

Question Asked 3 years ago



Qingqing Zhao

6.57 · University of Nottingham

How can I generate stress strain curve in Abaqus?

I would like to generate the stress strain curve for a cubic unit cell with many elements in Abaqus. Is it just outputting stress and strain of one element and combine them together? Thanks.

Finite Element Method

Finite Element Analysis

Abaqus

Share

5 Recommendations

Popular Answers (2)



Etienne Lebard

École des Mines d'Albi-Carmaux

3 years ago

Hello Qingqing,

For one given element, you can extract any strain and stress time curve.

In the "Tools" menu of the postprocessor, select "XY Data" then "Create". Select "ODB field output" or "ODB history output" depending on how you chose to stock the results you want to combine.

Assuming you chose "ODB field output", in the pop-up window first chose the "position" of the output variables you want to combine (integration point in general but there are other possibilities). Then click on the "Elements/Nodes" tab, then "Edit Selection": you can either chose to "pick from viewport", or enter "element labels", "element sets" or "internal sets".

Once you picked the element (or elements) you want to combine results on, go back to the variables tab and pick the ones you're interested in. Let's assume you're simulating a traction on a one element cube in the Y direction and you want to plot the normal-stress vs true normal-strain: pick S22 and LE22. Once you've done that, save these variables so ABAQUS keeps them in memory (for example, let's call S22 "normal stress" and LE22 "true normal strain").

Again, go in the "Tools" menu => "XY Data" => "Create" but this time chose "Operation on XY Data". In the "Operators" menu, look for and double click on "Combine(X,X)", then double click on the XY Data you want to use as input argument (LE22 so "true normal strain") and finally double click on the XY Data you want to use as output of your function (S22 so "normal stress").

Click on "Plot Expression" or "Save As...", and you have your stress-strain curve.

If you want to display the average stress in a mutli-element model vs the average strain, you can also create XY Datas containing stress and strain for each element, and then use the "avg((A,A,...))" operator in the "Operate on XY Data" menu, and finally combine the results. Though it can get pretty painstaking if you have a lot of elements, in which case I would recommend using a script.

36 Recommendations



Shyam Mohan Panamoottil

Rhondium Ltd

3 years ago

Hi Qingqing,

If you will be repeating this process for several models (with some differences between each), I would recommend putting this process into a script, which can then be run from the command line. It is relatively simple to create a script that loops over all elements, calculates average stresses and strains and then outputs them to a file.

5 Recommendations

Ad



Question followers (87)

See all



Similar Questions



How to get Force - displacement curve in Abaqus 6-11.1?

Someone explain to me some methods to get the Force-displacement curve from Abaqus for whole model.

6 answers added



How do I extract stress at some nodes using abaqus python script?

Hello, Can someone suggest me a way to extract S11 from abaqus .odb file, using python script? I do not have node set. Say, I want to find S11 of...

11 answers added



How can I solve the problem of excessive element distortion in 3D modeling of machining with ABAQUS?

I have a problem (excessive elements distortion) in 3D modeling of machining with abaqus explicit. I modeled the tool as a rigid body. I tied to...

27 answers added



How can I extract node coordinates in abaqus?

How can I extract node coordinates in abaqus?

9 answers added



How can I fix this error in Abaqus - '119 elements have missing property definitions?

Error in job newPartJob: 119 elements have missing property definitions. The elements have been identified in element set ErrElemMissingSection.

22 answers added



How can I fix this error in ABAQUS? Excessive distortion at a total of # integration points in solid (continuum) elements

Hi every one I have a model in Abaqus with large deformation. When I run this model, Abaqus exit with this error: "Excessive distortion at a



Etienne Lebard
École des Mines d'Albi-Carmaux

3 years ago

Hello Qingqing,

For one given element, you can extract any strain and stress time curve.

In the "Tools" menu of the postprocessor, select "XY Data" then "Create". Select "ODB field output" or "ODB history output" depending on how you chose to stock the results you want to combine.

Assuming you chose "ODB field output", in the pop-up window first chose the "position" of the output variables you want to combine (integration point in general but there are other possibilities). Then click on the "Elements/Nodes" tab, then "Edit Selection": you can either chose to "pick from viewport", or enter "element labels", "element sets" or "internal sets".

Once you picked the element (or elements) you want to combine results on, go back to the variables tab and pick the ones you're interested in. Let's assume you're simulating a traction on a one element cube in the Y direction and you want to plot the normal-stress vs true normal-strain: pick S22 and LE22. Once you've done that, save these variables so ABAQUS keeps them in memory (for example, let's call S22 "normal stress" and LE22 "true normal strain").

Got a question you need answered quickly?

