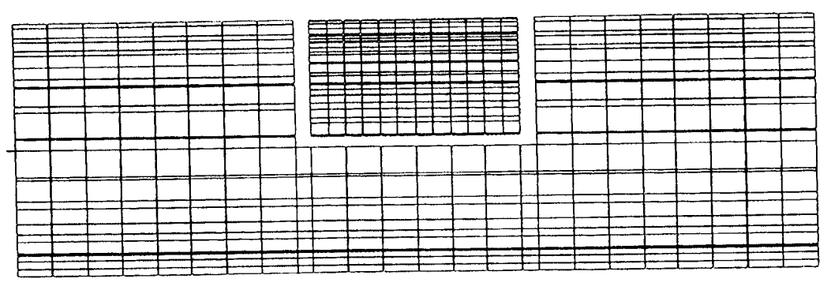
  Tube assembly - spacers.
  → Tube assembly part will interact with other parts by "contact"
- Adjust the location of spacers to fit tube assembly

Jun-19-99

Jun-20-99 14:00~16:45

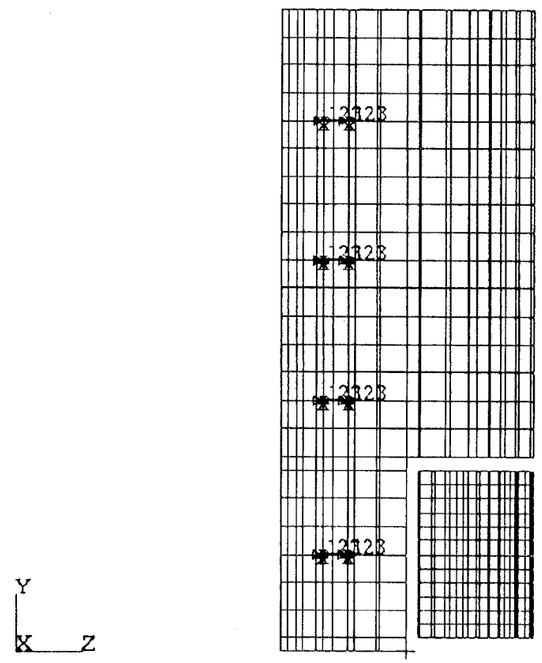
- Change the outer barrier for edge impact model

Outer Barrier.



Jun-20-99

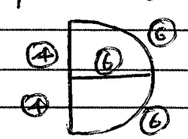
* Boundary condition on the full model



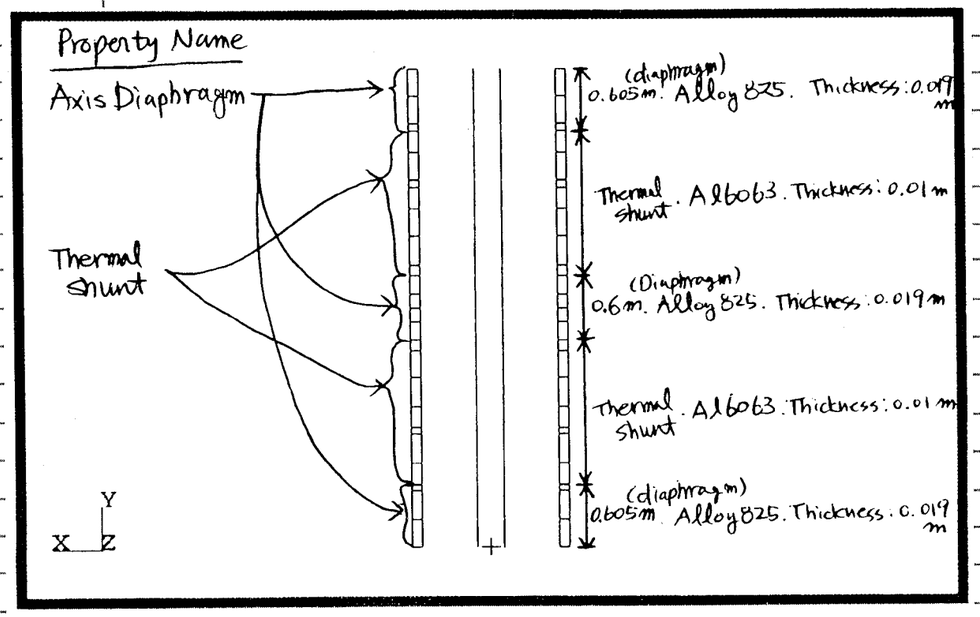
Jun-20-99

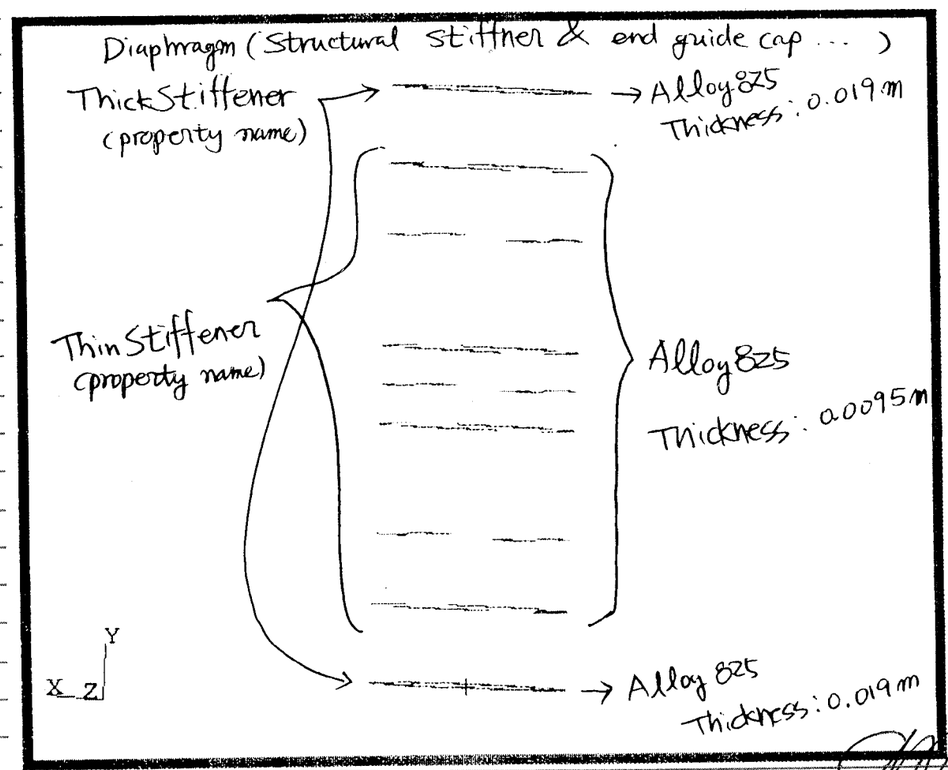
20:30~24:00

- Make a stone to the edge, input properties and test.
  : Impact location (0, -0.39, 0.8145)



sweep: 10





Jun-20-99

Element set names (of full model)

OUTERBAR
THERMALS
AXISDIAP
THINSTIF
THICKSTI
ROCK
SPACER
TUBEASSE
INNERBAR

Jun-20-99

Jul-21-99 14:00 ~ 18:00

- Make fine mesh at the edge side.
  - Change the element arrange of outer barrier
  - Rebuild { transient-outer-barrier
    edge-outer-barrier
    outer barrier lid
    inner " "
  - Equivalence

Jun-21-99

Jun-22-99 15:00 ~ 18:00

- Check waste form characteristics report revision 1

Jun-22-99

21:00 ~ 22:00

- Discussion of material property with Mr. Oh.

Things to be changed.

A. Fuel Assembly spacers : 16 → 8
   - Inner Part assemblies tubes

B. Fuel Assembly types : 15x15 → 17x17
   - Equivalent assembly tube thickness for bending
   - Equivalent density of fuel assembly spacers
   - Equivalent material properties (estimation)

C. Material Properties
   Zircalloy & Inconel Alloy 718

Jun-22-99

22:00 ~ 23:30

- Remake the contact definition between outer barriers and inner barriers and outer barrier and stone

Jun-22-99

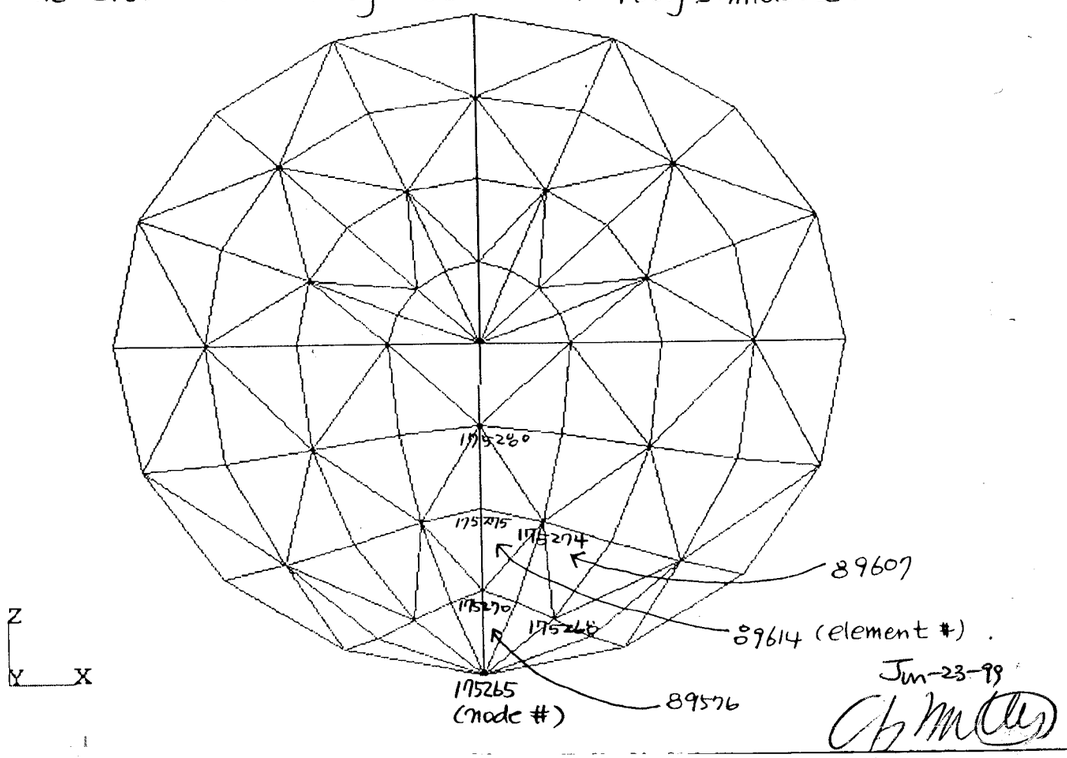
Jun-23-99 13:00 ~ 13:30

- Meeting with Dr. Krauthammer.
  1. Take the most severe case. Make the change, Run.
  2. Compare it with the previous one. How much differences?
  3. Does the rock break? crush? crack?
  4. Material property to rock. (elastic? elasto-plastic?)
  5. If rock stays in elastic domain, no need to go.
  6. Check poisson's effects.

Jun-23-99

14:00 ~ 17:35.

- Check ~~mo~~ Jun-23-99 node and element of the stone. Test run.
- The cross section of stone on half_middle_0° model



Jun-23-99

21:00 ~ 24:00

- Begin to calculate equivalent properties.
  - Area of guide tube (m^2)
    24 X each area = 1.8525×10^{-4}
  - Area of Instrument (m^2)
    1 X each area = 7.718×10^{-6} (same as guide tube)
  - Area of Fuel rods (m^2)
    264 X each area = 2.18559×10^{-3}
  - Area of Fuel Pellets (m^2)
    264 X each area = 0.0139147 (m^2)
  - Area of spacers (m^2)
    $0.214^2 - 1.8525 \times 10^{-4} - 7.718 \times 10^{-6} - 2.18559 \times 10^{-3} - 0.0139147$
    $= 0.02950227\ m^2$

Jun-23-99

Jun-24-99 14:30 ~ 18:00

- Calculate the equivalent density.
  - Density of Zircalloy : 7140 kg/m^3
  - Density of fuel pellets : After sintering, the density of pellet increases to ~94% of original
    - Before sintering : 10.97 g/cm^3
    - After " : 21.282 g/cm^3 ⇒ 21282 kg/m^3
  - Total Mass.
    - Mass of guide tube
      $1.8525 \times 10^{-4} \times 4.858 \times 7140 = 6.065$ kg
    - Mass of Instrument
      $7.718 \times 10^{-6} \times 4.585 \times 7140 = 0.253$ kg
    - Mass of Fuel rods
      $2.18559 \times 10^{-3} \times 4.585 \times 7140 = 71.549$ kg
    - Mass of Fuel pellets.
      $0.0139147 \times 4.585 \times 21282 = 1357.768$ kg
    - Mass of spacers
      $0.02950227 \times 0.025 \times 8 \times 8190 = 48.325$ kg
      Assumed thickness # of spacers.

Total mass

$6.065 + 0.253 + 71.549 + 1357.768 + 48.325 = 1483.960$ kg

Average density for spacers.

$$\frac{1483.960}{0.0295027 \times 0.025 \times 8} = 251495.6 \text{ kg/m}^3$$

Jun-24-99

21:00 ~ 24:00

- Calculate equivalent bending resistance of tube assembly.
  - Original tube assembly thickness = 0.015m
  - Cladding area of the cross section = 2.18559×10^{-3} m^2
  - Equivalent cladding area = $B \times H = 0.010213 \times 0.214$
  - $I = \frac{BH^3}{12} = 8.340909 \times 10^{-6}$ m^4
  - equivalent cladding box thickness.

$$I = \frac{bh^3}{12} - \frac{b_1 h_1^3}{12} = 8.340909 \times 10^{-6} \text{ m}^4$$

$b, h = 0.214$m, $b_1 = h_1$

$b_1 = h_1 = 0.2114$m

$t = 0.214 - 0.2114 = 0.0026$m : thickness

- The equivalent tube assembly thickness with regard to the beam action of the fuel assembly.

$= 0.015 + 0.0026 = 0.0176$m

- Equivalent yield stress estimation
  a. area of quarter cross section of tube assembly.
  $= 0.00377$ m^2
  $P_y = 6.8614 \times 10^5$ N
  b. area of quarter cross section of the fuel rod cladding $= 5.4640 \times 10^{-4}$ m^2
  $P_y = 2.34952 \times 10^5$ N (σ_y of Zircalloy in 385°C (725°F) is 430MPa)
  c. Equivalent yield stress (σ_y) = $P_{total} / A_{total} = 213.39$ MPa
- Equivalent estimation of the tube assembly
  a. 172 : 195 = 213.39 : E
  E = 241.92 GPa

Jun-24-99

Jun-25-99 15:00 ~ 18:00
- Rebuild tube assembly and spacers.

Jun-25-99

Jun-26-99 14:00~18:20
- Check material Property. make an input file. Work on element overlay error.

Jun-26-99

21:00 ~ 24:00
- Fix element error
- I think it's one of MSC/PATRAN's bugs
- Verify - element - duplicate

Jun-26-99

Jun-27-99 14:00 ~ 16:30
- Check the results of running model. Modify input file

21:30 ~ 23:50
- Check the results. Write the report

Jun-27-99

Jul-12-99 13:00~18:00
- Start to rebuild the inside of full-edge model.

Jul-12-99

Jul-14-99 15:00~18:00
- Build the edge-impact model

20:00 ~ 00:00
- Finish edge impact model. Debug and run.

Jul-14-99

Jul-15-99 15:00~18:00
- Check results and rerun
- Result values are too low. It should be checked.

Jul-15-99

Jul-16-99 14:00~19:00
- Fix contact problem. Rerun.

Jul-16-99

Jul-18-99 20:00 ~ 24:00
- Collect data from edge impact model.

Jul-18-99

Jul-19-99 11:00 ~ 12:00
- Meeting with Dr. Krauthammer
  - Heavier rock = increase weight, run it. increase diameter.

    Worst case : M, S, E

    Failure : Outer barrier, inside

    Failure condition.

    Summary

    Conclusion

    Recommendation.

