An introduction to Salome & Code _Aster: A simple linear statics analysis of a piston J.Cugnoni, CAElinux.com, 2005

Updated to comply with SaloméMECA 2010.2 by C.Andersen, for CAELinux.com, 2010

Overview:

- Geometry module:
 - Import and prepare STEP file format geometry, group creation
- Mesh module:
 - Meshing geometry in preparation for FE study
- Aster module:
 - Creating a linear statics study using the wizard
 - Running the calculation
 - Viewing the run-time performance results
 - Editing a command file with Eficas
- Post-processing module (VISU)
 - Post-processing the result of the study

Geometry module: Importing geometry

- Start SaloméMECA and select Geometry module, choose New in the dialog
- Select menu File → Import
 - Choose **STEP** file format in the pull-down menu
 - Select the file '**Piston.stp**'
- Click the **Fit All** icon in the View tool-bar 2
- If the geometry is presented as a wire-frame, click menu $View \rightarrow View Mode \rightarrow Shading$

•	SALOME 5.1.4	
<u>File Edit View T</u> ools	Window Help	Salome 5
🗋 📄 🗟 💥	🗈 🚱 Geometry 🕹 🚳 🎥 📚 🔛 🖉 🕬	
	Activate module	
	You're activating module Geometry . Please, select required action by pressing the corresponding button below.	
	<u>N</u> ew <u>Open</u> <u>L</u> oad <u>C</u> ancel	



Geometry module: Preparing geometry; creating groups

- Expand Geometry in the object browser and right-click 'Piston.stp_1' and select Create Group
- Select Shade Type Face 💿 🔳
 - Under Group Name enter 'Press'
 - Move the mouse pointer over the piston geometry and press LMB to select the top, circular face.
 - Once the top, circular face is selected, press Add, followed by Apply.
 - The geometry group '**Press**' is now created
- Creating the 'hold' group
 - Enter the name 'hold' for this new group
 - Pressing and holding *CTRL* while using *RMB* to rotate the piston geometry, select the two upper halves of the holes going through the piston hold *SHIFT* to select two individual faces
 - Click Add, followed by Apply and Close
- To make sure the correct face groups have been created, expand **piston.stp_1** in the object browser, select both groups (pressing and holding *CTRL* or *SHIFT*) in the object browser, right-click and select **Show Only**
- To return to a normal view, right-click **piston.stp_1** and select **Show Only**







An introduction to Salome & Code _Aster: a simple linear statics analysis of a piston J.Cugnoni, 2005 / C.Andersen, 2010 for CAELinux.com

Meshing the geometry

- Switch to the MESH module
- Click Create Mesh 👍
 - Change the name from Mesh_1 to Piston
 - Under Geometry, make sure piston.stp_1 is selected, otherwise select piston.stp_1 in the object browser and click the blue arrow.
- Click Assign a set of hypotheses select Automatic Tetrahedralization
 - Click Cancel in the dialog Hypothesis Construction Max Length
 - Click the 1D tab, under Hypothesis, click the gear icon and select Average Length
 - Under Length, change the value to 0.005 click OK
 - Click Apply and Close
- Expand Mesh, right-click the Piston mesh and select Compute
- When the mesh is computed, close the dialog
- Right-click on the **Piston** mesh (notice the yellow !-sign is gone) and select **Convert** to/from Quadratic
 - Convert to quadratic will automatically be selected
 - Click Apply and Close

•	SALOME 5.1.4 - [Study1]	
<u>File E</u> dit <u>V</u> iew Mesh Controls	Modification <u>T</u> ools <u>W</u> indow <u>H</u> elp	SALOME 5
	📕 Mesh 🕹 😸 📂 🔛 🚏 🛤 🥙 🦂 🍪 👋 绌 🔹	💎 » 🔕
/ 7 8 0		» 🔯 »
Object Browser		x
Name Val	Create mesh	A 33
🖻 🐼 Geometry GEOM	Name Piston	
□ piston.stp_1 □ Press □ piston.stp_1 □ piston.stp_1 □ piston.stp_1	Geometry piston.stp_1	
	3D 2D 1D 0D	
	Algorithm <none></none>	
	Hypothesis <none></none>	
	Add. Hypothesis None>	
	Assign a set of hypotheses	
	Automatic Hexahedralization Close Help	
	Automatic Tetrahedralization	

