Tutorial to CAE of Sensors and Actuators

Michael Nierla

October 21, 2014

1 Working at the chair

The preparation of the assignemt tasks can only be done on the computers in the LSE computer lab Room 02.029 (unless you have valid license for Ansys, NACS and Gid). *Computers in our lab:*

- *lse81.e-technik.uni-erlangen.de*

- *lse82.e-technik.uni-erlangen.de*

- *...*

- *lse88.e-technik.uni-erlangen.de*

To get access to those computers you have to sign a "Request for user authorization". You will then receive one of the user accounts *cae01* through *caeXX*. The password for those accounts has to be set together with our adiminstrator (Daniel Babel, Room 02.032). Use only those accounts to log to the computers above.

2 Working from home

To work from home you need

- the (free) `x2go`-client
  → http://wiki.x2go.org/doku.php

- an user account at LSE

- an user account at RRZE (the one you need to login into studon)

- the (free) `AnyConnect` vpn-client from RRZE
  → https://www.rrze.fau.de/dienste/internet-zugang/vpn/cisco-vpn.shtml

The RRZE vpn-client is required as the student computers can **only** be reached from inside the FAU network. To setup `AnyConnect` follow the instructions given in `Anyconnect_Windows.pdf`, `Anyconnect_Linux.pdf` or `Anyconnect_MacOS.pdf`. After installation use the `FAU-Fulltunnel` option to login with your **RRZE user account**.
With `x2go`-client you can directly build up a graphical connection to the machines lse81 - lse88. To setup the `x2go`-client do the following:

1. download and install the `x2go`-client

2. start `x2go`-client and select `Session -> New Session ...`

3. set `Host` to one of

   - lse81.e-technik.uni-erlangen.de
   - lse82.e-technik.uni-erlangen.de
   - ...
   - lse88.e-technik.uni-erlangen.de

4. set `Login` to your **LSE user account** (i.e. *caeXX*)

5. select `Session Type -> XFCE`

Hint 1: It may be useful to create a session for each of the computers lse81 - lse88 to avoid that all groups work on the same machine at the same time.

Hint 2: To check if a computer is already used by another group, connect to the computer, open a new terminal and type `htop`. You will then get a graphical overview over the computer usage (can be closed with F10).

Important note: Inside the NACS gui some sections (e.g. the section `Results`) are very large. Therefore it might be possible, that some buttons are not visible as they are out of the screen. If you encounter this problem you can try to win some space by grabbing the icon bar between menu bar and `Analysis` tab and move it asside. Another option is to change the resolution of your screen.

3 Software

The assignments preparation has in general three steps and each step requires different software tools:

1. Preprocessing/Meshing $\rightarrow$ Ansys

2. Simulation $\rightarrow$ NACS

3. Postprocessing $\rightarrow$ Gid and Gnuplot, Excel, etc.

4 Documentation

Documentations for NACS, Gid, Ansys and Gnuplot are provided via Studon. **Visit StudOn frequently to check for any updates of the script or assignment sheets! We will also send an Email, if the sheets are updated.**

5 Handing in your results for the assignments

Each assignment requires you to hand in some results due to a certain date. This due date can be found in the header of each assignment sheet. In the general case the results which have to be handed in are described explicitely (e.g. hand in a screenshot of ...). Additionally you have to provide a file called `results.txt` where you shall document what you did and in some cases also why you did it (e.g. why do we need this kind of boundary condition). Most of the points will be rewarded for this `results.txt`, so make sure that it

- gives a comprehensible overview of what you did

- answers all questions asked in the assignment tasks

- is written such that it can easily be understood (no sentences over 834 lines!)

All files which shall be graded have to be place to the corresponding assignment folder `/home/userHome/stud/CAESAR/group<Group#>/assignment<Assign#>/`
until the given due date. Later changes are not possible and will not be considered in the grading process.
Hint: It is advisable to hand in not only the required files but also your Ansys mesh script and your python script (.py) which will be created automatically if you simulate with NACS. In case that you have wrong results we might find the error in the mesh or in the simulation and maybe it was not your fault.

6 Handing in your results for the projects

In general there is no big difference between handing in your results for the projects and your results for the assignments. The two differences are:

1. Put your results to `/home/userHome/stud/CAESAR/projects/project<Project#>/`

2. You **have to** hand in all your mesh and simulation scripts as well as your presentation in addition to the otherwise required files.

7   Get help

If you have any problems regarding hardware, software or one of the assignments please check the FOP (frequently occuring problems) section first. Most software problems can be solved by restarting the software and most simulation problems can be solved by checking your scripts step by step (especially Ansys mesh scripts). If you nevertheless have problems or find bugs in the software write an email to

- daniel.babel@fau.de ($\rightarrow$ for technical problems)

- michael.nierla@fau.de ($\rightarrow$ for questions regarding assignment tasks or theory)

Note 1: We cannot answer all questions and solve all problems instantaneous. So start early with your assignments and come to the practical tutorials if you have problems.
Note 2: The practical tutorials are not meant to be the place to prepare your assignments. Neither the number of computers nor the time available in the practical tutorials is sufficient to solve the assignments. Those tutorials are only meant to be question hours.

8   Suggested way for preparing a simulation model

- Think about the results you want to achieve in advance (material parameters, wavelengths, velocities, influence of defects, ...)!