Jul-19-99

20:00 ~ 24:00
- Finish collecting data from edge impact model.

Jul-19-99

Jul-20-99 12:00 ~ 15:00
- Build test model to compare with results.

Jul-20-99

21:00 ~ 24:00
- Begin to write the report.

Jul-20-99

Jul-21-99 10:00 ~ 12:00
- Run half-middle impact model with 2m diameter rock.

Jul-21-99

15:00 ~ 17:30
- Check the results up to 10 ms
- It looks critical.
- Around 4.4ms, there was a sudden expansion of stress & strain.

Jul-21-99

21:00 ~ 23:00
- Run full model with edge-impact with 2m diameter rock.

Jul-21-99

Jul-22-99 14:00 ~ 18:00
- Check the result from edge impact model

Jul-22-99

20:00 ~ 24:00
- Run 1.5m rock case & write the report

Jul-22-99

Jul-23-99 09:00 ~ 09:30
- Meeting with Dr. Krauthammer

Density

Diameter	Density (kg/m³)
1m	2550
1.5m	8606.25
1.75m	13666.41
2m	20400
3m	68850
3.5m	10933.13
5m	318750.0
7m	874650.0

Jul-23-99

10:00 ~ 12:00
- Check 1.5m rock case.

Jul-23-99

14:00 ~ 17:00
- Check 1.5m rock case. Run 1.75m rock case.

Jul-23-99

21:00 ~ 24:00
- Check 1.75m rock case.

Jul-23-99

Jul-24-99 14:00~18:00
- Write the report. Check results
- Ch2. Code Validation.

Jul-24-99

Jul-25-99 20:00~01:00
- Check the results & write report

Jul-25-99

Jul-26-99 11:00~12:00
- Stress-strains in failure model were abnormal.
- Lower strain gives a higher stress. Why?
- Strain of steel fails around 12%
- Check failure strain.
- Report the slowness of the Abaqus.

Jul-26-99

13:00~17:00
- Check results & write report
- Code Validation

Jul-26-99

21:00~03:00
- Check results & write report
- Failure model

Jul-26-99

Jul-27-99 13:00~18:00
- Check results & write report
- Failure model.

Jul-27-99

21:30-03:00
- Check results & write report
- Failure model.

Jul-27-99

Jul-28-99 16:00~17:00
- Meeting with Dr. Krauthammer.

Jul-28-99

21:00~03:00
- Check failure model & write report.

Jul-28-99

Jul-29-99 10:00~10:30
- Meeting with Dr. Krauthammer.

Jul-29-99

11:00~16:00
- Write the report.
- Write the ~~report~~ result part

Jul-29-99

Jul-29-99

Jul-30-99 09:30~10:00
- Meeting with Dr. Krauthammer.

Jul-30-99

12:00~15:00
- Write the result part

Jul-30-99

15:00~17:00
- Check the final report with Dr. Krauthammer.

Jul-30-99

23:00~00:00
- Print the report and final check.

Jul-30-99

Aug-4-99 10:30~11:30
- Meeting with Dr. Krauthammer.
  - Reduce the line spacing from double to single.
  - Convert the report from MS-Word to Corel Word Perfect.
  - Change the color of ABAQUS pictures.

Aug-4-99

Aug-5-99 14:00~18:00
- Convert MS-Word file to WordPerfect.

Aug-5-99

Aug-6-99 11:30~12:00
- Meeting with Dr. Krauthammer.
- Failure criteria
  - Define locations of peak values.
  - Cite reports
  - Expand discussion for each case-result.
  - Sec 2.4, P-3. Fig 14

- Rock size conversion by energy.

$$\frac{1}{2} m_1 v_1^2 = \frac{1}{2} m_2 v_2^2$$

- What happens to the rock?
- Conversion to WPB

Aug-6-99

13:00~17:00

- Run other model. Convert report

Aug-6-99

21:00~03:00

- Convert report

Aug-6-99

Aug-7-99 20:00~00:30

- Finish converting report.

Aug-7-99

Aug-8-99 13:00~17:00

- Write the report.

Aug-8-99

20:00~03:00

- Write the report.

Aug-8-99

Aug-9-99 10:00~12:00, 13:00~15:00

- Write the report with MS-WORD

Aug-9-99

15:00~16:00

- Meeting with Dr. Krauthammer.

Aug-9-99

16:00~18:00, 20:00~02:00

- Check the failure model results and write the report

Aug-9-99

Aug-10-99 10:00~11:00

- Meeting with Dr. Krauthammer.

Aug-10-99

13:00~18:00, 20:00~04:30

- Convert the report to the WPB

Aug-10-99

Aug-13-99 14:00~17:30

- Take care of questions

Aug-13-99

Aug-23-99 10:00~11:00

- Meeting
- Integrate contact pressure (not with node option)
- Make a copy of contact option (small & tied)

Aug-23-99

11:00~14:00

- Rearrange and send files.

Aug-23-99

Aug-26-99 13:00~14:00

- Build small model to see the long term response.

Aug-26-99

Aug-27-99 11:00~12:00

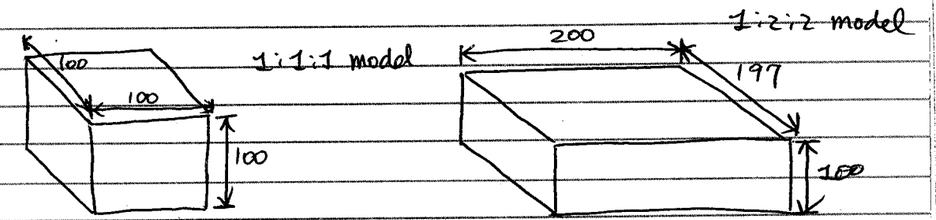
- Meeting with Dr. Krauthammer.

Aug-27-99

13:00~18:00

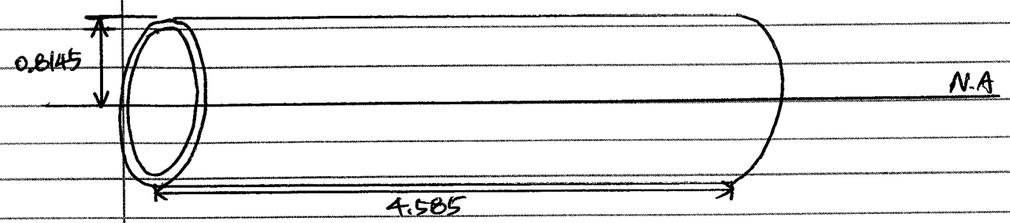
- Start to build the aspect ratio model.
- The aspect ratio which have been used
  - The outer barrier

*Dense Mesh (mm) * Coarse Mesh (mm)

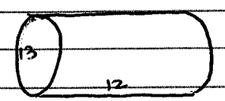


Aug-27-99

- Test model (m)

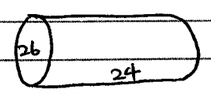


① 1:4:4 (mm)



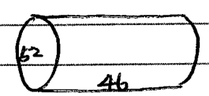
# of element : stone : 240
container : 156
396

② 1:2:2 (mm)



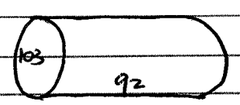
# of element : stone : 240
container : 624
864

③ 1:1:1 (mm)



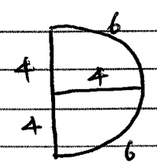
# of element : stone : 240
container : 2392
2632

④ 1:0.5:0.5 (mm)



# of element : stone : 240
container : 9476
9716

⑤ Stone.



Sweep : 8

- Node number. & Element number.

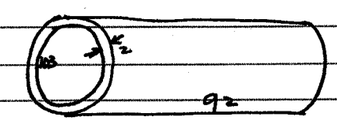
	top of container	stone (impact)	top and 1/4
1:4:4	20 (18)	392 (382)	~~705 (~~ L-Aug-27-99
1:2:2	node element 699 (495)	"	705 (480)
1:1:1	732 (507)	"	744 (520)
1:0.5:0.5	801 (576)	"	824 (599)

Aug-27-99

Aug-29-99 13:00 ~ 18:00
- Keep building the aspect ratio model.
  21:00 ~ 24:00
- Build aspect ratio model.

Aug-29-99

Aug-30-99 13:00 ~ 18:00
- Build a refined mesh.
- 0.5:0.5:0.5 (mm



Stone : 240
container : 18952
19192

node # / Element #

	top of the container	top and quarter
0.5:0.5:0.5	894 / 760	917 / 783

Aug-30-99

22:00 ~ 24:00
- Build a refined model.

Aug-30-99

15:00 ~ 16:00
- Meeting with Dr. Krauthammer.

Aug-30-99

Aug-31-99 17:30 ~ 18:30

- Meeting with Dr. Krauthammer.
  21:00 ~ 24:00
- Area of guide tube (24 were used)

$$A = \frac{1.219^2\pi}{4} - \frac{(1.219-0.041\times2)^2\pi}{4} = 1.5173\times10^{-1}\ cm^2$$
$$= 1.5173\times10^{-5}\ m^2$$

- Area of the instrument tube (1 was used)
  same as guide tube.
- Area of fuel rod

$$A = \frac{0.950^2\pi}{4} - \frac{0.8192^2\pi}{4} = 0.18175\ cm^2$$
$$= 1.8175\times10^{-5}\ m^2$$

- Area of the fuel pellet (264 were used)

$$A = \frac{0.8192^2\pi}{4} = 5.2707\times10^{-5}\ m^2$$

- Area of spacers

$$A = 0.214^2 - 24\times1.5173\times10^{-5} - 1.5173\times10^{-5} - 264$$
$$(1.8175\times10^{-5} + 5.2707\times10^{-5}) = 0.02670\ m^2$$

- Calculate the equivalent density.
  Density of Zircalloy: 7140 kg/m³
  " fuel pellets
  After sintering, the density of pellets increases to ~94% of the original:
  Before sintering
  10.97 g/cm³
  After sintering
  21.282 g/cm³

  Density of Inconel alloy 718: 8190 kg/m³

* Total mass *
- Mass of guide tube
  $24\times1.5173\times10^{-5}\times7140\times4.585 = 11.921$ kg

- Mass of instrument tube
  $1.5173\times10^{-5}\times7140\times4.585 = 0.497$ kg
- Mass of the fuel rods
  $264\times1.8175\times10^{-5}\times7140\times4.585 = 157.078$ kg.
- Mass of fuel pellets
  $264\times5.2707\times10^{-5}\times21282\times4.585 = 1357.763$ kg
- Mass of spacers
  $8\times0.025\times0.02670\times8190 = 43.735$ kg.
  ↑ # of spacers ↑ the assumed thickness

* Total mass *
$11.921+0.497+157.078+1357.763+43.735 = 1570.994$ kg

Average density for spacers.

$$\frac{1570.994}{0.02670\times8\times0.025} = 294193.6\ kg/m^3$$

Aug-31-99

Sep-1-99 10:00 ~ 12:00
- Meeting with Dr. Krauthammer.

Sep-1-99

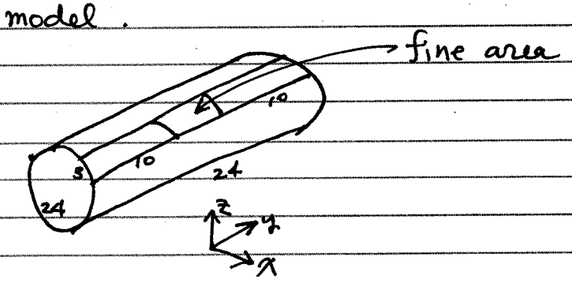
Sep-3-99 14:00 ~ 17:00
- Build and analyse the tied model.

Sep-3-99

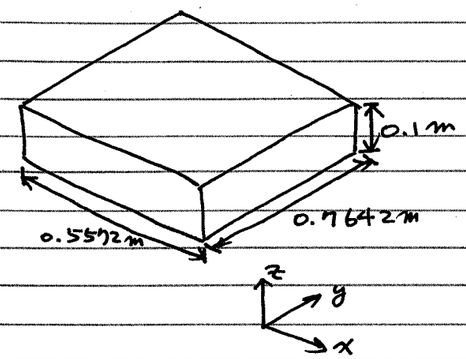
Sep-5-99 13:00 ~ 17:00, 21:00 ~ 23:00
- Finish the tied contact model issue.

Sep-5-99

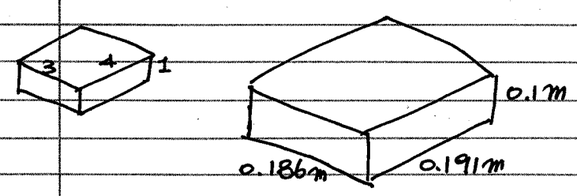
Sep-6-99 14:00 ~ 18:00, 22:00 ~ 01:00
- Start to build finer mesh to analyse aspect ratio model.



- fine mesh area -

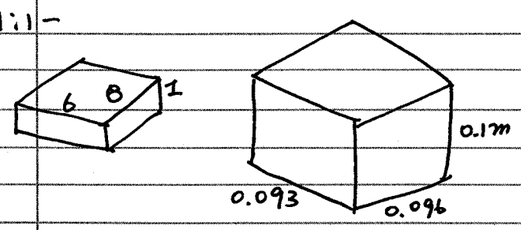


- 1:2:2 -



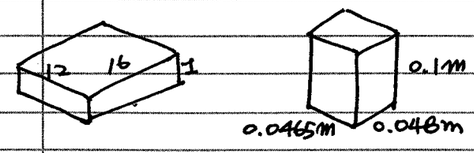
Impact location	# of element
node #2939	container: 648
ele #1729	stone: 240

- 1:1:1 -



Impact location	# of element
node #3008	container 684
ele #1742	stone 240

- 1:0.5:0.5 -



Impact location	# of element
node: 3170	1068
ele: 1811	

Sep-6-99

Sep-7-99 21:00~24:00

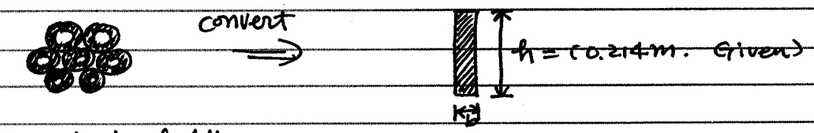
- Recalculate the equivalent material properties.

* Equivalent bending resistance of the tube assembly.

- Original tube assembly thickness = 0.005m
- Cladding areas of cross section = # of cladding × Each area
  $= 264 \times 1.8175 \times 10^{-5} m^2$
  $= 4.798 \times 10^{-3} m^2$
- (from sponsor) use 13×10^6 psi ($= 6894.76 \times 13 \times 10^6$ Pa $= 89.63$ GPa)

$$n = \frac{E \text{ of Zircalloy}}{E \text{ of } 316L} = \frac{89.63}{195} = 0.46$$

- Equivalent cladding area = n × cladding area
  $= 0.46 \times 4.798 \times 10^{-3}$
  $= 2.205 \times 10^{-3} m^2$
-



Equivalent cladding area
$= 2.205 \times 10^{-3} m^2$

- Get thickness b

$0.214b = 2.205 \times 10^{-3}$
$b = 0.0103 m$

$$\Rightarrow I = \frac{bh^3}{12} = \frac{0.0103 \times 0.214^3}{12}$$
$$= 8.412 \times 10^{-6} m^4$$

- Get the equivalent thickness of box which was converted from cladding

$$I = \frac{bh^3}{12} - \frac{b_1 h_1^3}{12} = 8.412 \times 10^{-6} m^4$$

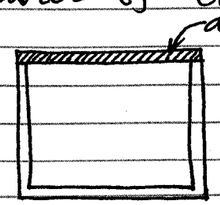
where $b, h = 0.214$
$b_1 = h_1$

$b_1 = h_1 = 0.211 m \Rightarrow$ thickness $= b - b_1 = 0.003 m$

- The equivalent tube assembly thickness with regard to the beam action of the fuel assembly.
  $= 0.005 + 0.003 = 0.008 m$

* Equivalent yield stress estimation.

- Quarter of cross sectional area of tube assembly.



