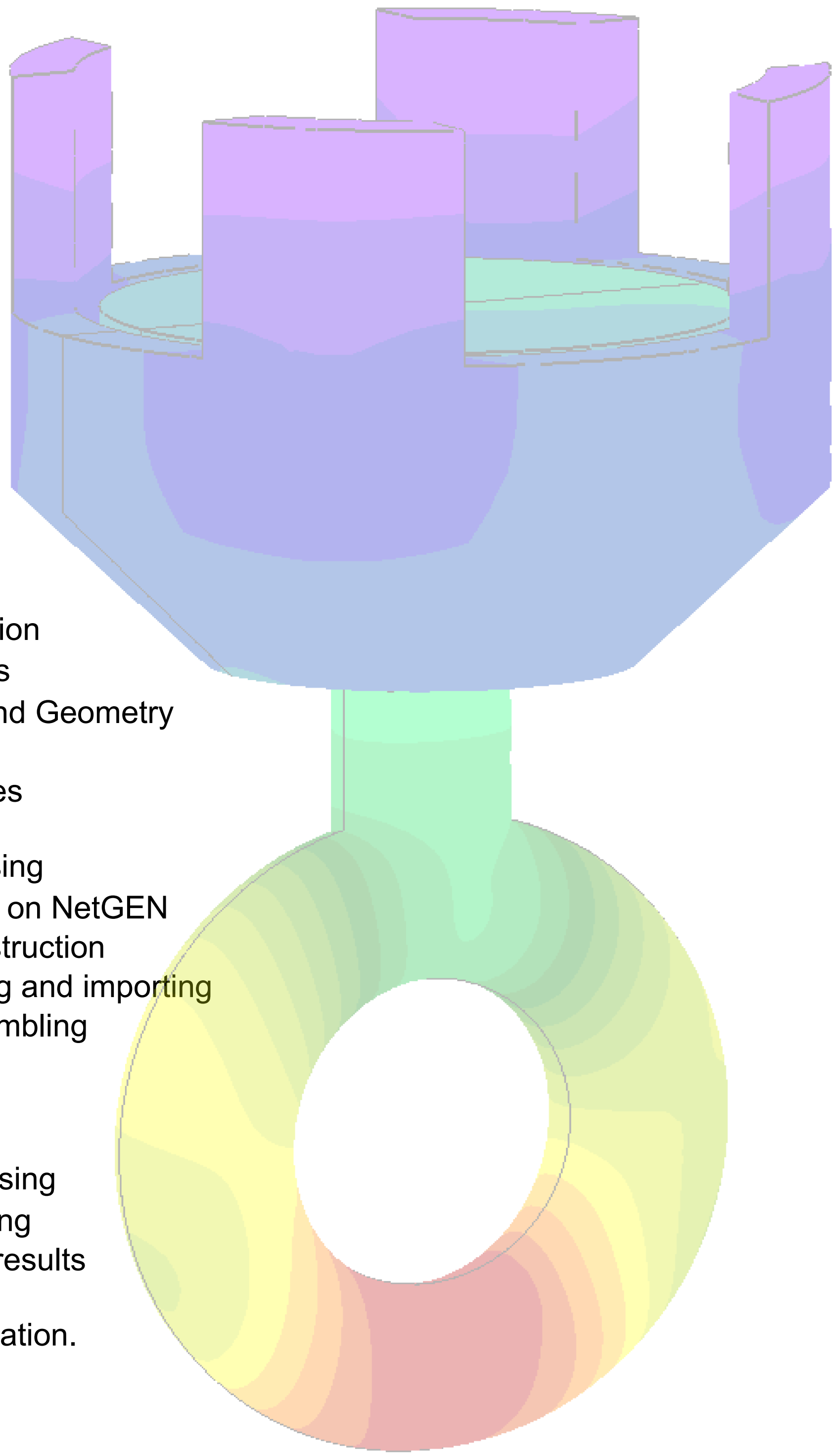
Advanced CalculiX Tutorial

Eng. Sebastian Rodriguez

www.libremechanics.com



Content

Introduction

Usual FEA applications involve analysis where multi body assemblies are loaded to determine the contact behavior between each other, some of this data are friction, relative displacement and contact pressure. This kind of cases of study are specially complex and requires much more technological resources due to the nonlinearity characteristics of the case, where a slight change on the geometry, mesh or frontier conditions overcome on a totally different behavior.

CalculiX offers a highly custom process to control contact parameters such as, contact pressure, over closure behavior, clearance superposition and penetration; this characterized Calculix as preferred over other FEA/FEM applications on the development of especial cases where common mechanical assumptions are not sustainable such as:

- Special material coupling
- Multi phase contact
- Introduction of soft and moisturized surfaces

This technical advantage and the capacity to allow important changes on the solution central process involve the risk to inaccurate results due the lack of experience in the application of advanced parameters.

For this precise tutorial it will be assumed that the user its already familiar with the working environment of CalculiX CCX and CGX modules, if this is not the case it is recommendable to refer first to some beginner tutorial for example:

- Getting Started with CalculiX by Jeff Baylor
- Short Tutorial For Using CalculiX GraphiX (cgx) As Preprocessor by Guido Dhondt
- How To Install CalculiX 2.4 multi-thread under Ubuntu 11.04 and later. by Libre Mechanics

This tutorial is intended to be a simple and easy way to introduce the user to the multi body contact handling on CalculiX, please notice that some scientific and technical data may not be a representation of any real life case; further contact theorical explication will be omitted in order to maintain the simplicity of this document.