- Draw a sketch of your model on a sheet of paper!

- Are there any symmetries in the object? Use axes/planes of symmetry!

- NUMBER THE KEYPOINTS ON YOUR SHEET OF PAPER!

- Make the first try as simple as possible (ONLY FOR MODELING !):

    – Neglect the maximum element size for wave propagation
    – Reduce the necessary boundary distances

- Color your model!

- Think about the type of analysis needed and the necessary parameters (number of time steps, time increment per step, frequencies, ...)

- Define the correct boundary conditions:

    - Which conditions do I need?
    - Which conditions are applied automatically?
    - What do they mean for the current physical field?

- Define the result you want to see:

    - Am I interested in just one or a few special points?
    - Where should these points be positioned?
    - Do I want to see the complete results of my simulated area?
    - What time steps am I interested in?

- For the first tries, simulate just as many time steps as absolutely necessary!

- AND FINALLY: Compare the simulated results with expected results:

    - Are the simulation results physically possible?
    - Does the order of magnitude make sense?
    - Compare the results using an analytical solution of a simplified problem.

9 Frequently occuring problems

9.1 Meshing, Ansys Problems

- Hanging nodes → collapse model with `AGLUE` or similar

- Not connected areas → collapse model with `AGLUE` or similar

- Regions or groups not written to file (not visible after loading to NACS)
  → make sure to use `REGION` and `GROUP` command before using `WRITEMODEL`

- Ansys reports errors including `...NACS...`
  → make sure that a file called `start145.ans` is in your home folder

- Ansys reports errors regarding internal functions (e.g. `LSEL` or `LESIZE`)
  → check `ansysCommandReference.pdf` if you used the command correctly

9.2   Analysis, NACS Problems

- Results are not written out → check if you defined output in NACS correctly, i.e.

  1. the desired quantity was selected under
     `Results -> Set Result`
  2. gid output was enabled under
     `Results -> Result Format`
  3. text output was enabled under
     `Results -> Set Result -> Monitor Results`

- I cannot store results as the `Apply` button is outside the screen → try to win some space by grabbing the icon bar between menu bar and `Analysis` tab and move it aside. Another option is to change the resolution of your screen.

- Results look completely wrong or the simulation fails → check if

  1. you set the right boundary conditions under
     `Boundary Conditions -> Set Condition`
  2. the boundary is too close to the rest of the geometry
  3. you need to include additional regions (e.g. for **stray fields**)
  4. you assigned the right materials under `Material` and that the material contain all needed parameter for the simulation (e.g. density for mechanics and acoustics)
  5. your spatial discretization (meshsize) and temporal discretization (timestep) are fine enough → perform parameter study
  6. you specified enough timesteps so that the desired effect can take place

9.3   General, technical problems

- No files are written / I cannot save files anymore
  → check if your quota is exceeded by typing `quota` in a linux terminal

- My quota is exceeded by I need more space
  → clean up your old files, delete not needed .gid outputs (can get very large) or move files to storeBack or storeNoBack

10 Performing a parameter study for finding a fitting meshsize and timestep

As the finite element method is based on a spatial and temporal discretization of space and time and the computer resources are limited you will face the problem of choosing your meshsize and timestep (only for transient simulations) such that:

- the resulting discretization error can be neglected

- the problem gets not too expensive to calculate

A good way to satisfy both aspects is to perform a parameter study on meshsize and timestep. To perform a parameter study on the **meshsize** do the following steps:

- include one or more observer points to your model (not too close to any sources but also not too close to the boundary)

- create a very coarse mesh (finest structure modelled by only a hand full of elements) and a very fine mesh (finest structure resolved by very many elements)

- create three to five additional meshes with meshsizes between the coarest and the finest meshsize

- perform on each of these meshes the same simulation and write out results at the observer points

- plot the results at the observer points over the meshsize (e.g. with gnuplot)

- does the curve converge, i.e. do the results approach a certain value if the mesh gets smaller?
    - yes → take the coarsest meshsize for which convergence is achieved
    - no → create even finer meshes and repeat the meshstudy
    - no although mesh is very, very fine → check if your observer point is located at a point of singularity (e.g. a reentrant corner)
        * yes → take other observer point
        * no → check your simulation for errors

To perform a parameter study on the **timestep** do the following steps:

- include one or more observer points to your model (not too close to any sources but also not too close to the boundary)

- choose a very coarse timestep (e.g. five timesteps per period length for a sinusoidal excitation) and a very fine timestep (e.g. 50 timesteps per period length for a sinusoidal excitation)

- select three to five additional timesteps between the coarest and the finest timestep

- perform for each timestep the same simulation and write out results at the observer points (make sure that you adapt the total number of timesteps such that all simulations take the same time!)

- plot the results at the observer points over the timestep (e.g. with gnuplot)

- does the curve converge, i.e. do the results approach a certain value if the mesh gets smaller?
  - yes $\rightarrow$ take the coarsest timestep for which convergence is achieved
  - no $\rightarrow$ create even finer timestep and repeat the timestepstudy
  - no although timestep is very, very fine $\rightarrow$ check if your observer point is located at a point of singularity (e.g. a reentrant corner)
    * yes $\rightarrow$ take other observer point
    * no $\rightarrow$ check your simulation for errors