$0.008 \times 0.214 = 0.00171\ m^2$

$P_y = \sigma_y \times A = 1.72 \times 10^8 \times 0.00171 = 2.941 \times 10^5\ N$

- Quarter of cross sectional area of fuel rod cladding.
$= \frac{1}{4} \times$ All cladding area $= \frac{1}{4} \times 264 \times 1.8175 \times 10^{-5}$
$= 0.0012\ m^2$

$P_y = \sigma_y \times A = 430 \times 10^6 \times 0.0012 = 5.16 \times 10^5\ N$

(σ_y of Zircalloy in 385°c (725°F) is 430MPa)

(BAUER, A.A., and Lowry, L.M., Tensile Properties and annealing characteristics of H.B. Robinson Spent Fuel Cladding, ~~Battlelle Memorial~~ Battelle
L - ~~Avg~~ Sep-7-99
L - Sep-7-99
Memorial Institute Columbus Laboratories, 505 King Ave., Columbus, Ohio 43201, 1978)

- Equivalent yield stress

$$\sigma_y = \frac{1/4\ P_{total}}{1/4\ A_{total}} = \frac{(2.941 + 5.16) \times 10^5}{0.00171 + 0.0012} = 278.38\ MPa$$

- Equivalent estimation of the tube assembly
172 : 195 = 278.38 : Equiv.
Equiv. = 315.61 GPa

Sep-7-99

Sep-8-99. 12:00~13:00
- Meeting with Dr. Krauthammer

Sep-8-99

Sep-9-99 14:00~15:00
- Meeting with Dr. Krauthammer
- Dyna 3D, LS-Dyna, Abaqus.
elastic, elasto-plastic, elastic-plastic-linear E.
, full non-linear.

Sep-9-99

Sep-13-99 13:00~18:00, 21:00~23:00
- Build models to compare with experimental data.

Sep-13-99

Sep-23-99 14:00~16:00
- Work on the Abaqus code validation

Sep-23-99

21:00~24:00
- Work on Abaqus code validation
- 1:1:1 elem #40 (beam B)
Impact location
node : 32, ele : 50
- ~~elasto~~ - plastic
*plastic
43000., 0.
- ~~elasto~~ - kinematic plastic
*plastic
43000., 0.
64880., 0.2
- elastic - perfectly plastic.
43140., 0.
43200., 0.022
46573., 0.025
53850., 0.05
58216., 0.075
60399., 0.1
62582., 0.125
64038., 0.15
64765., 0.175
64880., 0.2

3. 1:1:1 elem # 320 (beam C)
Impact location : node 245, ele : 304
plastic definitions are same as the previous case.

Sep-23-99

Sep-24-99 15:00~18:00
- The Abaqus Code Validation

Sep-24-99

21:00~01:00
- The Abaqus Code Validation
1:1:1 element # 2560 (beam D)
Impact location : node 507, element : 1645

Sep-24-99

Sep-25-99 13:00~18:00
- Build simulation result CD's.

Sep-25-99

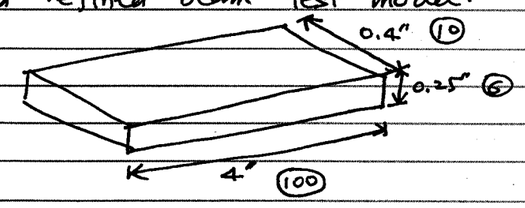
Sep-26-99 15:00~16:00
- Meeting with Dr. Krauthammer.
- Mesh identical, material properties have to be exact same.

Sep-26-99

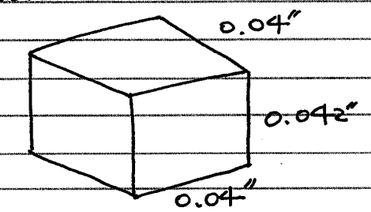
18:00~20:00
- Build backup CD

Sep-26-99

Sep-27-99 14:00~18:00
- Build refined beam test model.



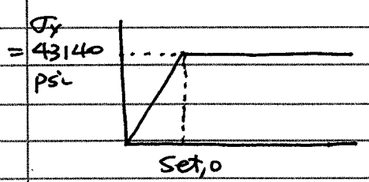
Aspect ratio.



≃ 1:1:1

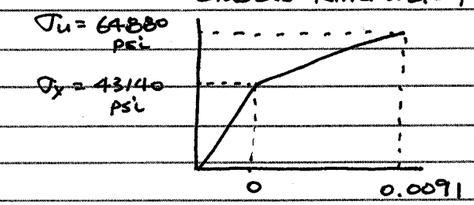
Given Young's modulus : 2.4×10^6

- Elasto-plastic



*plastic
43140., 0.

- Elastic-kinematic Plastic



*plastic
43140., 0.
64880., 0.0091

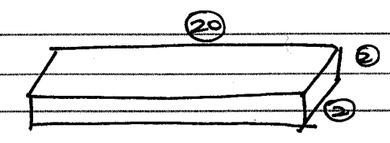
Impact location
node: 698 element: 1635

Sep-27-99

Sep-28-99 14:00 ~ 16:00
- Finish refined mesh model.

Sep-28-99

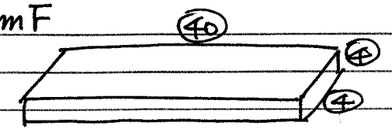
Sep-29-99 14:00~18:00
- Build 1:2:2 element #80 model
- Beam E



Impact node: 174, Impact element : 154

Sep-29-99

Sep-30-99 14:00 ~ 18:00
- Build 1:2:2 element # 640 model
- Beam F



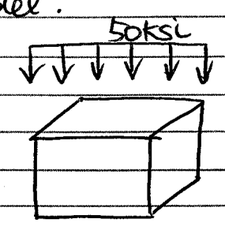
Impact node : 245, Impact element : 305

Sep-30-99

Oct-4-99 14:00 ~ 14:30
- Meeting with Dr. Krauthammer.
- Set up elastic one element simulation to test element material properties between three codes. Make sure every element has same gauss integration points.
- What is C3D8R and C3D8I?

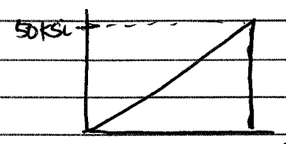
Oct-4-99

Oct-6-99
- Check the property of element and build one element model.
-



$E = 8.8 \times 10^6$ psi
$\nu = 0.3$
$\rho = 0.1$
support : fix at the bottom
$\sigma_y = 43$ KSI

Load.

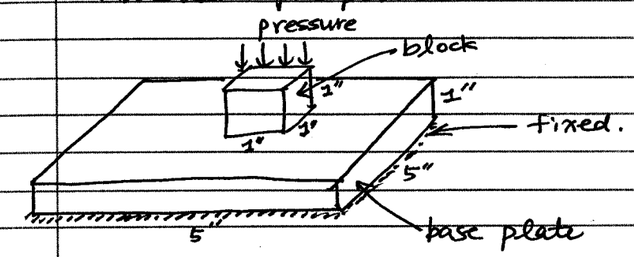


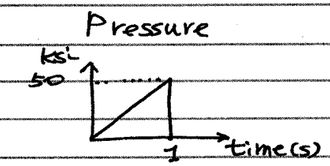
Oct-6-99

Oct-7-99 14:00 ~ 14:30
- Meeting with Dr. Krauthammer.
- Reformulate the previous one element model.
  Change B.C.
  Try elasto-kinematic plastic.
  Use real steel property.

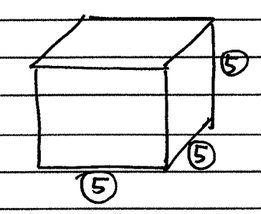
Oct-7-99

Oct-10-99 20:00 ~ 01:00
- Build one and 125 elements model.
- Material properties





- Block
  Material steel
  $\sigma_y = 43$ ksi, $E = 2.9 \times 10^6$ psi, $\nu = 0.3$, $\rho = 0.328$ lb/in^3
  Base plate
  $\sigma_y = 100$ ksi, $E = 90 \times 10^6$ psi, $\nu = 0.5$ (0.49999 was used)
- Check
  Node : 24, Element : 3
- 125 elements brick case.



Properties are all same.

- Check
  Node : 29122, element : 141

※ It takes too much time and disk space. As the size of the element reduces due to the pressure, the time step is getting shorter and shorter. So, we should change

the material property of the base plate to "Rigid"

Oct-10-99

Oct-11-99 14:00~18:00

- Build one element and 125 elements model on rigid base.
- All properties are same except the base plate is rigid. checked node 6, checked element 1. for one element case.

Oct-11-99

Oct-15-99 14:00~14:30

- Meeting with Dr. Krauthammer.

Oct-15-99

Oct-24-99 14:00~15:00

- Meeting with Dr. Krauthammer.
- Read paper of Lui & Jones about the Abaqus Modeling
- Tabulate & compare.
  make time step, show each value at every time step.
- Does it have any meaning after yielding?
- Check models in Jin's work.
- Check all data given by Dr. Jones.
- Simulate with wood (∵ soft material) $E = 2\times10^6$ psi roughly.

Oct-24-99

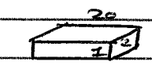
Oct-27-99 14:00~18:00

- Check the model properties and compare with LS-DYNA models.

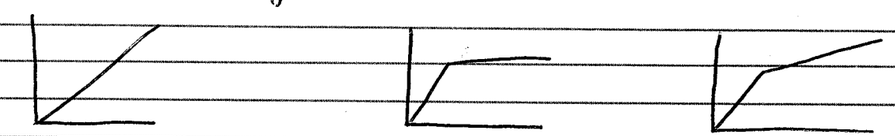
- Beam Model.
  1. Steel beam.
     $\gamma_0 = 0.283$ lb/in^3 (weight density), $\nu = 0.3$
     $\sigma_y = 43.14\times10^3$ psi
     $\sigma_u = 64.88E3$ psi
     $E = 30E6$ psi

     $E_t = 300{,}000$ psi
  2. Striker
     $V_{initial} = 451.6$ in/s, $\nu = 0.3$
     $\gamma_0 = 687.5$ lb/in^3
     $E = 30E6$ psi
     $\sigma_y = 20E4$ psi
     $E_t = 30E4$ psi
  3. # of elements.

Beam	bi	element #	40	(1:1:1)	20:2:1
	ci	"	320	(")	40:4:2
	di	"	2560	(")	80:8:4
	gi	"	80	(1:2:2)	20:2:2
	hi	"	640	(")	40:4:4



  4. Tested Modelings



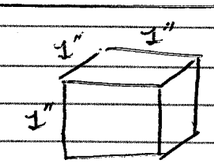
- Elastic - - Elasto-plastic - - Elasto-kinematic plastic -

Oct-27-99

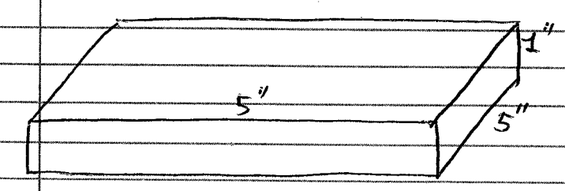
20:00~01:00

- Build the code validation model.

One Element Model.



: 1X1X1 mesh or 5x5x5 mesh



: 5x5x1 mesh.

- Material Properties
  a. Elastic (Node #6, Ele # 1 for 1x1x1 model)
     - Brick (Node #282, Ele # 139 for 5x5x5 ")
       $\gamma = 0.328\ lb/in^3$, $E = 2.9E6$ psi, $\nu = 0.3$
  b. Elasto-plastic
     - Brick
       $\gamma = 0.328\ lb/in^3$, $E = 2.9E6$ psi, $\nu = 0.3$
       $\sigma_y = 43000$ psi
  c. Elasto-Kinematic Plastic
     - Brick
       $\gamma = 0.328\ lb/in^3$, $E = 2.9E6$ psi, $\nu = 0.3$
       $\sigma_y = 43000$ psi, $E_t = 2.4E5$ psi
  d. Base Plate for all cases.
     $\gamma = 0.328\ lb/in^3$, $E = 9E10$ psi, $\nu = 0.3$
     $\sigma_y = 4.3E10$ psi, $E_t = 2.4E5$ psi.

Oct-27-99

Oct-28-99 14:00~16:00
- Check the document of the one element model.

Oct-28-99

20:00~01:00
- Build the one element model.

Oct-28-99

Oct-29-99 14:00~18:00
- Build beam impact model.

Oct-29-99

20:00~24:00
- Build beam impact model

Oct-29-99

Oct-30-99 14:00~18:00
- Finish models, print, and check the results

Oct-30-99

20:00~24:00
- Set up data, begin to write

Oct-30-99

Oct-31-99 14:00~18:00
- Check the result, analyse the one element model.

Oct-31-99

20:00~02:00
- Write the report
- Pressed Block part.

Oct-31-99

Nov-1-99 13:00~14:00
- Write the report (pressed block part)

Nov-1-99

14:00~15:00
- Meeting with Dr. Krauthammer.
- Found the penetration of element.

Nov-1-99

15:30~18:00
- Fix the contact problem of the impact beam model

Nov-1-99

20:00~24:00
- Write the report about the pressed block part.

Nov-1-99

Nov-2-99 14:00~16:00
- Write the report about the pressed block part.

Nov-2-99

20:00~24:00
- Write the report about the pressed block part.

Nov-2-99

Nov-3-99 15:00~18:00
- Write the report about the impact beam.

Nov-3-99

20:30~01:30
- Write the report about the impact beam.

Nov-3-99

Nov-4-99 14:00~16:00
- Write the report about the impact beam.

Nov-4-99

20:00 ~ 02:00
- Write the report about the impact beam.

Nov-4-99

Nov-5-99 11:15~12:15
- Meeting with Dr. Krauthammer.

Nov-5-99

14:00~18:00
- Change the Young's modulus and yield stress, and rerun
- Nothing's changed.

Nov-5-99

Nov-7-99 14:00~18:00
- Check the material properties from documents

Nov-7-99

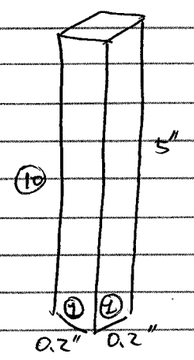
21:00~03:00
- Rebuild different type striker model.

Nov-7-99

Nov-8-99 14:00~14:30
- Meeting with Dr. Krauthammer.