As the user is probably aware by now, the document make a number of simplifying assumptions as the tutorial progressed, this is done in the interest of gaining a clearer understanding of these fundamental without getting bogged down in special details and exceptions. By no means it hast the complete history of contact handling on CalculiX, it is much broader in scope that can be presented in a single document such as this, but it is sincerely hoped that this tutorial will enable one to do a better job on the definition, solution and study of this kind of analysis.

Command conventions:

- ***CGX commands***
- ***Console commands***

CHAPTER ONE

Case definition

The designed case for this tutorial present an assembly of a rotatory hook on a top fixed base which is loaded with a constant force, the contact area is form by the two bodies on a uniform conic area, the concentricity of the faces ensure the mechanical connection between the two bodies, the eccentricity on the second hook load, that is not usually observed on real life designs, its intended to create an unbalanced torque on the contact interface which may lead to some slides and displacement on the assembly.

The goal of this hypothetical case is to determine the pressure contact between the bodies and the distribution of loads, besides the displacement and stress effects of a common static-plastic analysis. To do this a 3D model of the hook its already produced and will be used for this tutorial.

Sizing and Geometry

The hook model can be easily be downloaded from the web for this tutorial, the model is saved as IGES which is the Initial Graphics Exchange Specification format a common type of file for any CAD application. ***See chapter six*** . The geometry has special face division to define the loads on the Y and Z axis which later will be a really aid on the set construction.

If the user chooses to create the model by its own the principal dimensions (in mm) can be inferred by the user, for the assembly, it should be no problem on doing this while some rules are respected:

- No assembly relations gaps.
- No over closure.
- Create the same face divisions for loads.

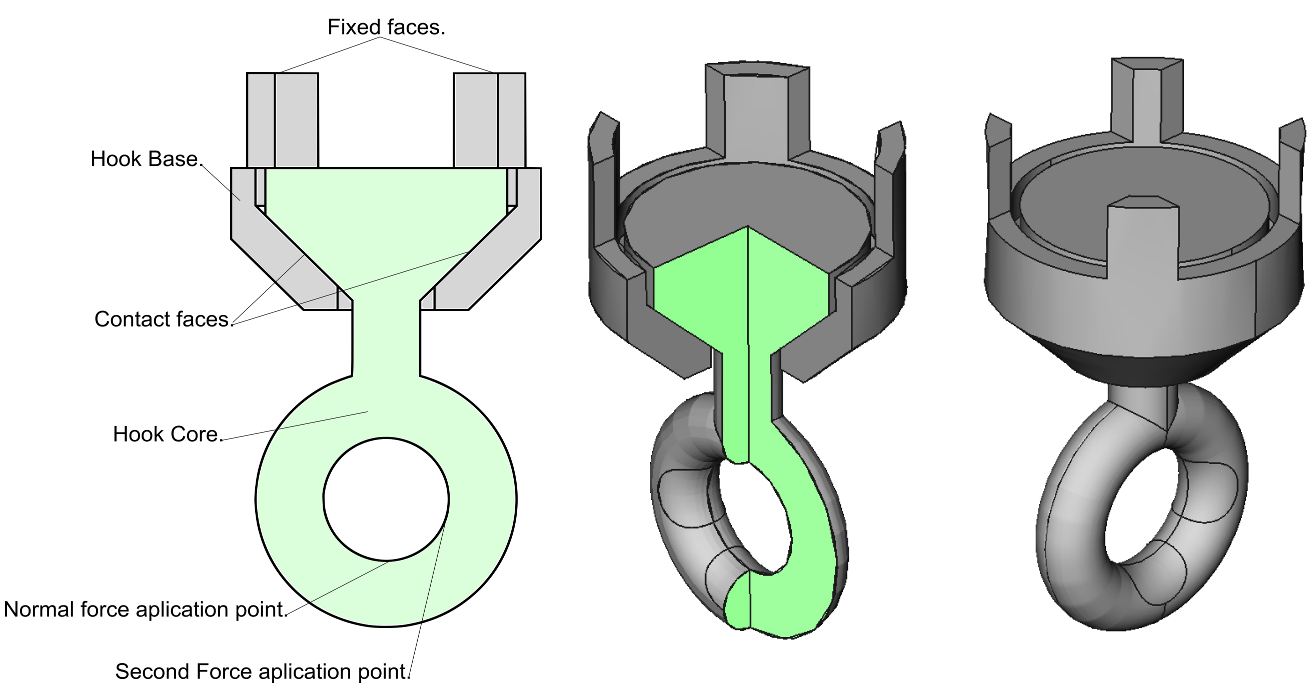


Image 1: Hook geometry definition

- Save the two bodies in a two different files but make sure the share the same coordinate system, pre assembly its necessary so when they are put together again they will be in the same position as the the CAD program.

The set faces are created by projecting a line or sketch over a surface and cutting it to create an independent dace, so the loads will be applied on a face not a point.

Units

CalculiX as others CAE programs relies the responsibility of defining the correct unit system on the user and its ability to interpret results and make changes and conversions to optimize the problem. This means that CalculiX will not ask for any physical magnitude and will not indicate any of these in the results; but making clear that this do not means that the results are dimensionless, it just means that the user is the one who distinguishes the units.

