Error face midpoint problem

SPECFEM

Dec 2018

3 / 7
Mar 2019

Jun 2020

ljhwang Speaker Series — Dec '18

Dear all,

I'm trying to run a model with topology and CPML in SPECFEM3D_Cartesian.But some problems came to me when I was run database generation. The error is as follow:

error element face midpoint distance : 6.640625E-02 594.957275

error corner: 1 xyz 523625.188 3602220 1297.69434

error corner: 2 xyz 523644.656 3602220.25 1283.00208

error corner: 3 xyz 523637 3602199.50 1275.03577

error corner: 4 xyz 523617 3602199.25 1288.56860

xcoord : 523644.656 523625.188 523617.562 523637.438

ycoord : 3602220.25 3602220 3602199.25 3602199.50

zcoord : 1283.00208 1297.69434 1288.56860 1275.03577

STOP error element face midpoint.

This problem has been bothering me for a long time. Maybe the edge curve is too fluctuating? Could you please help me to figure out how to solve this problem?

I would really appreciate your help,

Thanks,

Zixuan Wan I Chengdu university of technology

created Dec '18 | last reply Jun '20 | 6 replies | 412 views | 4 users | 2 likes

2 MONTHS LATER

pokayoke — Feb '19

I also have the same problem
when I mesh a very small scale mesh model,
for example 4x4km 192x192 maybe ok
but 256x256 I will receive the error like this
Does anyone know what's this error mean?

I will be grateful for any help you can provide.

21 DAYS LATER

istoneuw — Mar '19

Hello,

I am not an expert in the code, but have run into similar issues in the past (i.e., similar error invoking the element face midpoint distance). How I ultimately was able to get around the problem was by changing the TOL_DIST variable in specfem3d/src/shared/get_element_face.f90. This value is typically set to 1.e-4, which I found was too small for the higher resolution topography meshes I was building. Changing this value to 1.e-3 fixed the error and (seemingly) did not cause any instabilities in the code or my outputs.

So I would suggest trying to increase your TOL_DIST value, and seeing if that avoids the error.

Ian

1 Reply — 2

kaoutar

Hello,
I just started using specfem3d, and I want to simulate wave propagation on a homogenous medium with 5 sides PML and a free surface at the top of the model.
But I am facing a problem with it : when I use the codes "xadd_CPML_layers_to_an_existing_mesh" and "xconvert_external_layers_of_a_given_mesh_to_CPML_layers" it generates a "free_or_absorbing_surface_file_zmax" which contains only "0" value.
What I did, is replace it with the "free_or_absorbing_surface_file_zmax" that I had before using the CPML codes.
But when I ran the xspecfem3d code, I have the error :
error element face midpoint distance: 3.4609373E+10 - edge length: 5.4250000E+10 tolerance: 5.4250004E+07
I tried to change the TOL_DIST variable like you said, but it didn't work for me.
Do you have an idea on how I can fix this problem ?

Thank you
Kaoutar

6 MONTHS LATER

pokayoke — Sep '19

I really appreciate your answer.
This solution works for me.

9 MONTHS LATER

kaoutar — istoneuw — Jun '20

Hello,
I just started using specfem3d, and I want to simulate wave propagation on a homogenous medium with 5 sides PML and a free surface at the top of the model.
But I am facing a problem with it : when I use the codes "xadd_CPML_layers_to_an_existing_mesh" and "xconvert_external_layers_of_a_given_mesh_to_CPML_layers" it generates a "free_or_absorbing_surface_file_zmax" which contains only "0" value.
What I did, is replace it with the "free_or_absorbing_surface_file_zmax" that I had before using the CPML codes.
But when I ran the xspecfem3d code, I have the error :
error element face midpoint distance: 3.4609373E+10 - edge length: 5.4250000E+10 tolerance: 5.4250004E+07
I tried to change the TOL_DIST variable like you said, but it didn't work for me.
Do you have an idea on how I can fix this problem ?

Thank you
Kaoutar

istoneuw — Jun '20

Hello Kaoutar,

Unfortunately, I have never been able to successfully run Specfem using PML boundary conditions, so I cannot directly address your issue. One important thing, though: I do not think the free_or_absorbing_surface_file_zmax would be the same between the CPML and original version of your mesh. The node and element ID's do not stay the same between the two meshes, so the values listed in the free surface file for one will be different from the values listed in the other. I think your best bet would be to try and figure out why "xadd_CPML_layers_to_an_existing_mesh" and "xconvert_external_layers_of_a_given_mesh_to_CPML_layers" are not producing a proper free surface file. Or, if your mesh is not too complicated and does not have topography, you may be able to write a script yourself to find the ID's of nodes and elements at the top surface of the PML mesh and writes a free surface file.

Sorry I can't be of more help!

Ian

18 DAYS LATER

kaoutar — Jun '20

Dear Ian,

Finally the cpml worked for me.
It actually works when I enter the pml thickness manually, but not when I use the code "add_CPML_layers_to_an_existing_mesh".
In another subject, do you know how to implement a pressure source in an acoustic or elastic domain using SPECFEM3D ?
I have read the manual, but I only find FORCESOLUTION or CMTSOLUTION sources.

Thank you in advance , and have a nice day

Kaoutar

Reply

Suggested Topics

Topic	Replies	Views	Activity
Specfem3d_Cartesian AMD HIP Port Merge — SPECFEM new-feature	4	80	Nov '20
Specfem_Globe & Fully Anisotropic Models — SPECFEM seismology specfem3d	0	61	Mar '20
SPECFEM MPI Installation — SPECFEM installation specfem3d	4	105	Jun '20
Specfem2d: S-wave pattern of explosion source (Dirac source time function)? — SPECFEM specfem2d	3	86	Jun '20
How does GMSH adopt PML boundary absorption — SPECFEM specfem2d	0	40	Aug '20

Want to read more? Browse other topics in SPECFEM **or** view latest topics.