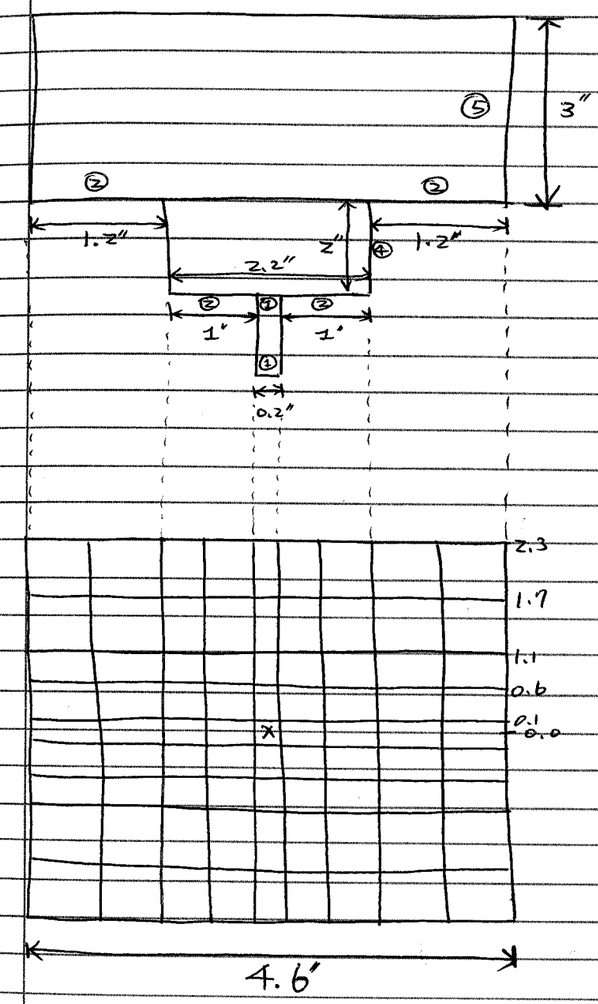
Nov-8-99

Nov-11-99 21:00~02:00
- Build various strikers model.
- <Case 2.>

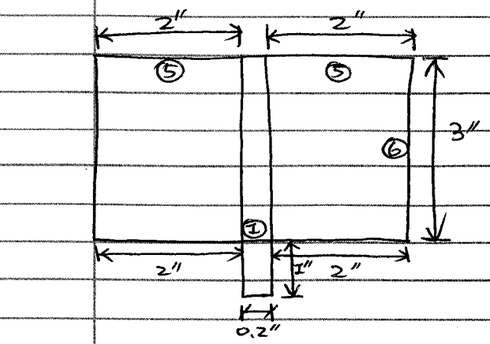


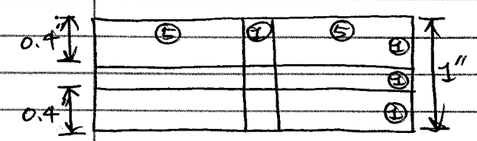
$\gamma = 55.188\ lb/in^3$

<Case 3>



<Case 4>





Nov-11-99

Nov-12-99 15:00 ~ 17:00
- Write the report about various strikers

Nov-12-99

Nov-14-99 14:00 ~ 18:00
- Write the report about the strikers

Nov-14-99

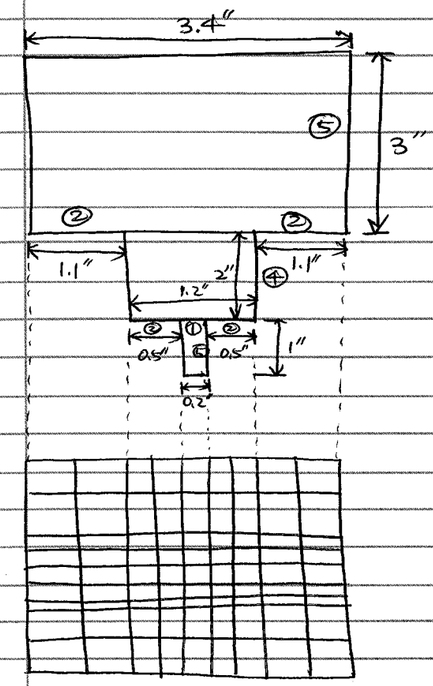
20:00 ~ 23:00
- Write the report about the strikers

Nov-14-99

Nov-16-99 21:00 ~ 01:00
- Modify the striker number 3 model and write the report.

Nov-16-99

Nov-16-99



$\gamma = 0.294\ lb/in^3$

Nov-16-99

Nov-17-99 01:00 ~ 03:00
- Write the report about strikers.

Nov-17-99

21:00 ~ 03:00
- Write the report about strikers.

Nov-17-99

Nov-18-99 14:00 ~ 14:30
- Meeting with Dr. Krauthammer.

Nov-18-99

Nov-21-99 14:00 ~ 18:00
- Change the report.

Nov-21-99

Nov-26-99 14:00 ~ 18:00
- Test elasto-plastic model of striker models.

Nov-26-99

Pages 138 Through 179 Are Intentionally
Left Blank

July – 28 – 98

Meeting #1

Tool : Patran & Abaqus (5.7) for modeling.

Criteria : Follow the regulation of energy center.
(Qulity insulance stuff)
* Regulation Part 72 *

Contents : 1) Temp – time relations (1 ~ 100 ~ several years)
2) Accidental impact event (Falling Rock)
3) Any Chemical Changes of the tube.
i.e – Physical reaction, Collosion, radiation, chemical materal change gas ... etc

Considerations : A. material properties ⇒ How to model.
1) $\sigma - \varepsilon$ curve (temp. function)
2) thermal conductivity
3) E, σ_y, ν, σ_u, % elongation of tangent modulus.
4) Density
B. Gravity
C. Impact
1) point
2) flat
3) corner (line)
} depends on what type of rock?
D. Convection of the air part.
E. B.C
F. Welding part (sliding definition)

Dimension : m, kg (mass). Newton (force).
pascal

Materials : ASTM & Matal's handbook
A 516 : Carbon steel (ASTM 01.04. 247 pg)
6063 AL : Aluminium
316B 6A : stainless steel
Alloy 825 : Nickel based alloy (ASTM B829-96)
: Nickel & Nickel alloy

* ASTM doesn't have enough information.
Try Metal's handbook

[signature]
7/28/98

July – 29 – 98

1. Complete the finding of material properties. (Done)
2. Modeling practice (patran) to compare with "ingrid + patran"
* We need to be familiar with "patran"

<Note> ingrid → patran

(brief procedure)

→ ls-ingrid $i =$ ○.

→ Patran (P3)
files import the file ○

→ Modification

3. Most of material Properties are found except $\sigma - \varepsilon$ curve & A516 steel (density & thermal conductivity)

4. A516 density
Temp.

[signature]
7/29/98

July - 30 - 98

1. What information we need for material properties.

   $\sigma - \varepsilon$ curve (explicit)
   - Normal temp. elevated temp.
   - What is max temp (250°c or higher)

   * Let's find all information
     ⇒ Material Science Library.

2. keep going to run Patran.
3. Rock size ? at Yacca Mt.
4. Air gap?
5. Compare Patran vs. "patran + Ingrid"
6. * Finaly we checked missing data.

7/30/98

Aug - 6 - 98

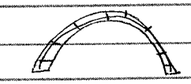
1. Patran work.

   2D Arc 2point option & sweep
   (brief summary)

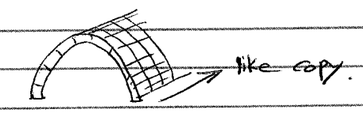
   Geo-metry
   - a. Creat Point
   - b. Creat curve 2Darc 2point (or point)
     - Coord ~~or~~ 1 x
     - 0.2 y
     - 3 z
   - c. Creat surface
   - d. transform surface translate transvector <0 10 0>

FEM
- a. Creat mesh seed

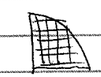
- b. Creat mesh surface Quad 4.



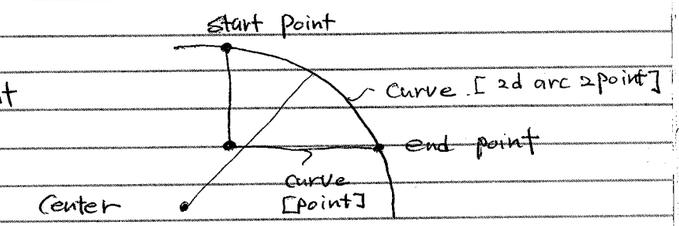
- c. Sweep element extrade



<Note> Each part should be done separately.
(mesh, material property, color ... etc)
Otherwise, everything will be mess up.

2. 

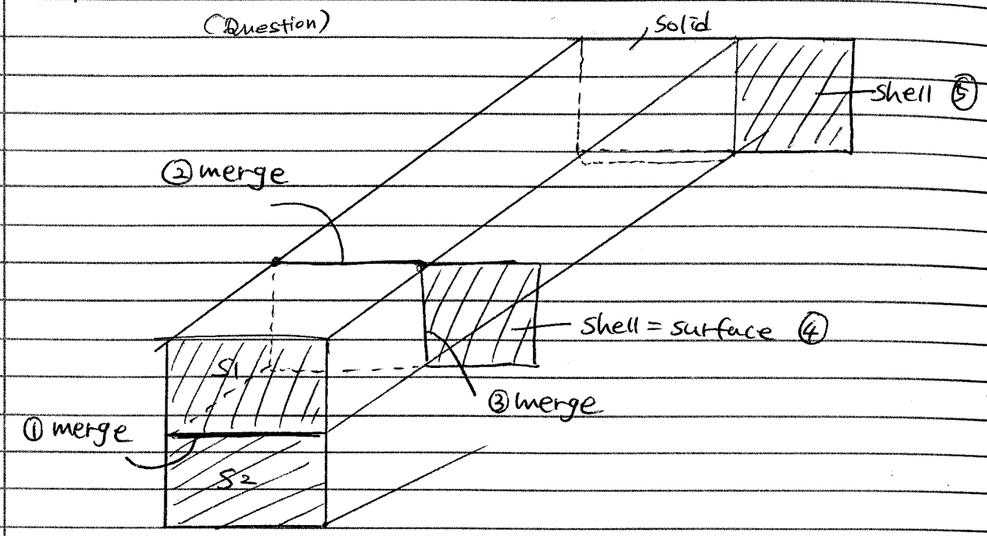
   use two-D arc 2 point



3. We need exact member positions (m)

8/6/98

Pattan Ques ?
(Question)



① Barrier 1 & 2. After sweep the interface can be merged?
② After translate two different mass, they can be merged?
③ Solid + shell. can be merged?
④ Shell = surface?
⑤ How to copy shell?

* I believe these are all to creat meshes!
other things are straight forword!

8/6/98

Aug - 7 - 98

AHC 8/20/2000

1. Node numbers are ~~overlaped~~ overlaped at same position (separate or connected)
2. Inner barrier + diaphram ⇒ How to fit the node position
3. Other parts are sweep straight

4. Creat parts. innerside to outside
(innerbarrier & diaphram first)
only part we need to fit the positions.

8/7/98

Aug - 8 - 98

• Problems.
1. How to define the overlaped surface.
2. How to see separated parts.
3. How to define the connected element.

* Shell is not working.
All solid elements ⇒ Discuss it. (Not proper)

• Use group post → for separation
• Load / b.c → contact
• ½ ?

8/8/98

Aug - 10 - 98

Meeting #2

• Keep going meshs

Pattran
1. part view : [group] - [post] [default (all parts) name

2. merge node : [FEM] - [Equivalence]

* When sweep, define [4 Quad]

8/10/98

Aug - 11 - 98

• Creating mesh

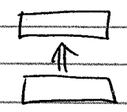
Pattran
1. transform - millor

2. Group
- default : geometry
- thermal shunt
- tube
- waste

⇓

Creat whole model (named package)

* copy each groups
transform



8/11/98

Aug - 13 - 98

Some of assumptions are necessary.

1. thermal shunt and guide rail should be shorter than original to fit the inner barrier surface.

2. thermal shunt length is assumed as it has same length with the inner barrier.

8/13/98

Aug - 17 - 98

1. Creat mesh (diaphram and air parts)
we need check exact position of diaphrams (so complicated)
* group
- diaphram
- air

Almost 80% of meshs has been created.

2. Dimension's are not exactly fit.
ex) inner part's length vs. inner barrier length.

8/17/98

Aug - 18 - 98

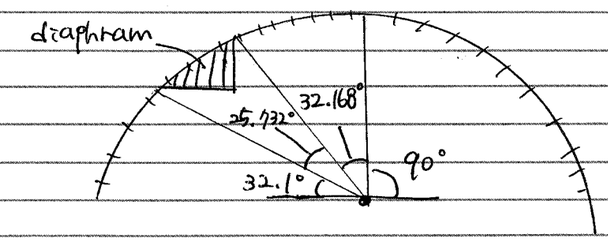
• inner barrier
Geo - trans - revolve ⇒ It's not working.

8/17/98

Aug - 19 - 98

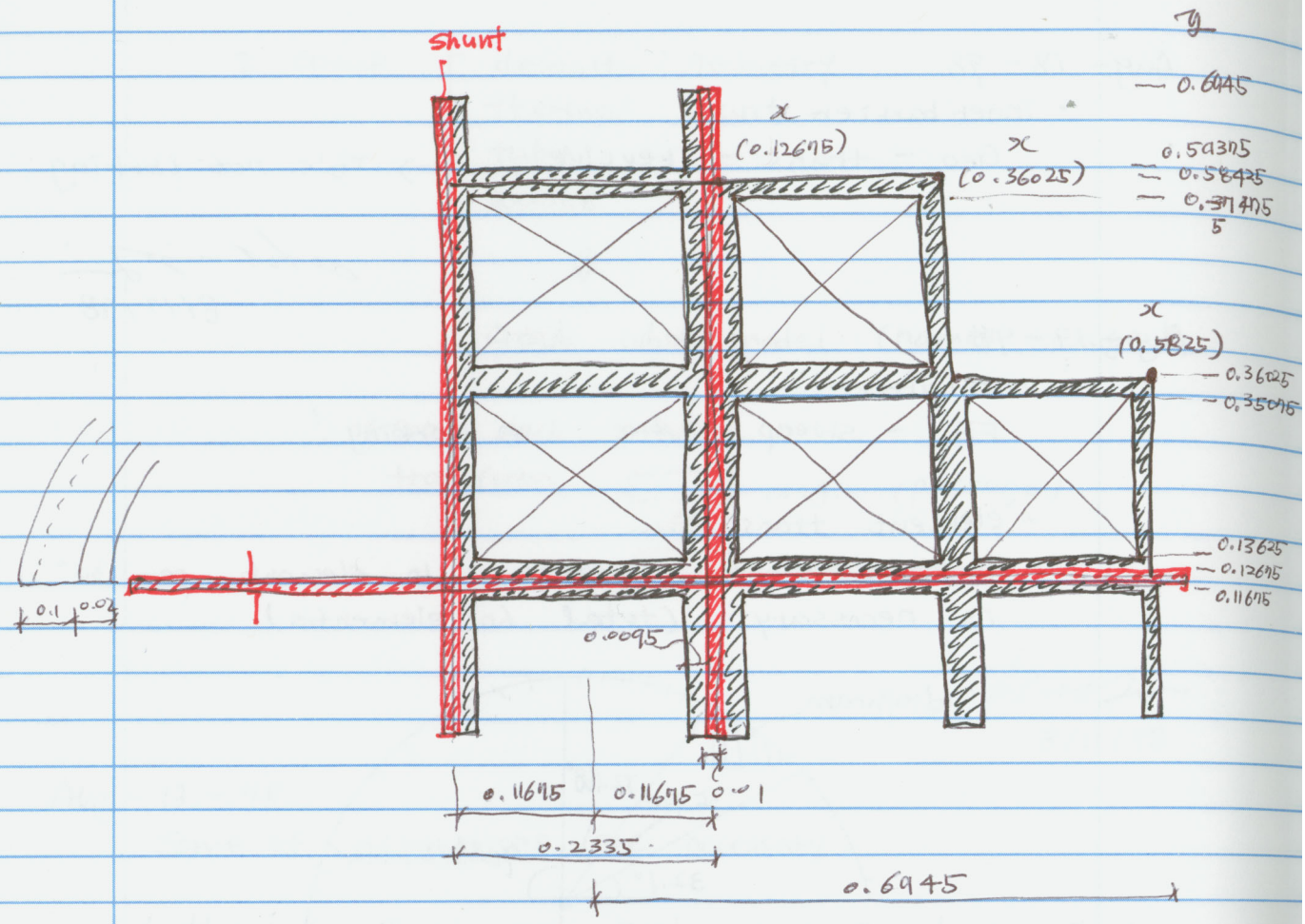
• inner barrier
FE - sweep - Arc ⇒ working!

* element trans form
to make smooth surface 10 element per 30°
is necessary (total 60 elements)

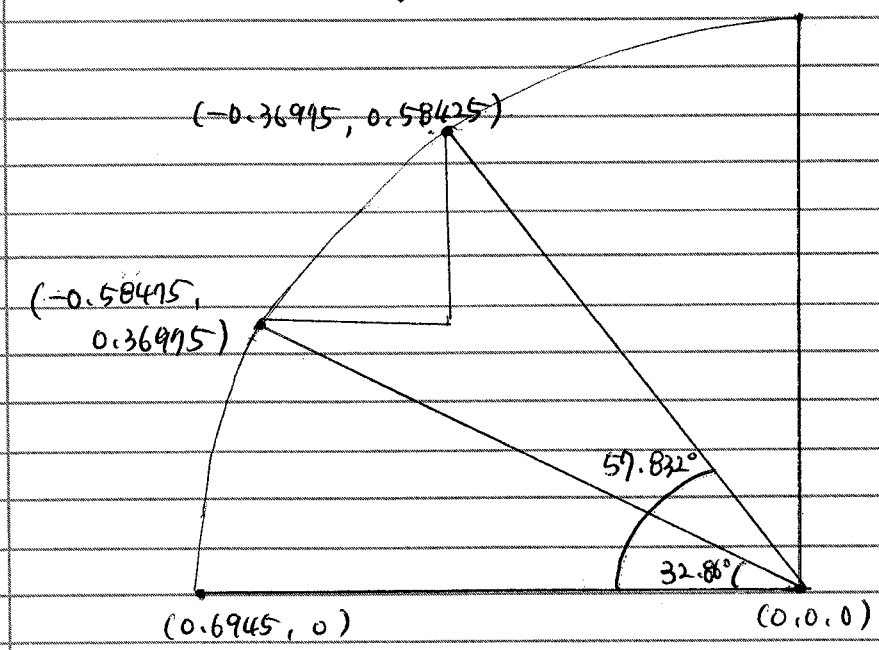


• problem
How to connect inner barrier part and diaphrams?
Do we need?
We will discuss about it at a next meeting!