The equations used to solve the system do not have anything to do with the units the user is pretending to use, this become useful for simplify the model process but also become a risk for mistakes on the units selections which will lead to huge error on reading results, as some people call Garbage-in garbage-out case.

A common set of units combinations is presented on Chart (1) for the SI system, where the colored one will be the units arrangement choose for this tutorial.

As well as selecting the input format for loads and properties, also, the output results data is inferred. For example, lets define the units for a common simple equation and then determined the solution format; for the equation velocity = length / time, the input

Length	Time	Mass	Force	Pressure	Velocity	Density	Energy	Gravity
m	s	Kg	Kg m/s2	N/m2	m/s	Kg/m3	Kgm2/s2	9.81
m	s	Kg	N	Pa	m/s	m Kg/l	J	9.81
m	s	g	mN	mPa	m/s	micro Kg/l	mJ	9.81
m	s	Mg (ton)	KN	KPa	m/s	Kg/l	KJ	9.81
m	ms	Kg	MN	MPa	Km/s	m Kg/l	MJ	9.81e-6
m	ms	g	KN	KPa	Km/s	micro Kg/l	KJ	9.81e-6
m	ms	Mg (ton)	GN	GPa	Km/s	Kg/l	GJ	9.81e-6
mm	s	Kg	mN	KPa	mm/s	M Kg/l	micro J	9.81e+3
mm	s	g	micro N	Pa	mm/s	g/mm3	nJ	9.81e+3
mm	s	Mg (ton)	N	MPa	mm/s	Mg/mm3	mJ	9.81e+3
mm	ms	Kg	KN	GPa	m/s	M Kg/l	J	9.81e-3
mm	ms	g	N	MPa	m/s	K Kg/l	mJ	9.81e-3
mm	ms	Mg (ton)	MN	TPa	m/s	G Kg/l	KJ	9.81e-3
cm	ms	g	daN	10^5 Pa (bar)	dam/s	Kg/l	dJ	9.81e-4
cm	ms	Kg	10^4 N (KdaN)	10^8 Pa (Kbar)	dam/s	K Kg/l	hJ	9.81e-4
cm	ms	Mg (ton)	10^7 N(MdaN)	10^11 Pa (Mbar)	dam/s	M Kg/l	10^5 J	9.81e-4

Chart 1: Common unites arrangement, Table taken from Impact Finite Element Program documentation http://impact.sourceforge.net/index_us.html

formats are length=mm , time=s, as it is wanted to determine an speed we can infer without solving the equation that the result will be presented in mm/s as show in the table, it will be an error to read the result on m/s , ft/min or km/h without a conversion. This example may seem very basic but represents the same principle for all equations on the solving process.

Properties

Distinguishing multiple bodies from each other on an assembly on CalculiX also allows to apply different properties of materials and loads, most real designs uses groups of pieces from different materials to increase resistance or to reduce costs and weights, commonly this inherent characteristics of the material are determined by real live experimentation, measure and prediction.

MATERIALS	ASTM A36 Steel	Copper, Cu; Cold-Worked
Density (g/cc)	7.85	7.94 8.93 8.96
Hardness, Brinell	--	89
Hardness, Rockwell A	--	35
Hardness, Rockwell B	--	51
Hardness, Vickers	--	100
Tensile Strength, Ultimate (MPa)	400 - 550	--
Tensile Strength, Yield (MPa)	250	--
Elongation at Break (%)	20.0 21.0	--
Modulus of Elasticity (GPa)	200	110
Compressive Yield Strength (MPa)	152	--
Bulk Modulus (GPa)	140	140
Poissons Ratio	260	350

Chart 2: Material for the hook assembly.

For this matter a combination of copper and A36 steel will be used to represent a composite group of pieces, the respective properties are listed bellow and were taken from MatWeb page.

CHAPTER TWO

Pre-processing

The meshing process will be carried out by NetGen, which is a magnificent 2D and 3D tetrahedral meshing program and its completely compatible with CGX, NetGen interface its very intuitive and can be easily become an external tool for CalculiX studies.

In this tutorial the two bodies will be mesh it on different times and files which will allow to overview the re assembly process.

1. Import the geometry

2. Heal the geometry: Usually, depending which CAD program the user choose to create the geometry some errors regarding the face orientation are presented which causes troubles on the 3D meshing, NetGen can fix this drawback by reorienting the faces of the model in order to create a single positive volume enclosure per body.

3. Mesh: The mesh density will be set to very fine in order to avoid some roughness issues in the contact faces.

4. Rename the Set faces: The faces names given early on the document will be different in this step because NETGEN boundary selection does not allow to rename faces on string names, the faces will be giving a known integer value for forward differentiation for example the fix face will be called the face 9999. Please noted that a good name will be some high number in contrast to the maximum normal number of faces of the model. By clicking on rename the face will be know stores as the face 9999 and will be easily identify on CGX , doing this for the rest of the faces on the both bodies will result on the definition above:

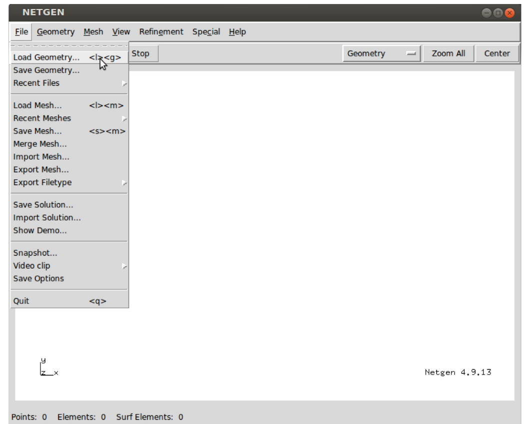
5. Exporting the mesh: The mesh now containing the set faces will be saved on the default .VOL file type extension, even so NetGen allows to save the model on .msh ABAQUS type file it does not saved the faces with it, so it will be necessary to later convert the .VOL file on CGX extracting the set definition.