Again, go in the "Tools" menu => "XY Data" => "Create" but this time chose "Operation on XY Data". In the "Operators" menu, look for and double click on "(X,Y)", then double click on the XY data you want to use as input (LE22 so "true normal strain") and finally double click on the XY Data you use as output of your function (S22 so "normal stress").

Click on "Plot Expression" or "Save As...", and you have your stress-strain curve.

If you want to display the average stress in a multi-element model vs the average strain, you can also create XY Datas containing stress and strain for each element, and then use the "avg((A,A,...))" operator in the "Operate on XY Data" menu, and finally combine the results. Though it can get pretty painstaking if you have a lot of elements, in which case I would recommend using a script.

36 Recommendations



J. A. Beltrán-Fernández
Instituto Politécnico Nacional

3 years ago

In accordance with Mr. Lebard, to plot stress_strain, use XY data => Field output=> choose the stress=> the single element or the hole model (from elements/nodes)=> save

after do the same thing for the strain

After come back to XY data=> operate on XY data => choose "combine" from the the available operations and introduce your stress and strain => plot

I hope to help you

John

2 Recommendations



Qingqing Zhao
University of Nottingham

3 years ago

Thanks for your responses! I really appreciate it.

Qingqing



Shyam Mohan Panamoottil
Rhodium Ltd

3 years ago

Hi Qingqing,

If you will be repeating this process for several models (with some differences between each), I would recommend putting this process into a script, which can then be run from the command line. It is relatively simple to create a script that loops over all elements, calculates average stresses and strains and then outputs them to a file.



Stress strain graph plotting for specific area in Abaqus?

I want to plot graph of stress vs strain, stain vs time for specific area of my geometry which I have attached. I can plot stress strain by...

9 answers added



how can I import or use .inp files in abaqus to view the results?

In a special analysis I need to use an .inp file for abaqus documentation.

18 answers added



Why is this error in abaqus XFEM method: too many attempts for this increment?

I simulate crack grows for a specimen in abaqus and use XFEM method. When my crack initiates and grows a bit and before full break the specimen I...

32 answers added

Sign up today to join our community of over
14+ million scientific professionals.

Reads	10318
Folk	87
Answers	14

Related Publications

Ergebnisse

[\[Show abstract\]](#)

Chapter · Jan 1990 · [Aircraft Design](#)

Arndt Hettig

[Read](#)

Second-developed program about crack growth simulation and analysis based on Abaqus

Article · Jan 2009

Y. Ma X. Xu J. Ning

[Read](#)

Model of elasticviscoplastic material with defects for Abaqus / Explicit (explicit scheme of integration)

[\[Show abstract\]](#)

File · Code · Sep 2011 · [Aircraft Design](#)

Yuriy Bayandin Oleg Naimark

[View research](#)

2. In the case of titanium sheet and plate, the elastic modulus can vary from 16 - 18*10^6 psi based on rolling method (actually, the texture.)

3. When performing analyses involving plasticity, it is proper to use the "true" stress strain curve rather than its common counterpart - the "engineering" stress strain curve.

4. Many papers available if you want to investigate.



Van-Tho HOANG
University of Ulsan

3 years ago

According to Qingqing, I would like to have another question that what node or element should I choose to draw stress-strain curve?

3 Recommendations



Sagar Shah
University of Massachusetts Amherst

2 years ago

I am new to ABAQUS and I am facing the same problem of obtaining average strains. As Mr. Shyam said scripting is the best way to go for large number of elements, can you help me with the code because I have relatively less experience scripting for ABAQUS?



Etienne Lebard
École des Mines d'Albi-Carmaux

2 years ago

Hi Sagar,

There is a feature in ABAQUS (at least v6.12 and the following ones) that records the actions you're doing (like clicking, selecting an element, etc.) and puts them in a script (in Python). I think it is called "Macro Manager". You can edit what you obtain, test it until you have a functional script (which gets you the results you want).

3 Recommendations



Mehdi Lotfi
Islamic Azad University Central Tehran Branch

a year ago

hi

guys could you help me how can I analyze the micropile's behavior ?

specially in Abaqus.

I don't know which nodes I should select.....



Abbas AbdulMajeed Allawi
University of Baghdad

a year ago

I agree with Etienne Lebard.

1 Recommendation



Amjad Albayati
University of Baghdad

a year ago

really valuable answer Etienne



Paramveer Sharma
Indian Institute of Technology Madras

5 months ago

What if I want to calculate the homogenized stress and strain plot for which the formula is $\text{STRESS}(i) \cdot \text{EVOL}(i) / \text{Sum}(\text{EVOL})$



Jide Soyemi Soyemi O.B
The Federal Polytechnic, Ilaro

3 months ago

@Lebard gave a well explained answer to the question



Can you help by adding an answer?

Answer

Enter your answer