8/19/98



Brief coordi. of System. (mesh geometry)



8/19/98

Aug - 20 - 98

Meeting #3

· We don't have to connect each parts. Just sliding stuff. After then, we will fix details.

<u>Assum</u> : Inner part lengths are (half).

2.2175 m (Jin), 2.3075 (Jim)

⇒ Not a big difference.

8/20/98

Aug - 22 - 98

· trying to combine the meshs

8/22/98

Aug - 25 - 98

· Problem

1. If I $\boxed{\text{copy}}$ a group, the copied group is changed with the original group.

2. $\boxed{\text{translation}}$ needs a lot of ram space.

otherwise, some of data missing (use better machine)

3. Equivalency – group option

* Finally I successfuly combined separated group meshs.

8/25/98

Aug – 26 – 98

Meeting #4

- Check the material properties.
  We need some of assumption.
- Heat flow in air (Boundary condition)
  or thermor loading.

- support conditions (S.S or something else)?
- Abaqus is working.
  however, we need debuging (explicit version)
- Element number is so large.
  So Explicit version can creat error.

Future problem

1. We need to run trial cases to check the temperature effects.
2. Mainly good mesh ⇒ then parametric study.
3. Impact cases.
   (1 m diameter stone → elastic rock? stiff? soft?)
4. Check both results from explicit & implicit.
   make differences.
5. Over 100 yrs run ⇒ impact. state ⇒ go back to implicit

Input data

- Rock size (initial velocity)
- Material properties (everything)
- B.C and surface conditions
  ⇒ How to go to Abaqus (Both implicit & explicit)

8/26/98

Aug – 27 – 98

- Mesh. for detailed parts (diaphram, guide, cap··· etc) very complicated

8/27/98

Sept. – 2 – 98

- creating guide caps & stone
- All meshes are completed (except air part)
- Next set B.C contact surfaces. loading condition
  material prop.
  air? rock?

9/2/98

Sept. – 10 – 98

- Material properties setting
- We need exact material properties
- Let's assum all = 1 for unknown properties (just for check)

9/10/98

Sept – 14 – 98

- B.C disple < , 0 , > → only y-dir. restricted) simple
  < > → rotation free) -bc
- Impact & contact surface : Load case : imp – stone
  (initial vel. (20))

⇒ * Later, we have put exact numbers.

First run : bunch of error
⇒ Material properties ?
Loading conditions ?

9/14/98

A516

tensile strength: 380 MPa
yield strength: 205 MPA
% elong.: 27
E = 206 GPa
density = 8131 kg/m^3
Poisoon's ratio: 0.3
tangent E (post-yield): 0.6505 GPa

Alloy 825

density: 8140 kg/m^3
poisson's ratio: 0.42
tensile strength: 662 MPa
yield strength: 338 MPA
% elong.: 45
E = 206 GPa
tangent E: 0.772 GPa

316L (316B6A)

density: 7953 kg/m^3
poisson's ratio: 0.298
tensile strength: 482 MPa
yield strength: 172 MPA
% elong.: 40
E = 195 GPa
tangent E: 0.776 GPa

Rock ✓

poisson's ratio: 0.22
E: 32.7 GPa

6063AL ✓

Use the data you found for the aluminum

filler: helium

no structural properties assume. Rock properties.

9/14/98

Sept - 25 - 98

Meeting # 5

- We got information of material properties & B.C
- Start running and check all errors.
- Surface definition (inner barrier & assemblies)
  ⇒ impact ⇒ panatration ?

9/25/98

* Completed contact surface definitions

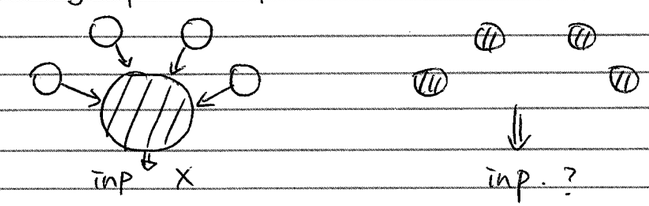
- assumption
  - Initial velo. of Rock - 100 → late change
  - Material prop. → should be checked
  - B.C, S.S at outbarrier surfaces

⇓

With these, initial running to check error.

<Note>

Each groups in pattran doesn't combineded.



9/25/98

Sept - 28 - 98

- rerunning job → still "no properties defined" - error message
  ⇒ however, I set the properties.
- Input data (Abaqus) looks ok. node # (1 ~ 70000)
  element # (little mess)
- Abaqus running : Abaqus job = prun 2 ⇒ need some correction
- Errors : N set = ?
  * step ~ Loading ~ * end ? check
- Big problem : Result can't drawn
  (space is not enough)

9/28/98

Sept - 30 - 98

· Temination of errors in inputdata (Nset = ABC)

· One problem : space (mail to beatric)

⇒ After getting enough space, running should be working.

9/30/98

Oct - 1 - 98

· Memory : Abaqus job = prun 2 memory = 1433169 (X work)
· Explicit : memory = 10,000,000 pre-memory

* post memory ?
Abaqus post res = ◯ memory = ◯

10/1/98

Oct - 8 - 98

· Explicit version errors
1) Change C3D8 → C3D8R element type
2) Move contact info. inside *step
or end step
3) delete Shell element
or input shell material properties

<Note>
Compress filename → zips → p model.db → p model.db.z
Un compress filename.z → open file

10/8/98

Oct - 9 - 98

· Test model (PMO test) ⇒ only $\boxed{\text{out barrier vs. stone}}$
check it work & how much memory it spend ... etc.
after then, add each parts step by step.

· result : looks good.
only 5% space it spent.
total elem 850

* pre - memory = 10,000,000
* still we have no idea about post - memory = ?

10/9/98

Oct - 10 - 98

· EL Print
· NE Print
→ standard, however explicit different.

* Still, Pmotest doesn't work.
no error message, but no analysis ?
We need check manual

10/10/98

Oct - 12 - 98

· explicit
- Abaqus job = ◯
- Abaqus job = ◯ convert = all interactive
- Abaqus post job = Pmo test

* Now it works !

* draw
* contour, variable = S11
* restart, number = 8 (number = 1, 2 ··· 8)

Next time : 1) model (out barrier, inner barrier, assemblies, stone model)

2) check B.C
material properties

3) Stone initial velocity
check Δ label (take default in patran)

elase

4) Contact surface check

10/12/98

Oct-16-98
Meeting #6
- Out barrier element aspect ratio ?

- when run, use batch file - qsub
  → to escape pre-memory problem.
- check internal memory with CAC

- Mesh setup (New one)

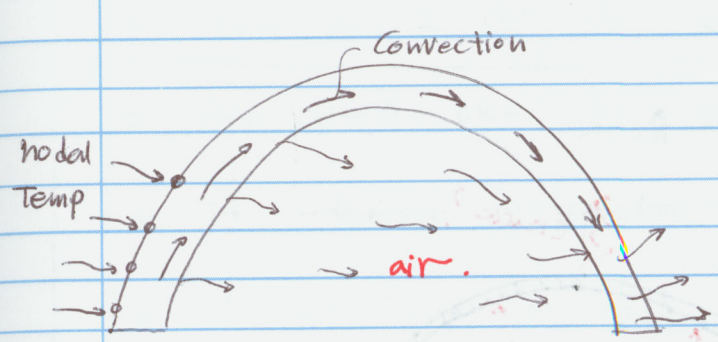
10/16/98

Oct-19-98
* Finally impact model completed.
We need - velocity setting (now - 400)
- B.C
- Contact b/w inner vs assem
- air

10/19/98

Oct - 29 - 98

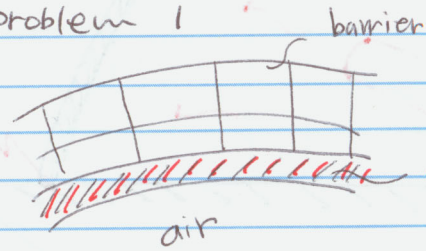
* Thermor
  - Model 1 : Thermo1



1. Temp [temperature (nodal)
   [time dependence

2. Convection
   [① Convection (elem. surface
   [time dependence
   [② Ambient temp.
   [time dependence

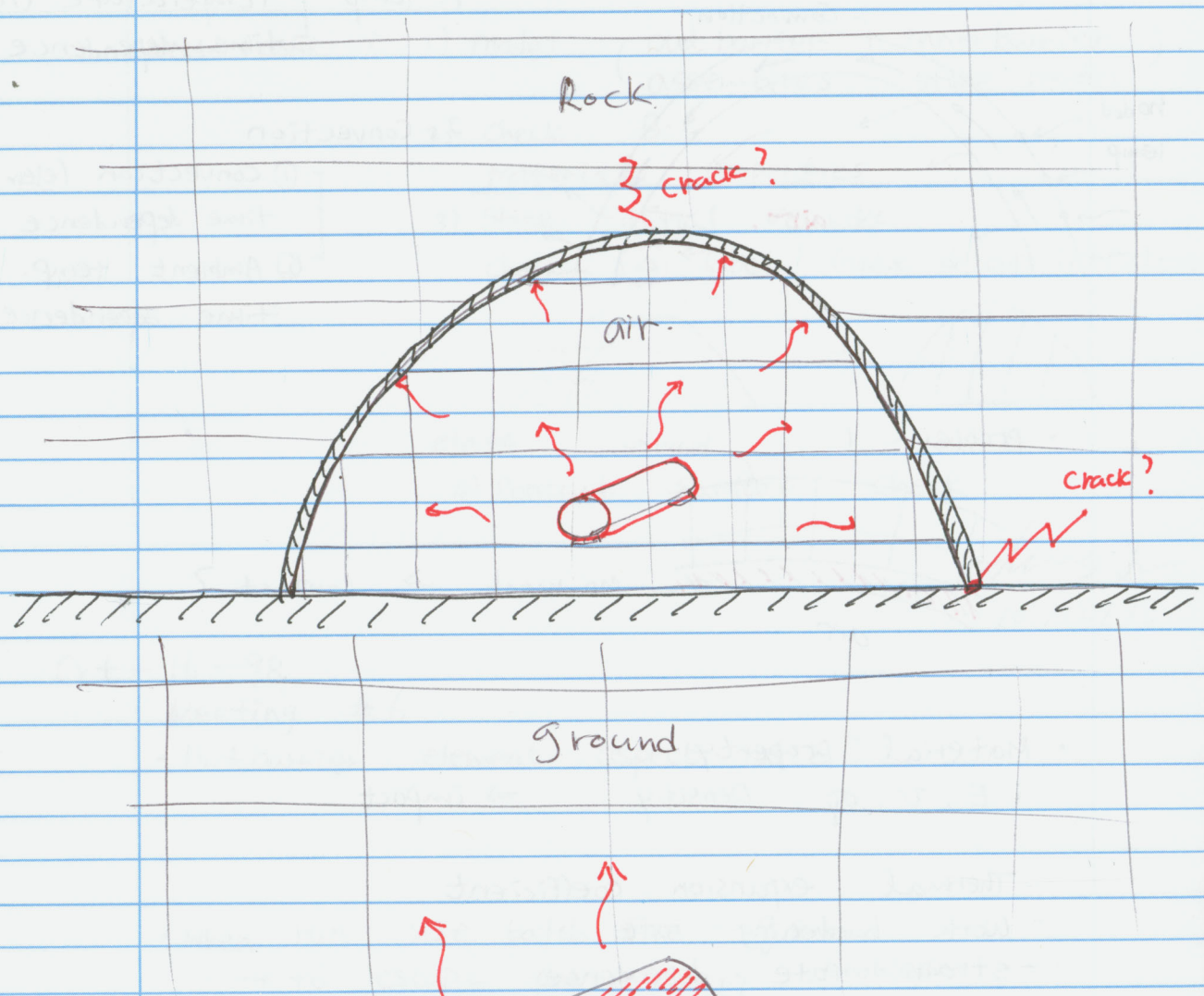
- problem 1

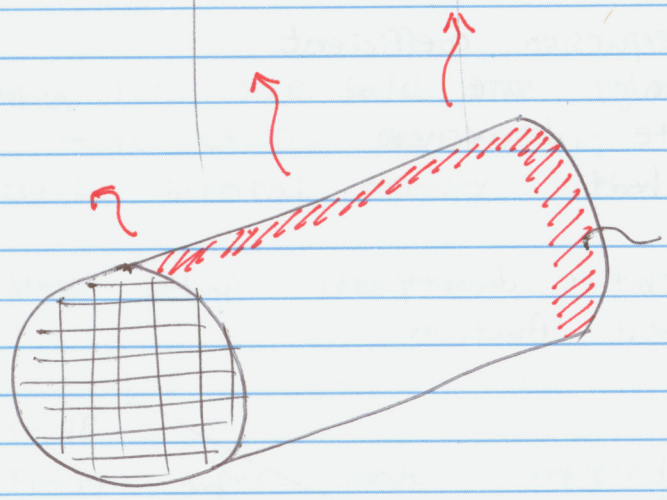


no mesh. → contact ?

- Material property
  : E, ν, σ_y, Density ⇒ impact

  - Thermal expansion coefficient
  - Work hardening rate
  - strain rate dependence
  - specific heat
  - conductivity
  - Initial relative density
  - Inelastic heat fraction





$T(x, y, z)$

$T = T(t)$

$T = T_0 + 0.1^\circ t$

t = month.

when $t = t_0$, all conditions are known.

1. Thermal contact definition.

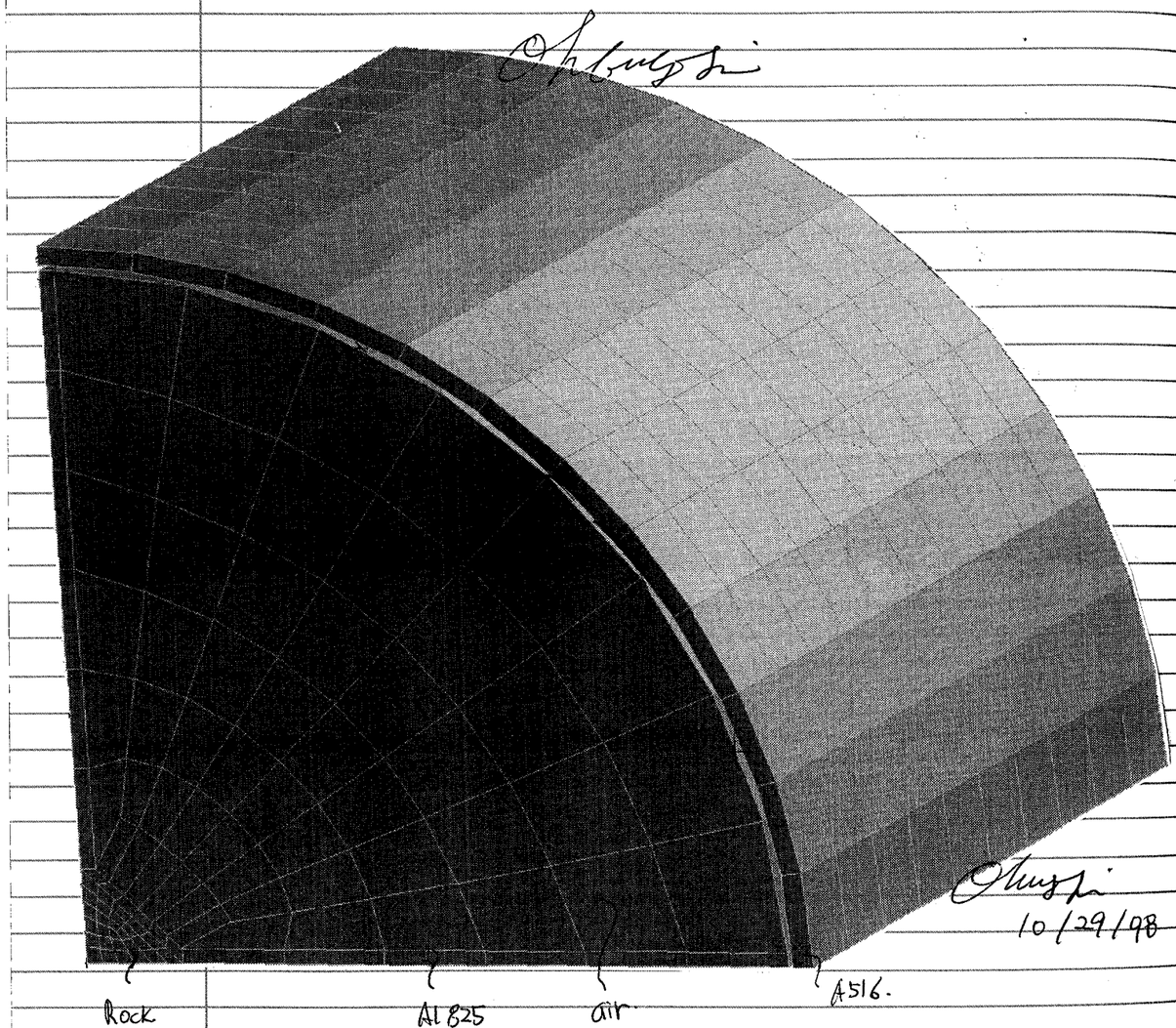
2. How can I cut the meshs at a certain point.