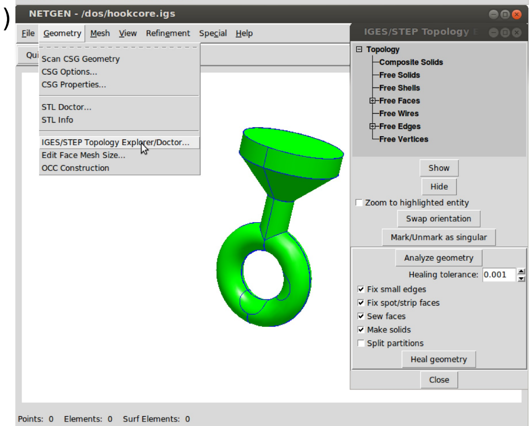
6. The .Vol file will be opened on CGX by the command:

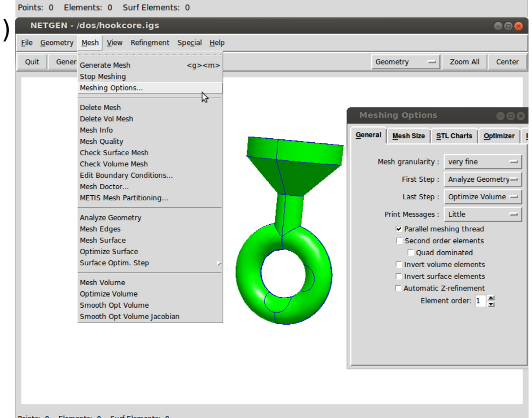
   - **CGX –ng meshfile.vol**

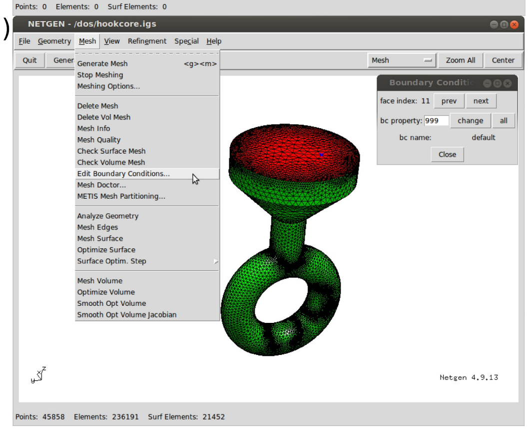
   To open the NetGen mesh on CGX

   Changing CGX parameter by the one set on the user system to invocate CGX program.
   The CGX window interface will appear whit the meshed model on it, for make sure the sets definition are also loaded the hole names will be revised by:

1)


2)


3)


4)


- **prnt se**
  To print all the sets names

This command will print on the console a rich list for all the set names including the amount of nodes, elements and faces of the set. Identify all the sets created on NetGen 9999, 8888, 777 and so on.

7. Importing data to CCX: The next procedure may seem a little over rate to this specific case but I will be give an idea of how to export

- **Send all abq**
  To export the mesh
- **Plot e 999**
  To plot the elements on the set set 999
- **Qadd hookbase**
  To create a new set named hookbase, then selected manually on CGX screen.
- **Send hookebase abq nam**
  To export the set hookbase on abaqus filetype containg just the element names.
- **Plot n set 888**
  To plot the nodes of the set 888
- **Qadd fix**
  To create a new set named fix, then select them manually on CGX screen.
- **Send fix abq nam**
  To export the set fix on abaqus filetype containing just the node names
- **Plot f set777**
  To plot the faces of the external elements of the set 777
- **Quadd master**
  To create a new set named master, then select them manually on CGX screen.
- **Send masterface abq sur**
  To export the set master on abaqus filetype containing just the faces names

This will create 4 different files, all.msh containing the whole mesh and 3 other files whit the names of the element, faces and nodes of the renamed sets, this files will be created automatically on the working folder taking the names of the sets.