An introduction to Salome & Code	Aster: a simple linear statics analysis of a piston
J.Cugnoni, 2005 / C.A	ndersen, 2010 for CAELinux.com

•	SALOME 5.1.4 - [Study1]	
<u>File E</u> dit <u>V</u> iew Mesh Controls	Modification <u>T</u> ools <u>W</u> indow <u>H</u> elp	CALOME 5
📄 📄 昆 🗶 🗈 🛍	🕨 Mesh 🛛 😼 📚 🖹 🚏 🎫 🏧 🍓 🍪 🚄 🔹 🤜) » 🔕
/ 7 8 0	A I & I & A & A & A & A & A & A & A & A	» 🔯 »
Object Browser		X
Name Val	Create mesh	• •
📴 😂 Geometry 🛛 GEOM	Name Piston	• "
in piston.stp_1	Geometry piston.stp_1	
Mesh SMESH	3D 2D 1D 0D	
	Algorithm Wire discretisation	
	Hypothesis Arithmetic 1D	
	Add. Hypothesis None> Automatic length	
	Assign a set of hypotheses Fixed points 1D	
	Apply and Close Apply Close Average length	
	Apply and close Apply Close Max Size	
	Nb. Segments	
	Source Edge	
	Start and End Length	
J		



•			SALOME 5.1.4 - [Study1]
<u>F</u> ile <u>E</u> dit ⊻iew M	esh Controls M	lodification <u>T</u> ools	s <u>W</u> indow <u>H</u> elp	SALOME 5
•	Mesh compu	utation succeed	•	• 🚰 🎫 🍇 🍓 🍪 绌 🛛 » 🤝 🗞
Compute mesh				🗌 🗱 👹 🥔 💖 🛷 🏄 🕅 🌈 🍋 » 🔯 »
۰ 🎕				VTK scene:1 - viewer:1 23
Name				
Piston				
(Mesh Infos				
	Total	Linear	Quadratic	
Nodes :	2722			
0D Elements :	0			
Edges :	696	696	0	
Faces :	4688	4688	0	
Triangles :	4688	4688	0	
Quadrangles :	0	0	0	
Polygons :	0			
Volumes :	8699	8699	0	
Tetrahedrons :	8699	8699	0	
Hexahedrons :	0	0	0	
Pyramids :	0	0	0	
Prisms :	0	0	0	
Polyhedrons :	0			
			C <u>l</u> ose	

An introduction to Salome & Code _Aster: a simple linear statics analysis of a piston J.Cugnoni, 2005 / C.Andersen, 2010 for CAELinux.com

•	Convert to/from quadratic	
Mesh 🥜 🛛	Piston odes on geometry	
 Convert Convert 	to quadratic from quadratic	
Apply and Clo	se <u>A</u> pply C <u>l</u> ose <u>H</u> elp	