3. We need a plan for groups.

4. If thermal deformation is $f(r, \theta)$. ?

5. If we have air mesh, then, that is a conduction prob.
We can't get heat flow information with Abaqus.

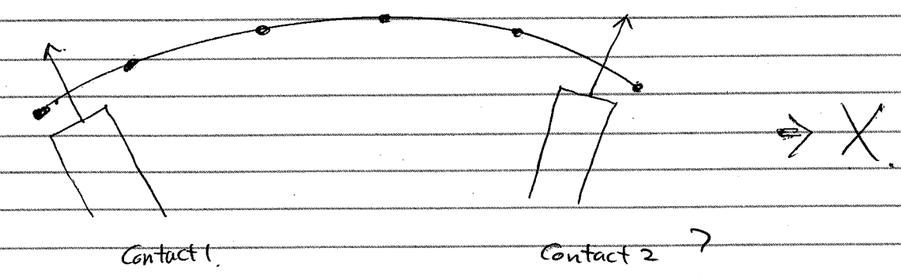
Ohbuyoshi
10/29/98



10/29/98

Nov – 22 – 98

1. First running of thtest 1 model (inflicit)
   - problems
     - pivot = 0 ? ⇒ B.C is too restricted
     - too large strains ? (5 times ··· etc)
     - contact 2 is discontinuous
       it means,



2. We have to choose proper thermal loading condition.
   ① static temperature
   ② initial condition
   Heat transfer
   Boundary
   ⇒ to model $\Delta t = t_0 + 0.1 \cdot t$
   * I think we need this option.

- Next problems.
  - ensure the conduction and contact definition is working.
  - apply heat transfer option.

11/22/98

Dec. – 4 – 98

· Modify the simple thermal model
· patran ⇒ No thermal contact definition.

* We need to set the thermal contact in Abaqus.
* First running ⇒ check warnings.

12/4/98

Dec – 8 – 98

· Modify the thermal model
· When initial condition = heat
use boundary ?

1. Zero pivot : B·C
2. Numerical singularity : Heat option

12/8/98

Dec – 10 – 98

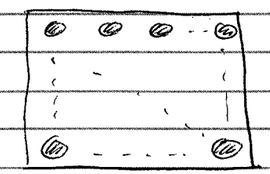
Meeting
– power point (CHWRA)

12/10/98

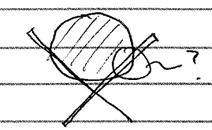
Jan – 15 – 99

Meeting #7

1. No need thermal analy
2. Mesh size
3. CO_2 material information – we got it.
4. Nuclear fuel

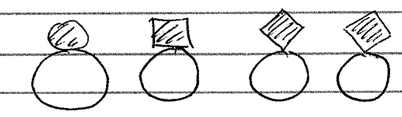


5. B·C

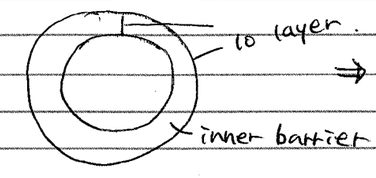


? Simply supported ?

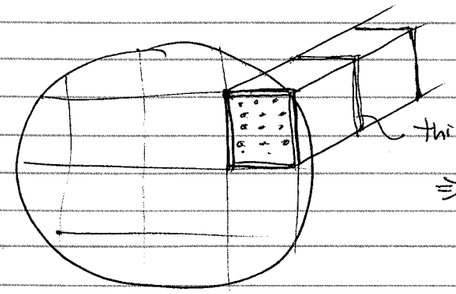
6. Max thermal – at 100 yrs. → During 2 impact event.
7. Rock – sphere, cube
Ø – 1m Rock
B·h = 1m



8. Rate of collision until 1000 yrs.
↓
Two cylinders. each cylinder 10 layer → 1 layer collosion per 1000 years.



⇒ After 1000 year.



this plate take impact like Stiffener.

⇒ material properties.

* procedure
  ① 100 year. impact — Lim
  ② 1000 year. corrosion of layers
  ③ Temp. $PV = nRT$

  $\frac{PV}{T} = const.$ ⇒ gas pressure

1/15/99

Jan – 20 – 99

* New Mesh Model
  1. Plan
     · Barrier model + Stone ~~perst~~ first.
       (more detailed mesh)

  2. We set new out barrier & inner barrier mesh.
     · Dimension has been little bit changed.

1/20/99

Jan – 21 – 99

* Patran mesh (out, inner, stone) = completed

1/21/99

Jan – 22 – 99

Meeting #1 (Lim & Oh)

1. Aspect ratio.

2. Entire Body Model 24 case

   half model 12 case.

3. Modeling schedule.

   1). Base model : ① Abaqus running for 1 case
   ② check all possible errors.
   ③ Running time & mesh size.

   2) Base parametric study

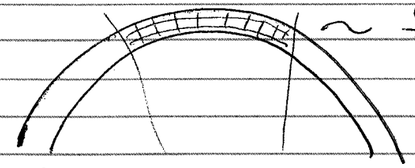
   Inner part mesh ⇒ until (Feb)

4. Whole mesh analysis.

1/22/99

Jan - 22 - 99 (Continued)
Meeting # 8

1. Increase 60 → 120 mesh seed → aspec ration.
2.



~ Siner mesh.

3. Check

1/22/99

Jan - 23 - 99
* B.C set : patran

1/23/99

Jan - 25 - 99
* all set : load / B.C ⇒ input data created

1/25/99

Jan - 26 - 99
* All set : Disbuging ⇒ only problem boundary condition
⇒ check

1/26/99

Jan - 27 - 99
* First running
- Abaqus memory problem (Post)
- B.C is not working
(message : Boundary multiplication should be 0 ?)

so, we decided all 0 ⇒ means. Fixed condition

- Sub is not working.

1/27/99

Jan - 28 - 99
1. Impact running time 0.2 ms (about 40% of space)
2. Converting done.
3. Looks fine. all data is ready to go.
(post memory problem)
- Node # 30,000 ⇒ how to deal ?
element # 20,000
4. First model done.
<Note>
• Impact initial velocity. 12.4 m/s
too small (small effect on cylinder)
we check 1000 m/s initial velocity

• running time is 0.2ms
→ we need more.

*however, memory problem is so serious.

1/28/99

Feb - 4 - 99
• output data ⇒ cut plane and see inside.

set clipping = on
set clip plane = (0, 0, 1)] → good option

2/4/99

Feb - 5 - 99

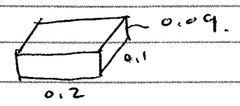
- 3 cases [5m, 6m, 8m] ⇒ Obviously 8m is critical situation.
- inside pressure
- Temperature
  : step (implicit) → impact (explicit)
  thermal stress → initial stress of impact
- Mesh sensitivity on small scale model (Hyperthetic)

2/5/99

Feb - 6 - 99

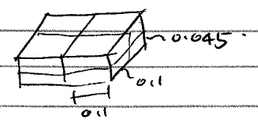
* Hypothetic model

1. aspect ratio 1:2.



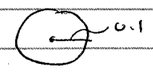
element size.

2.



Check the effect of fine mesh !!

3. Stone



2/6/99

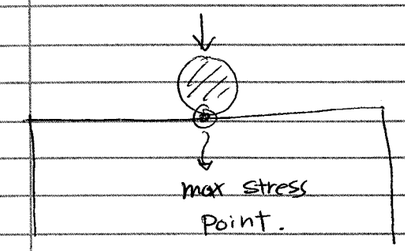
Feb - 7 - 99

Model 1 : 500 m/sec → 50 m/sec (Hypothetic model)

2/7/99

Feb - 8 - 99

1. model 1. done! rock → r = 0.2.
impact → 200 m/sec



Stress : Sylinder ≈ 3.1E+8.
Mises : " ≈ 2.2E+8.
max !

2/8/99

Feb - 13 - 99

1. ~~Mom~~. model 2 (finer mesh model)

Max stress = 2.05 E +8.

2. Conc.: The mesh size doesn't effect too much
   - more dense mesh around impacted area,
   - and possible rough mesh around other area.

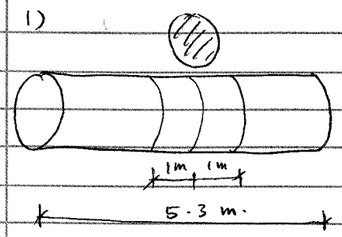
⇒ Reduce mesh size !

2/13/99

Feb - 15 - 99

1. meeting.

1)



⇒ 12,000 element mesh. << 15,000 msb.

2) Summary our work

3). Inside mesh.

4). B.C ?

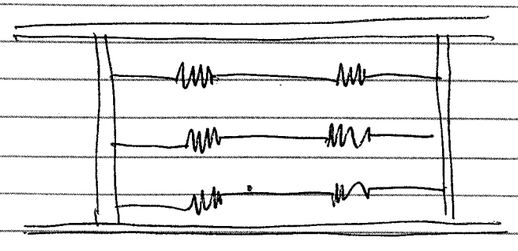
5) Vijay ⇒ 2 computers (Dopson, Cuellar)

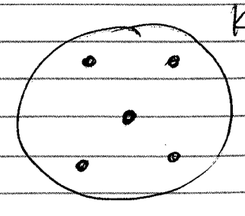
Check possible space. in them.

6) now study Shell (thick Shell)

⇒ possibly ~~apply~~ applicable. ? A+C 8/30/2000

7). Inner part model ?





$K = \frac{AE}{L}$

2/15/99

Feb - 21 - 99

* CNWRA manual check.
* No plate spacing : where this idea from?

1. Calculate total density of all length assembly & Fuel.
   ⇒ parametric study [5 division plate spacing / 8 " / 10 "] ⇒ Find critical one.

2. Fuel load properties?

2/21/99

Feb - 22 - 99

Meeting # 2 (Lim & oh)

1. Shell mesh → thickness is not drown
   → stress value reasonable.

2. aspect ratio 1:2 - 1:4 what happened?

2/22/99

Feb - 24 - 99 A+C 8/30/2000

~~CRWMS~~ CNWRA (M&O) [3-D model : stress check - only out barrier
2-D model : stress & displ. check - only for assemblies.]

- assumption ① Fuel assembly weight - neglected - small stress
  ② 3-D model - aspect ratio warning - disregard.
  : stress gradient of each element is not significant.

- problem [No properties of fuel rod & others in side assemblies
  - we need to assume the plate spacing.
  - we need more detailed figures.]

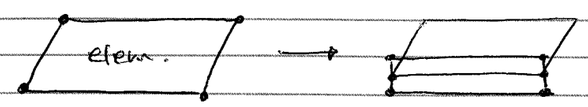
*meeting

1. Check shell theory manual ⇒ Which shell element, They can use
2. If it's not working ⇒ 3D aspect ratio

· 119 / cm³ → check the fuel waste density

*Shell model

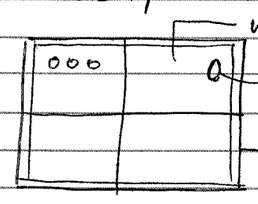
- Correcting data
- Wave (Vibration problem) - Lim
- Theory manual check.
  i what kind of shell calculation create thickness view?
  we need more check!



2/24/99

Feb - 27 - 99

* Fuel density calculation



What kind of plate?
Density of fuel?
assembly is known

* Fuel material (Uranium dioxide)

Density : 10.97 g / cm³ → should be 2X when Sintering
σu : 6.9 X 10⁸ pa (10,000 psi)
E : 1.72 X 10¹¹ pa

2/27/99

Feb - 28 - 98

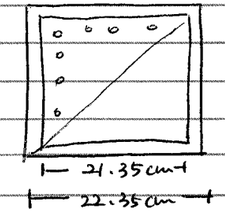
*Shell Theory

: All shell analysis assumed thickness change is constant & uniform.

- This assumption should be considered of primary importance in some applications.
  such as ① failve of pipe
  ② over - pressure vessel.

* Questions

1. Zircaloy spacers in the fuel assembly (material properties & spacing distances)
2. Material density is increased ~98% when Sintering what about σu & E ?
3. Cladding means (or has) any mechanical resistance (σ & ε)



⇒ 213.5mm

PWR

15 X 15 Fuel assembly.

pelled o.d : 9.1 mm = 0.0091 m
Cladding thickness : 0.6mm = 0.0006m
9.1 X 0.6 X 2 = 154.5mm = A of rod.

total A of fuel rod = 18738 mm²
A of fuel assembly plate = 26844.25 mm²
A of 1 assembly = 45582.25 mm² (inside)

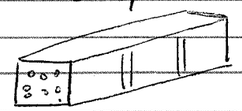
Fuel material → Fuel pellets → sintering & pin loading
→ Fuel rods (pin) → insult support grid plate (spacing)
→ ~~Fuel~~ assembly
Fuel

2/28/99

Mar – 1 – 99

Meeting check point

1. Shell model is not reasonable
2. Fuel assembly check.



15X15

3. Questioned problems.
   - 3-D & shell combination
     - 3-D → high density mesh around impact
     - shell → other part
   - 3-D mesh : up to 50 ms → No problem
   - check storage problem.

3/1/99

Mar – 5 – 99

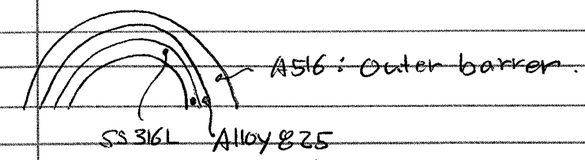
* Questions

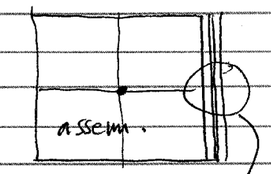
1. Assembly, 15X15, pellet O.d = 9.1 mm, Cladding thickness = 0.6 mm
2. Cladding material (Zircaloy 2?) ⇒ material properties
3. Pellet density after sintering & burning
4. Fuel pin spacer ⇒ Zircaloy2 ⇒ material properties
5. 16 guide tube
   1 instrument tube
   1 spacer sleeve
6. Typical spacer spacing

3/5/99

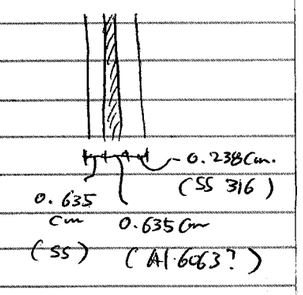
Mar – 6 – 99

1. Model check.?

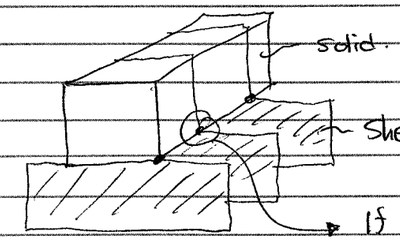




Also. check all detailed parts!