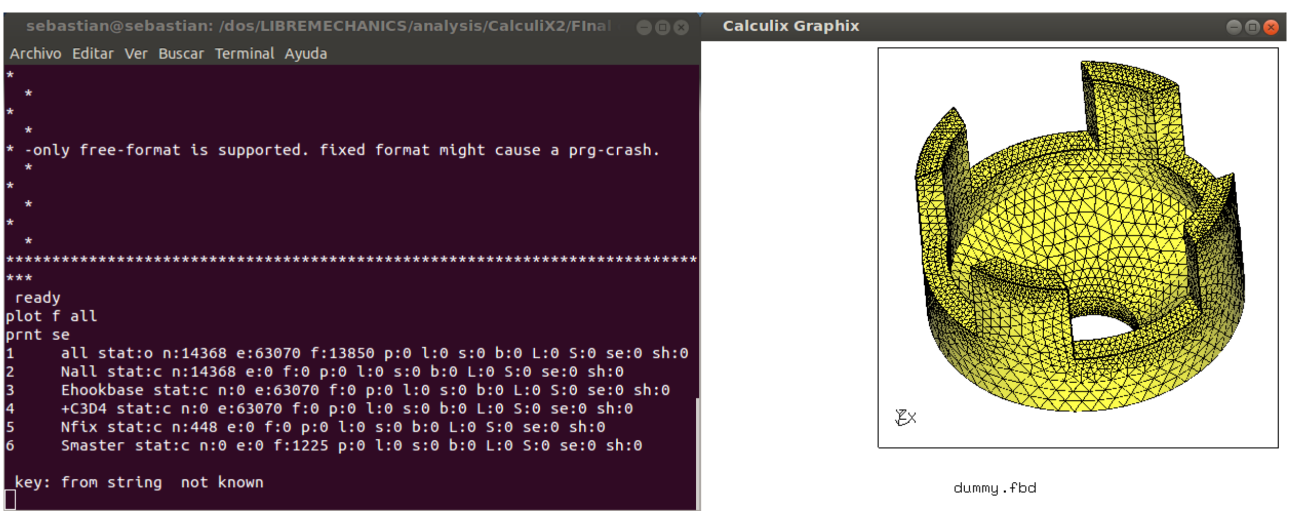
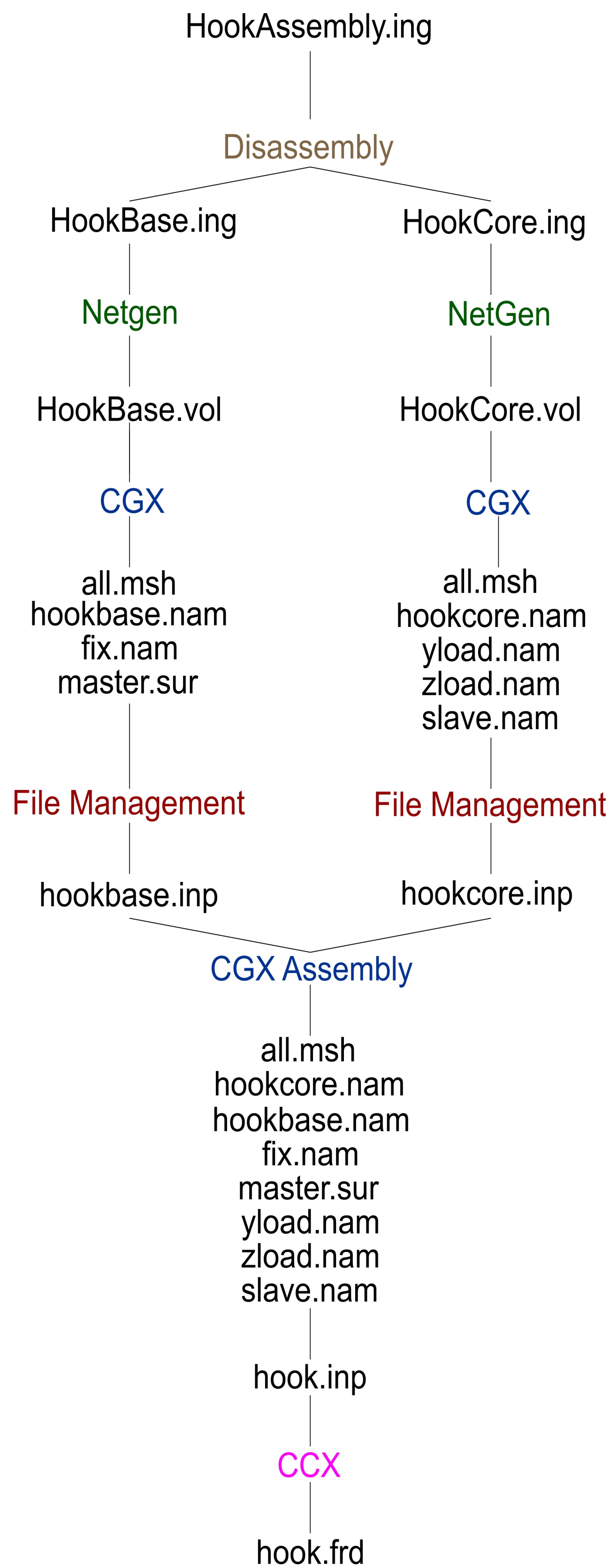


Image 2: prnt se list of the hook base part.

> *“The reason why there is not recommendable to load the parts with its set files separately its because the set files are related to the consecutive number of the nodes and elements on the mesh, when the re assembly its done, the meshes are combine but as all they nodes and elements naming begin at 1 there most be some renaming to correctly unite them in one file, this its done automatically and there is hard to tell how an specific node will be called, if the .nam file its loaded after the combination the load number of nodes it contains will be already be taking for another node in other body.*
>
> *To avoid this the set files are stored with its own mesh which they make reference to in a single .inp file, this chains the sets to the part it belong and allows to correctly load the groups when the mesh body its combined with other, this force CGX to rename the mesh and the .nam numbers on the set to match the new updated ones.”*

8. The next step is to combine this archives into a single .inp file just by coping their content and pasting on the new file, keeping the order in which they have been created; this will create a single file containing the mesh and the sets for the whole part, in this case there are only to pieces but in a bigger analysis this method of capsuling the parts with all its geometric characteristics allow to easily make changes on assembly, replace and optimize specific aspects.