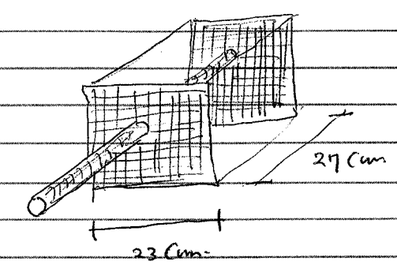
2.



If this point is not matched ? offset!

⇒ • Check contact. options.
• becomes very hard to model inner assembly.

3. Fuel assembly.

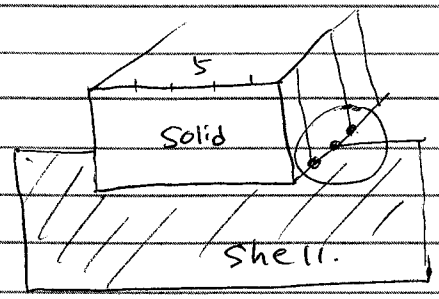


• Fuel Assembly model. Simplified.
• working ?

3/6/99

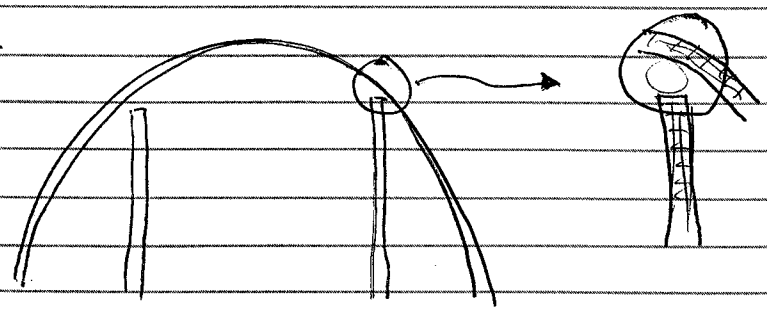
Mar - 15 - 99

1. Offseted point ~~Contant~~ Contact check.



⇒ No ~~prof~~ problem.

2.



Contact ?

⇒ No problem.

All Set!

3/15/99

Mar - 17 - 99

* Meeting (Lim & Oh)

3/17/99

Mar - 18 - 99

* Mesh size estimation ≈ 15000 element

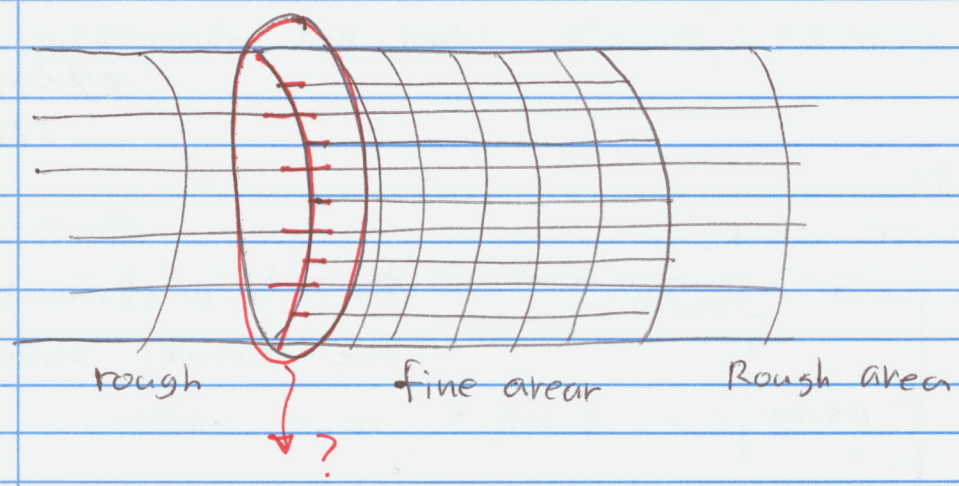
approx [out : 3500) → Quad 8
in : 2000) ⇒ 8000 element
others : 10,000 → Quad 4 → 1000 element]

3/18/99

Mar - 19 - 99
Meeting

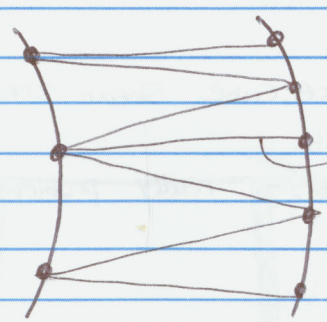
3/19/99

Mar - 20 - 99
* Transient area



Patran option.

Surface - Solid - different mesh seed



Creat curves by manually.

looks complicated., but
nochoice. !!

3/20/99

Mar - 21 - 99
· Outbarrier geometry including transient area
· mesh develope ment

3/21/99

Mar - 22 - 99

Outbarrier

transient - solid & small section

3/22/99

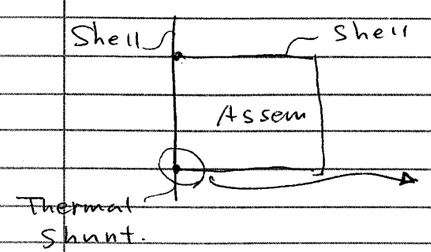
Mar - 23 - 99

innerpart shell members - possible error check.

3/23/99

Mar - 25 - 99

inner part check.



Use Solid element for the thermal shunt. !!

When equivalence, one element is gone.

3/25/99

Mar - 28 - 99

- Thick shell → can take care shear transmission?
  Then use shell elements.
- Aspect ratio 1:4 ⇒ got really problem.
- half model

3/28/99

(Continue)

- gravity

- gravity effects and model option with gravity.

3/28/99

Mar - 29 - 99

- Shell outbarrier and innerbarrier and contact
- Contact surface has problem
- geometry problem

3/29/99

Mar - 30 - 99

- Outbarrier half model. planning & geometry

3/30/99

Mar - 31 - 99

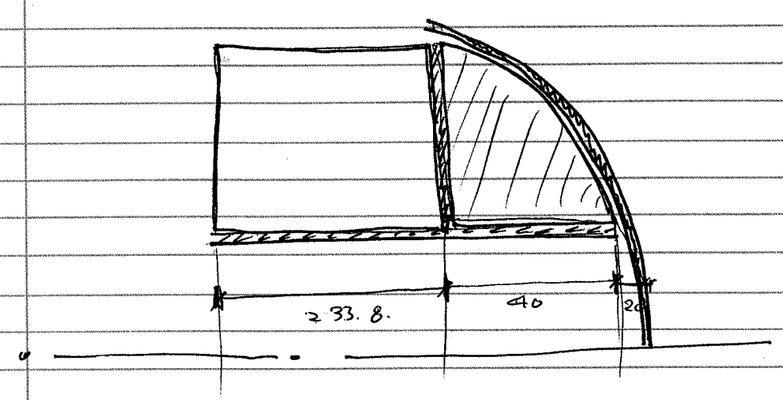
- shell + solid model for inner assembly → possible error check

- contilever beam & gravity check.

3/31/99

Apr - 12 - 99

* inner assembly : dimension should be modified
  (Not very much)



4/12/99

Apr - 13 - 99

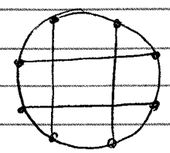
· inner part everything mesh

· patran work

4/13/99

Apr - 14 - 99

· innerpart new design



inner barrier & assembly is connected.

· use C3R8I element in abaqus

4/14/99

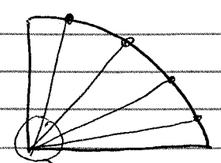
Apr - 15 - 99

· Inner part geometry (all connected, innerbarrier & assemblies)

4/15/99

Apr - 16 - 99

· Inner diaphram



Too sharp !!

It is expected later error.

Too much distorted when impacting

4/16/99

Apr - 17 - 99

1. Material propert

Density
E
σ_y
ν

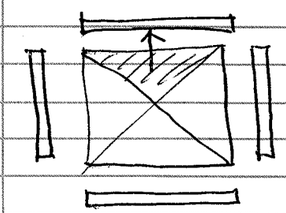
} E, b, h

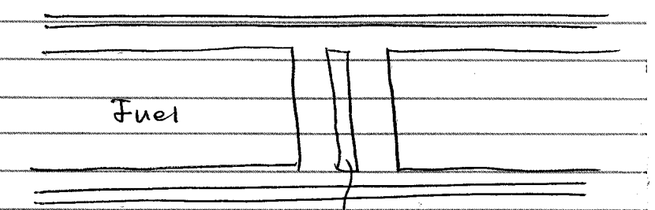
· If simple tension

$\sigma = E\varepsilon$, $P = \frac{EA}{L}\Delta$ $\Rightarrow E$? A ?

· If bending

$M = EI\phi$, $E \cdot I$ $\Rightarrow E \cdot b \cdot h$?





A of shell ? to satisfy $I_{total} = I_{shell} \frac{bh^3}{12} = \frac{bh_1^3}{12} - \frac{bh_2^3}{12}$

$\Rightarrow$ Shell t ?

$E = \frac{E_1 E_2}{E_1 + E_2} \Rightarrow E_{equivalent}$.

2. Inner part

4/17/99

Apr - 18 - 99

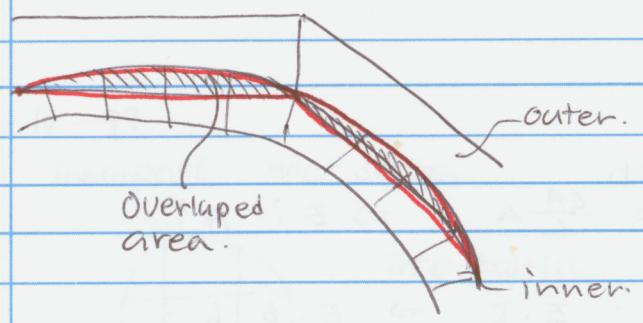
· Inner part completed but big problem.

out - inner barrier contact.

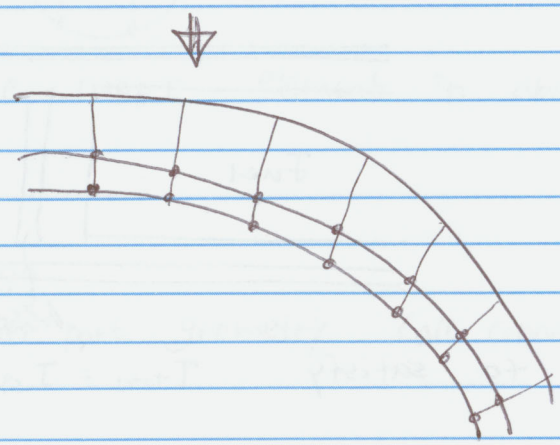
4/18/99

Apr – 19 – 99

* meeting



⇒ Create Contact errors.



⇒ Big mesh size!

4/19/99

Apr – 20 – 99

· Fix inner barrier
· geometry & meshes.

4/20/99

Apr – 21 – 99

· mesh completed, contact & loading
⇒ check first simulation.

4/21/99

Apr - 22 - 99

- inner Contact
- Shell contact , definition

4/22/99

Apr - 23 - 99

- first run
  - Shell is too sharp → warning
  - □ → ▱ distorted error
  - contact
  - unknow element set

- Error correction

4/23/99

Apr - 24 - 99

- Correcting errors of first model
- We will try without shell diaphrams

4/24/99

Apr - 25 - 99

- Trial run w/o shell diaphram
  - Shell contact option is working.

4/25/99

Apr - 26 - 99

- shell part (diaphram modify)
  - based on test model

4/26/99

Apr - 27 - 99
· unexpected error
  - error 39, 49 → computer has problem.
· Half mode try.

4/27/99

May - 1 - 99
* Meeting

· Half model modify (hmo) [contact / diaphram]

5/1/99

May - 2 - 99
· Half model completed (hmo)
  - error correction & data analysis

5/2/99

May - 3 - 99
· Material properties
  - Zinc alloy
· Time step of abaqus check
  - $\Delta t = \frac{2}{\omega}$ $\omega^2 = \frac{k}{m}$
  or use direct option in explicit version

5/3/99

May - 6 - 99
· Block option (Dyna 3D)
  : El set, el history (chosen element all history plot)
    → all automatic time steps are restored
  otherthing → just time step numbers.

5/6/99

May - 10 - 99
1. block test
2. simple model test

* history output, time = 0
* el history, elset =

⇒ Working !!

5/10/99

May - 12 - 99
· Half model (hmo) rerun with history file
  (input data modify)

5/12/99

May - 13 - 99
· Half model (hmo) completed. (data, output ⋯)

5/13/99

May - 14 - 99
1. Meeting
  - check 3 elements for out data
  - check inner sides
  - check support conditions.
  - summary [mesh info. / B.C / material properties / assumption]

2. Material properties equivalent estimation.

5/14/99

May - 15 - 99
· Model (hmo) modification and rerun

5/15/99

May - 16 - 99
· continue ...

5/16/99

May - 17 - 99
1. Meeting
- error check
- make perfect simulation
- report

2. Material properties re-estimation
- assembly thickness 0.0178

5/17/99

May - 18 - 99
1. Modified model and run
- change material properties
- equivalent dimensions

2. Results
- 0° hit system go to fail
- Inside has some damages (high level of stress)
- High accel & deform

5/18/99

May - 19 - 99
1. Meeting
- Check $\Delta t = \frac{l}{c}$ ~ deformspeed
v (wave speed) $= \frac{\sigma}{\rho c}$. $v > c$ convergence problem.
- change $E \uparrow \rightarrow c \uparrow$ for assembly
- check impact beam simulation

5/19/99

May - 20 - 99
· E, t change is not working
need new idea

· solution for wave / deform speed error
- check contact
- contact : small → tied option

5/20/99

May - 21 - 99
· rerun half (hmo) model

5/21/99

May - 23 - 99
· Report & data collection

<Note> C3D8R → aspect ratio 1:3
C3D20R → " 1:100 possible

5/23/99

May - 24 - 99
* Meeting

* Full model. hmfo. db mesh

5/24/99

May - 25 - 99
· Full model mesh completed

· First run ⇒ unknown error?
(guess contact problem)

5/25/99

May - 26 - 99
· Full model o.k.

· Full model run and check o.k

5/26/99

May - 27 - 99
· Full model contact and run

· Big problem
- assem _ diaphram contact error
(need to find other options for contact definition)

5/27/99

May - 28 - 99
· Full model completed

* [verify vector normal
[equivalent - check not used nodes at space somewhere.

5/28/99

May - 29 - 99
· Data Report
- half model (all deformation values)

5/29/99

May - 30 - 99
· Data Report
- half model (all deformations)

Check points	elem #	Full model with 0° node #
· Outbarrier	106276	198535
· Innerbarrier	72485	148321
· Structural - stifner	109362	132208
· thermal stunt	108353	134116
· Tube assem	115416	204909
· Spacer	110491	204993
· B.C	106109	198191

5/30/99

June - 1 - 99
· Full model
- assem - diaph contact correction
- some errors checked.

6/1/99

June - 2 - 99
· Full model completed and run
· data collection and error check.

6/2/99

June - 3 - 99
· Half model data collection
· Full model running check.

6/3/99

June - 4 - 99
· Full model 10 ms run completed.

6/4/99

June - 5 - 99
· Full model output check.

6/5/99

June - 6 - 99
· running check.

6/6/99

June - 7 - 99
· Full model 6ms & 200 ms runs.
· Contact force check.
(only "contour, var = cpress" available)

6/7/99

June - 8 - 99
· Full model 6ms again

6/8/99

June - 10 - 99
· running check
· Full model 200 ms data check.

6/10/99

June - 11 - 99
· Data collection for report

6/11/99

June - 15 - 99
· Impact beam simulation
- Abaqus reliability check.