9. Repeating this for the next body will end up in just two different .inp files that will be use for the construction of the model, notice that by this point the two bodies are not assembled there are just defined with is sets independently.

10. Assembling on CGX: to unite the bodies in a single mesh it is necessary to load the .inp files on a CGX session, this its done by reading the files making sure to add the new entitis and no replacing the old ones:

    - **Read hookbase.inp**
    - **Read hookcore.inp add**

    Notice the "add" option at the second command that ensure that the same named entities do not replace the ones on the hook base part.

    - **Plot e all**

    To make sure it is all there

    - **Prnt se**

    To see the sets, it is recommended to plot every one to make sure there are not corrupted.

    All the sets can now be exported as part of a single group of boundary and loads of the common mesh file, the total files names taking part on the analysis are show on the pre processing tree; the next processing chapter of the tutorial its based on the names there described.

CHAPTER THREE

Processing.

```
** ----------------------------(1)-----------------------------
*INCLUDE,INPUT=all.msh
*INCLUDE,INPUT=Nslave.nam
*INCLUDE,INPUT=Nyload.nam
*INCLUDE,INPUT=Nzload.nam
*INCLUDE,INPUT=Nfix.nam
*INCLUDE,INPUT=Smaster.sur
*INCLUDE,INPUT=Ehookcore.nam
*INCLUDE,INPUT=Ehookbase.nam
** ----------------------------(2)-----------------------------
*MATERIAL, Name=steel

*ELASTIC

200000,.26

*MATERIAL, Name=copper

*ELASTIC

110000,.35

*SOLID SECTION, Elset=EEhookcore, Material=steel

*SOLID SECTION, Elset=EEhookbase, Material=copper
** ----------------------------(3)-----------------------------
*SURFACE,NAME=Slave,TYPE=NODE
NNslave
*CONTACT PAIR,INTERACTION=contact, ADJUST=0.01,
SMALL SLIDING
Slave,SSmaster
*SURFACE INTERACTION,NAME=contact
*SURFACE BEHAVIOR,PRESSURE-
OVERCLOSURE=EXPONENTIAL
0.01,10
** ----------------------------(4)-----------------------------
*RIGID BODY,NSET=NNzload,REF NODE=840
*RIGID BODY,NSET=NNyload,REF NODE=971
** ----------------------------(5)-----------------------------
*STEP, INC=1000
*STATIC
*BOUNDARY
NNfix,1,3
840,1
971,1
*CLOAD
840,3,-30000
971,2,-5000
** ----------------------------(6)-----------------------------
*NODE FILE
U
*EL FILE
S
*CONTACT FILE
CDIS,CSTR
*END STEP
```

1. Invoking the set files: the files created to the analysis, storage in the same folder will be called for CCX on the header of the .inp file by the card:

   ***INCLUDE,INPUT= file.extension**

   Where the mesh, loads and set groups are loaded to the analysis cache. Actually the user can choose to combine all this files in a single .inp file and then on the bottom type the analysis definition, but in most cases this method is impractical by the big sizes of the mesh files and the time consuming the load of this text files to edit them.

2. The material definition of the two bodies as defined previously on chapter One:

   ***MATERIAL, Name=steel**
   ***ELASTIC**
   200000,.26

   This card define the properties of a named "steel" material following the plastic properties and defining the two necessary parameters, the elasticity module (E) and the poissons ratio (x).

   Taking in account that the loads on this analysis do not overcome the maximum tensile strength yield stress where the material show a non elastic properties and the *ELASTIC card is not longer recommended.

   ***SOLID SECTION, Elset=EEhookcore, Material=steel**

   This card assigns the properties defined con the "steel" material to an already existent set of elements named Eehookcore.

3. The contact definition of the on the file is restricted to a contact pair (many contact pairs may be defined on a single analysis as need it by the user) where a surface made by nodes or faces take the place of SLAVE and the other as MASTER.

   ***CONTACT PAIR,INTERACTION=contact, ADJUST=0.01, SMALL SLIDING**
   Slave,SSmaster

   Each contact pair its given a single name to assign the properties ruling the contact behavior between the two sets named "Slave" and "Ssmaster", notice the two additional properties ADJUST and SMALL SLIDING which respectively fix any gap or over closure produce by the discretization of a curve face where some element edge may move away or interfere with the other face and define mathematical case where the coupling its calculated only ad the beginning of the increment step and remains until the next one, this allow to simplify the contact analysis.

   ***SURFACE INTERACTION,NAME=contact**
   ***SURFACE BEHAVIOR,PRESSURE-OVERCLOSURE=EXPONENTIAL**
   0.01,10

   The over closure conduct its controlled by the EXPONENTIAL parameter of the SURFACE BEHAVIOR card. The exponential pressure over closure behavior takes the form in Figure 105.
   The parameters c0 and p0 define the kind of contact. p0 is the contact pressure at zero distance, c0 is the distance from the master surface at which the pressure is decreased to 1% of p0. A large value of c0 leads to soft contact, a small value to hard contact.

Defining correctly this two values regulates the whole contact performance and it requires a highly consciously manage by the user.

4. The faces that will be loaded with a single force it needs to be restricted to a single node, preferably bellowing to the set, that will be a representation of the whole set taking any property and connecting it to all the rest nodes on the set.

   This allow to apply any load and restriction to a single node not worrying about the geometry, center of gravity or distribution of the related nodes; if a distributed load where applied there is no way to control the direction of the resultant force on the nodes because the pressure over each element will be a normal resultant on the external face. If a single value concentrated load its applied on all the nodes of the set the resultant force will not be distributed on the face but concentrate on the edges of the faces where are more number of nodes together, also in most cases the variable number of nodes on the set means there is hard to know how many times the load will by applied.

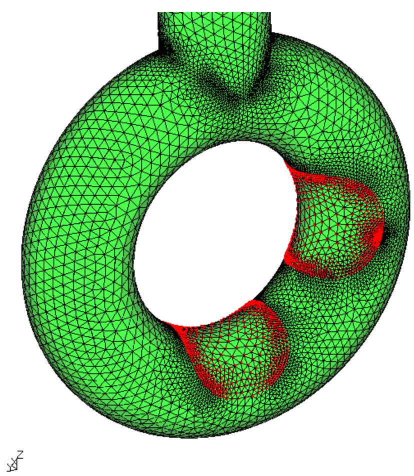


Image 3: Load faces and the set nodes.

   ***RIGIDBODY,NSET=NNzload,REF NODE=840**

   This card creates a rigid body equation on the set named "Nnzload" and defines a reference node "840".

5. In this case the loads and MPC's are called in to the STEP definition and related the previously created rigid body equations with the two reference nodes for the two load faces.

```
*BOUNDARY
NNfix,1,3
840,1
971,1
*CLOAD
840,3,-30000
971,2,-5000
```

   The fix faces on top of the hook base are restricted on the three displacement axis to ensure a complete clamping

6. The result data requested for this analysis like displacements, stress and contact information is define by the cards:

```
*NODE FILE
U
*EL FILE
S
*CONTACT FILE
CDIS,CSTR
```

   The results will be printed on the .frd file and will content the next data, U = displacements on nodes, S = stress on elements, CDIS = relative contact displacement and CSTR = the contact stress between the faces. The data that its no requested on this card will not be able to achieve without running again the case.

The processing of the Calculix input file its done by running the CCX command on any system console prompt or build in command line.

(optional)

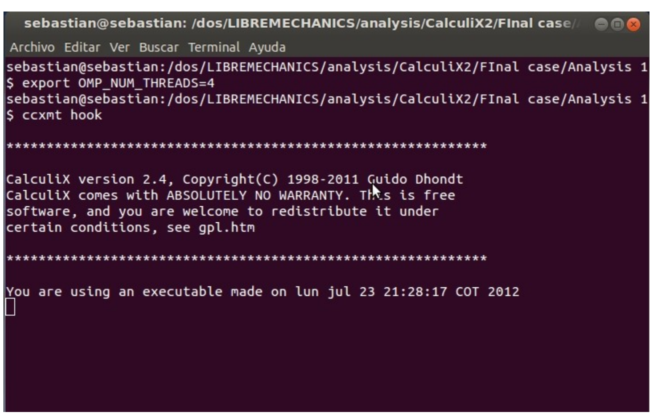
- **export OMP_NUM_THREADS= #**

This optional step will define a multi core , if the user does not have CCX compiled as an out of core application may want to check documentation.

- **CCX hook.inp**

  To run the CCX application over the .inp file previously explain.

Replace the CCX command to any given name to the CalculiX processor on local system, the .frd file will be saved on the working folder, measures most be taken to ensure there is enough disk space to store, the .frd file on big contact analysis usually exceeds by far the size of the rest of files on the working folder.



CHAPTER FOUR

Post-processing.

When the .frd file is ready CGX can now read the result file from the analysis

- **CGX hook.frd**

To read the result file on CGX module

This will load all the results but will not identify the sets previously created, when a complex assembly is handled its important to manage the same sets in other to plot just parts or especial sections of the model, this is done by reading again the names of the elements of each body and by this creating again the set names, convert the .nam element set names into a .inp an then:

- **read Ehookbase.nam**
- **read Ehookcore.nam**

To load the sets into CGX

- **prnt se**

To print all the sets names

- **plot e EEhookelements**

This is useful where a lot of bodies are involve in the same file or when there are enclose parts that must be plot one by one, if the user finds this procedure to exhaustive it can also allow to CGX automatically look for independent bodies and difference them by assigning a standard name (this may take some time that could be really long for heavy mesh)

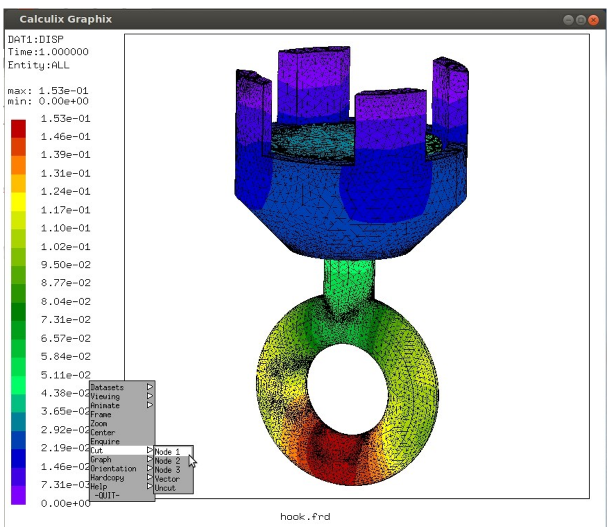
(optional)