<Note> Comparing with experimental data, we can check the abaqus reliability for our impact problem.

6/15/99

June - 18 - 99
· Impact beam data & result check.

6/18/99

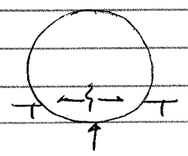
June - 20 - 99
· Final report outline.

6/20/99

June - 22 - 99
· Impact beam
· Lim & Oh discussion
· Impact beam

6/22/99

June - 23 - 99
1. Meeting
- change new information and run most critical case.
- what happen to the rock?



check this stress.

2.
2. Abaqus plastic option

3. Material properties re-estimation

6/23/99

June - 24 - 99
· Material re-estimation
· Half model innerpart modify

<Note> ① Since, Zincaloy and Inconel material properties are newly arrived, we need to estimate the equivalent properties
② We changed most of innerpart assemblies.

6/24/99

June - 25 - 99
· Half model modify

· Full model modify
(material properties & inner assembly parts)

6/25/99

June - 26 - 99
· Half model 0° running & data

6/26/99

June - 30 - 99
1. Meeting
- check all missing data
- complete half & full model
- check tension cut material property

2. Half model data collection

3. Full model correction (inside)

4. Full model continue.

6/30/99

July - 1 - 99

1. Full model 0° mesh completed & 500 ms running.
   <Note> until 6ms, the output looks fine.

2. Half model 0° 500ms running start
   some data collection.

7/1/99

July - 2 - 99

- Half 0° and Full 0° data collection.
- Tension cut material properties finding.

7/2/99

July - 3 - 99

- Half 0° and full 0° data collection
  (280ms) (280ms)

7/3/99

July - 4 - 99

- Collecting data continued
- Hmo stoped at 3282 min
- Fmo - n still running.

<Note> After we changed inner part assemblies,
data shows little smaller deformation value,
but not significant.
We hope 500ms is enough duration to see all data.

7/4/99

July - 5 - 99

- data collection.

<Note> Some deformation data for residual one is not exact.
But, since it's very small and 380ms is enough time, they might be acceptable because they are not major axis deformation.

7/5/99

July - 6 - 99

- data collection (Stress & Stains)
- Fmo - n (full model) re-run for 500ms

7/6/99

July - 7 - 99

- Half model (Hmo) data collection.

7/7/99

July - 8 - 99

- Half model (Hmo) data done.

7/8/99

July - 9 - 99

1. Full model, edge model (0°) mesh

2. Full model (Fmo - n) data collection.

7/9/99

July – 10 – 99
1. Full model edge (0°) mesh barriers.

2. Fmo-n model re-run (some machine error)

7/10/99

July – 11 – 99
· running check (Fmo-n)

7/11/99

July – 12 – 99
· Data check (Fmo-n)

7/12/99

July – 13 – 99
· Data check (Fmo-n, Hm30)

7/13/99

July – 14 – 99
· Data check, Fmo-n inner part modification

7/14/99

July – 15 – 99
· Data check (Hm30)
Fmo-n re-run

7/15/99

July – 16 – 99
· Data check (Hm30 done)

7/16/99

July – 17 – 99
· Data check (Fmo-n)

7/17/99

July – 18 – 99
· Data check (Fmo-n)

7/18/99

July – 19 – 99
1. Meeting

2. Summaries (running files)
3. Heavier rock → increase rock density
: Don't exceed 5m dia rock.

• Report
- Summary
- conclusion
- recommadations

4. Fmo-n10 (10ms, restart file = 5)
Start to run (at H30, ptph.psu.edu machine)
: No history file. only restart files.

5. July report plan
- · store all runned file (zip)
- · abaqus reliability test (impact beam)
- · equivalent rock size estimation
- · impact forces check
- · prepare figures

6. All models are runned for two cases.
   ① One is up to 500 ms. for history files
   ② another is up to 10 ms for output files.

7. Equivalent rock density : for bigger rock.

Rock size		density
1 m dia	—	?
2 m dia	—	?
3 m dia	—	?
4 m dia	—	?
5 m dia	—	?

7/19/99

July-20-99
- Fmo-n10 (10ms) - 5 contact force check
- Fmo-n1 (1ms) - 5 contact force check
- archive data

7/20/99

July-21-99
- Cubic corner (hmmcco - 500ms) run
- Cubic corner (hmcco - 10ms) stored

7/21/99

July-22-99
- Hm0 - 10 running.
- cubic flat - 500 checked
- hm30 - 500 checked
- hm45 - 500 ~~et~~ run
- cubic vertax - 500 run

7/22/99

July-23-99
1. Meeting - prepare report

2. hang on all 10 ms cases
3. re-run Hmo - 500, Feo - 500 (upto 500 ms) and check u.v.a files
4. run more than 500 ms for 1 case (Fmo-n) to see all possible curves.
5. Until sun, we'll complete all runs.

7/23/99

July-24-99
- 10ms runs completed. (figures and contact forces)
  - Fmo - 500 (320 ms)
  - Fmo - 10
  - Hm30 - 10
  - Hm45 - 10
  - Hmcpo - 10
  - Hmcfo - 10

7/24/99

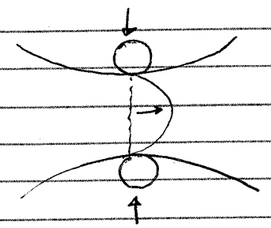
July - 25 - 99

1. 1 ms running.

Hm45 - 1
hmcp - 1
hmcf - 1
⇒ Start from 64 % used space.

2. Fmo-n-500 data check.

: job stopped at 320ms.
still looks bigger deformat at Y direction.



Contour effect?

However, the y direction deformation (S.E) is not so significant.
What's that mean??

7/25/99

July - 26 - 99

1. Meeting

2. Data check (Contact forces)
all 1ms cases done.

3. Fmo-n : assemble el number change - re-run

4. Fmo case - fail stone size estimation

7/26/99

July - 27 - 99

1. All data check. report figures.
2. Fmo-n impact rock size re-running.
3. Impact beam simulation data check

4. Failure model for Fmo (support impact)
5m, 3m, 1m runs

7/27/99

July - 28 - 99

1. Meeting

2. Failure model

7/28/99

July - 29 - 99

1. Meeting
- failure model investigation
- set subscrib file

2. Failure made Fm - series
- Start to run 1.5m, 2m, 2.5m Rocks
3m, 5m

7/29/99

July - 30 - 99

1. Meeting

2. Failure model check.

7/30/99

Aug-1-99

1. Failure model
   : 1.5m, 2m, 3m, 5m Running check.

2. Fm 1.5 Fm 2.0 Fm 2.5 data check
   <Note> ① outbarrier investigation
   at around $\boxed{2.5}$m rock, system might be start to fail.
   ② need to check 150ms (4.5 days to run?)

8/1/99

Aug-2-99

· Fm3m, Fm 5m data & running check
· Fm 2.5, 150ms runs & data check

8/2/99

Aug-3-99

1. Fm 5m, 3m data check (stress, strain)

2. Fm 1.5, 2. 2.5 running check

8/3/99

Aug-4-99

1. Fm 1.5, 2. 2.5 data check

2. Data archive, re-run Fm 3m

8/4/99

Aug-5-99

· Abaqus manual check
· Report correction

8/5/99

Aug-19-99

1. Data transfer

2. Fm-3m-10ms re-run

8/19/99

Aug-20-99

· Fm-3m-10ms run completed.
· Data transfer

8/20/99

Aug-23-99

1. Meeting

2. Data transfer to sponser

3. Fm-1m, Fm-3m re-run (90ms)

8/23/99

Aug-24-99

1. run check

2. contact force check

* All datas are ready to ship!!

* Code validation models
  : To check performance of abaqus, Dyna 3d ...

8/24/99

Aug - 26 - 99

1. Meeting
   - beam model verification - parametric study (displacement & strain energy)
   - Cylinder → also try parametric studies

2. pattran setting and mesh beam

3. Answer to sponsor (bunch of questions)
   - beam material properties (ultimate strain)
   - Mesh is lumped or consistent?

8/26/99

Aug - 27 - 99

1. Review the Lui's paper

2. Impact beam. parameter study (mesh density)

3. Meeting

8/27/99

Aug - 28 - 99

1. Impact beam continue

2. Fm - 1m, Fm - 3m running check

8/28/99

Aug - 30 - 99

· Impact beam continue (mesh portion)

8/30/99

Aug - 31 - 99

1. Impact beam continue (data discussion)

2. Meeting.

8/31/99

Sept - 1 - 99

1. Abaqus Double option (precision)

2. Beam impact model with double option
   Re - run

9/1/99

Sept - 2 - 99

· ls - dyna , ls - ingrid options
· Beam impact model creation

9/2/99

Sep - 4 - 99

· Beam impact model (dy3d) input file

9/4/99

Sep - 5 - 99

1. Beam impact model mesh completed

2. d beam b run and check

9/5/99

Sept - 7 - 99
1. d beam c and d beam d meshs

2. Data check d beam c & d beam d

9/7/99

Sept - 8 - 99
1. ls - stuff investigation

2. d beam g , d beam h (1:2:2)
running and check

<Note> Based on Liu's paper, creat proper mesh
⇒ Impact beam models.

9/8/99

Sept - 9 - 99
1. Meeting
- use abaqus, Dn3d, ls-dn3d
- use 6 cases, #40, #80, #640 ⇒ 1:1:1 and 1:2:2
- use some material properties
  · elastic
  · elastic - plastic
  · elastic - strain hardening

9/9/99

Sept - 10 - 99
1. Meeting
- use elastic, elasto plastic, elastic-tangent to compare codes.
- use full plastic for code variation
- ls - ingrid, dyna simulation.

9/10/99

(continue..)
2. ls - ingrid mesh modeling (beam case)
input generated using ls - ingrid.
"error" ⇒ unauthorized use.

9/10/99

sept - 11 - 99
· Dyna 3d input data modify

9/11/99

Sept - 13 - 99
1. Impact beam (E & strain hardening)
- parametric study

2. Result report

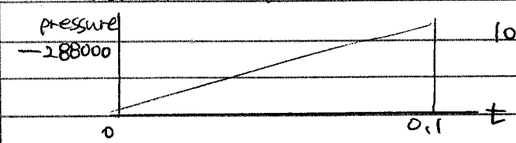
9/13/99

sept - 16 - 99
· Abaqus standard tensile simulation
- plastic
- static pressure

9/16/99

Sep - 19 - 99
1. Abaqus standard tensile simulation done.
- Data Report.

2. Abaqus explicit tensile simulation
- Duration 100ms (0.1 sec)



loading curve.

9/19/99

Sep - 21 - 99

1. Dn3d material test
   - material type #10

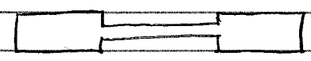
2. Dn3d material type #3 (elasto-plastic)
3. set and run

9/21/99

Sep - 22 - 99

1. Meeting

2. State - equation form $C_0 = 1$, $C_1 = K$ (bulk moduls)
   $V_0 = 1$, $C_2 = 0$
   ⇒ try.

3. Aba - material test (change )

9/22/99

Sep - 23 - 99

1. Dyna 3d E sensitivity test → totally different?
   (impact beam case)

2. Dyna 3d material plastic model → still not working.

9/23/99

Sep - 26 - 99

1. Meeting
   - Find proper option for plastic material type.

9/26/99

Sep - 27 - 99

· ls-dyna impact beam running (ndbeam b case)

· ls-taurus investigation.

9/27/99

Sep - 28 - 99

· ls-dyna beam impact case b → done
  <Note> ls-dyna performs pretty good for this case
  → creat all othe models.

9/28/99

Sep - 29 - 99

· ls-dyna material property check
  : elastic , elasto-plastic ...
  → Find proper options.

9/29/99

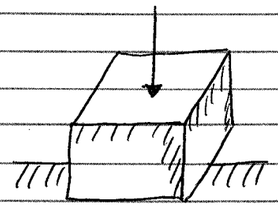
Sep - 30 - 99

· ls-dyna impact beam c, d, g, h done.

9/30/99

Oct - 4 - 99

* 1 element elastic : check the differences.



1) Integration point option
(make it same for all cases)
2) verification vs. validation ?
3) Thursday → meeting.

10/4/99

Oct - 5 - 99

· One element elastic, elasto-plastic test.
- Dyna, abaqus
- 1 point integration point
(effective & economical)

10/5/99

Oct - 7 - 99

1. Meeting
- One element & 125 element for ls-dyna cased.

2. 125 element done

10/7/99

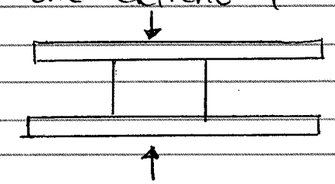
Oct - 13 - 99

· One element + base plate model (ls-dyna)
→ Need to be modified.

10/13/99

Oct - 14 - 99

* one element + double plate (ls-dyna)



· No flying but not good.
· plates deformed.

10/14/99

Oct - 15 - 99

* Meeting
- develope better case model

10/15/99

Oct - 18 - 99

· One element + plate model (lsdn, dn3d)
(material type & strength)

10/18/99

Oct - 19 - 99

· all model ls-dyna run
· all model dn3d set (ready to run)

10/19/99

Oct - 20 - 99

1. ls-dyna block model output check

2. dn3d model run

10/20/99

Oct-21-99
· ls-dyna impact beam model setup (b~g)
<Note> ls-dyna performed better than Dn3d.

10/21/99

Oct-22-99
· Dn3d impact beam model (b~g)

10/22/99

Oct-24-99
1. Meeting.

2. Striker model investigation
- velocity 10% reduce
- E & density 10% reduce } ⇒ reduce energy absorption.
wood $E = 2 \times 10^6$ psi
3. Review John's paper
- more realistic model plan

10/24/99

Oct-25-99
· Material option 20 : ls-dyna (full plastic option)
<Note> we need bunch of extra data
or need experimental study.

10/25/99

Oct-30-99
· ls-runs completed data report
· E check (beam striker)

10/30/99.

Oct-31-99
· Dn3d runs completed ⇒ data report.
· E model (beam and striker)

10/31/99

Nov-1-99
· Meeting
- Prepare report

11/1/99

Nov-2-99
· Report work continue
: graphs, discussions

11/2/99

Nov-5-99
· Meeting
- reviewing the report ⇒ result look good, but need little more investigation.

11/5/99

Nov-8-99
1. Meeting

2. E & D/F effect investigation.

11/8/99

Nov - 9 - 99

· Striker shape investigation
  - type 2, 3, 4

11/9/99

Nov - 14 - 99

1. Dn3d striker shape (Beamc)

2. Dn3d all other case (ep & e) running and data report.

11/14/99

Nov - 15 - 99

· ls-dyna all other cases (ep&e) running.
· Data report.

11/15/99

Nov - 16 - 99

· Impact striker type 2 modification
· Dn3d, ls dyna - beam c, beam d run.

11/16/99

Nov - 17 - 99

· Data arrangement for report

11/17/99

Nov - 18 - 99

· Meeting
  ⇒ Ready to ship!

11/18/99

Pages 255 Through 268 Are Intentionally
Left Blank

APPENDIX.

(Material Properties)

Information potentially subject to copyright protection was redacted from pages 270 through 283 of this scientific notebook. The redacted material is from the following reference:

ASTM International. "Standard Specification for Pressure Vessel Plates, Carbon Steel, for Moderate- and Lower-Temperature Service." West Conshohocken, Pennsylvania: ASTM International. A516/A516M. 1996

Information potentially subject to copyright protection was redacted from pages 284 through 286. The redacted material is from the following reference:

Bringas, J.E. "The Metals Red Book. Volume 2-Nonferrous Metals, The Metals Data Book Series." ISBN 0-9696428-1-4. Edmonton, Alberta, Canada: CASTI Publishing Inc. Year of publication unknown.

Element Managers are requested to put the following statement at the conclusion of "manual" Scientific Notebooks:

"I have reviewed this scientific notebook and find it in compliance with QAP-001. There is sufficient information regarding procedures used for conducting tests, acquiring and analyzing data so that another qualified individual could repeat the activity.

8/30/2000

MGPE

(Element Manager signature and date above line, Name of Element beneath line)"