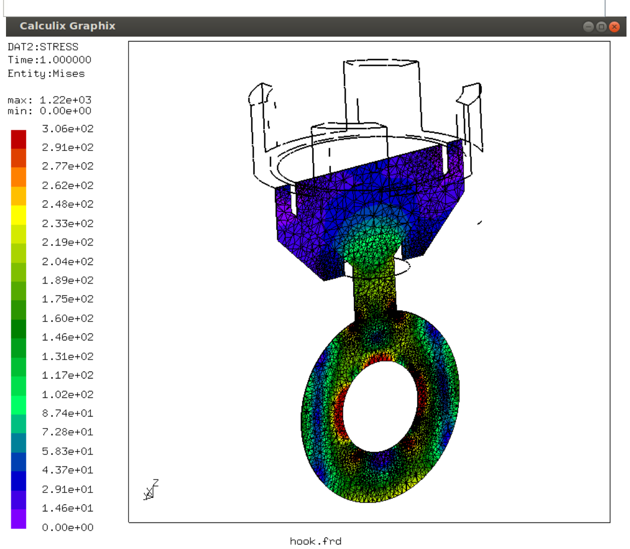
- **seta ! All**

To set the bodies automatically

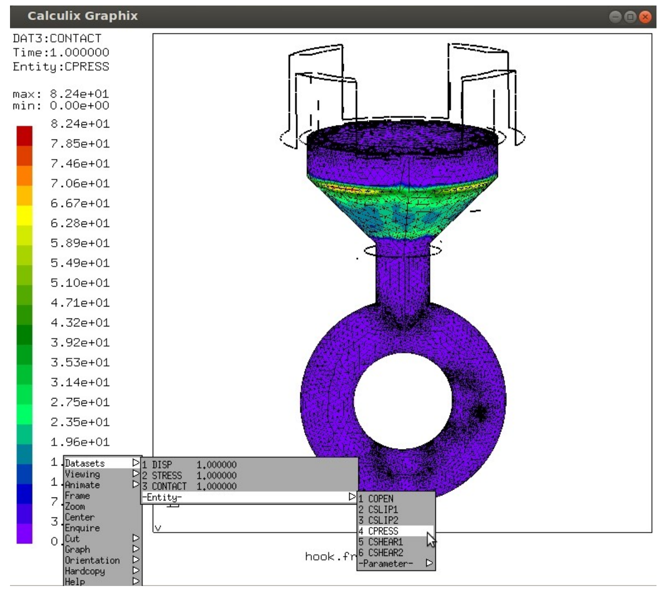
Cutting the model to see internal data it’s a great tool to simplify post processing,t he user can choose the plane orientation o exactly match the desire section by selecting 3 nodes where the plane will cross, this will smoothly cut the elements in that plane and expose the front and reverse face of the plane, usually keeping the body lines for reference.

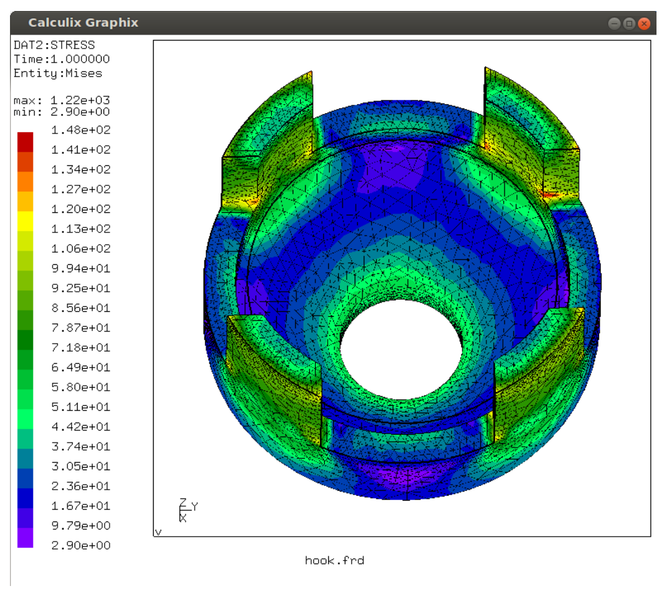
The ideal stress distribution on the coupling interface should show a continuity of the color degrade trow the other body, which indicate that the both surfaces are under the same load.





The contact pressure can be plotted on the external faces of the slave surface of the couple, the other pressure distribution doesn't need to be plot because it must be exactly the same due its generated by the two faces.





CHAPTER FIVE

Acquiring the case files

For the ease follow of this tutorial the different used and generated files named on the different chapters are available for download, allowing the user to skip or compare any step of the tutorial by its own. Please keep in mind that any file may vary from user to user by the meshing and computational conditions, but it does not meaning this difference will represent an error of processing.

- Geometry
- Mesh
- INP: Files
- Result Files

Most of the documents recurses as images and this tutorial its available at www.libremechanics.com and the sourceforge page

More Information

There are multiple ways to acquire more information about CalculiX and FEM analysis in general useful for further work:

- The CalculiX mail list
- The CalculiX yahoo group.
- B-converged web page.
- Libre Mechanics web page.

Please feel free to redistribute comment, suggest and contribute to this or any documentation found on **Libre Mechanics** web by contacting the author at contribute section.