Aster module – Creating a study using a wizard

- Switch to the Aster module
- Click the Linear Statics Wizard icon K
- In the following dialog **3D** is automatically selected click **Next**
- In the next dialog select the **Piston mesh** in the object browser under **Mesh** and click the blue arrow
 - Use Geometrical Groups should be selected this will import and create mesh groups created in the Geometry module
 - Click Next
- In the next dialog enter a Young's Modulus (E) value of 70e9 (or 7000000000) and a Poisson's ration of 0.3
 - MKS SI-system is used here and thus all dimensions are in meters, and pressure will be in Pascals 70e9Pa corresponds to a E-module of 70GPa
 - Click Next
- In the next dialog **Boundary Conditions** are imposed on the model
 - Double-click the first row in the **Group** column, press the down-arrow and select the '**hold**' group
 - Leave values for **DX,DY** and **DZ** at **0** and click **Next**
- In the next dialog a group for applying pressure must be selected
 - Again, double-click the first row in the first column, pull down the menu and select the **Press** group
 - Double-click the first row in the second column and enter a pressure value of **300e3** or 300000
 - Click Next
- In the next dialog a name and a destination for the created command file must be selected
 - Click the file dialog button
 - Navigate to the folder that should contain the study files and type in a name for the command file click **Finish**

•	Qt-subapplication			1
Isotropic linear elastic Model definition	study			
What kind of model do you	want to work on?			
3D			Ŧ	
				_
	< <u>B</u> ack	<u>N</u> ext >	Cancel	
•	Qt-subapplication			
Isotropic linear elastic Mesh selection	study			
Select a mesh from the S	alomé object browser			
Piston				
 Use mesh groups Use geometrical group 	ns			
	P 3			
		1		-
	< <u>B</u> ack	<u>N</u> ext >	Cancel	

•	Qt-subap	olication		
Isotropic linear elas Material propertie	s tic study s			
Young's modulus and	Poisson ratio def	initions		
Young's modulus (E)	70e9		(E >= 0)	
Poisson's ratio (ν)	0.3		(-1 <= v <	(= 0.5)
		< <u>B</u> ack	<u>N</u> ext >	Cancel
•	Qt-subap	olication		
Isotropic linear elas Boundaries condit	itic study ions			
	ad dagrage of fra	odom on groupo		
	ed degrees of fre	eaonn on groups		
Group	DX	DY	DZ	_
Press	0	0	0	
hold				

•			Qt-subapp	plication 🖤 🔵		
'	Isotropic linear elastic study Boundaries conditions					
	4	Adding press	ure on meshes gr	roups		
		Group	Pressure			
	•	Press	300000			
				_		
I						
				< <u>B</u> ack <u>N</u> ext > Cancel		
•			Qt-subapp	plication 🖤 🖲	•	
1	lsot I	ropic linear ela s Filename for writti	stic study ing command file			
,	Aste	r command file	/home/claus/Dowr	nloads/Piston/Piston.comm		
				< <u>B</u> ack <u>F</u> inish Cancel		

• Expanding the Aster tree in the object browser a new study should now be present:



- Right-click linear-static and select Run
- A terminal window should now appear and a lot of text will scroll by
- After the calculation has finished, the terminal will disappear and you will be presented with a green tick on the linear-static study:
- .

•

Aster Aster	ASTER
占 😱 linear-static	ок
🕂 Data	
🕂 Astk parameters	
⊕ Results	

• Take a moment to thoroughly enjoy this seemingly innocently looking icon, because you WILL learn to love it once you start playing around and your studies fail :)

• Expand the Results tree in the Object Browser, there will be 4 entries:



- •
- Right-clicking each entry will give you options to **Read as Text** or **Edit as Text** or other
- **job-id**: Self-explanatory
- **linear-static.mess**: This file contains messages produced by the calculation and should be the first place to search for errors.
- linear-static.resu: This file contains among other things, short description of errors encountered and a detailed summation of run-time performance, i.e. how long time it took for each command to run and overall time consumed, in this case:
 TOTAL_JOB: ELAPSED: 9.42 seconds
- **linear-static.rmed**: This file containing the results to be used in the Post-processing module

Aster module – Editing a command file

- Should a mistake have been made during the wizard, such as selecting a wrong group, entering a wrong value for pressure etc., the command file can easily be edited from within the Aster module:
 - Expand the linear-static study tree and expand the Data branch like so:



- When right-clicking the **Piston.comm** command file, 3 choices are presented:
- Read as Text : Will display the command file in a window in read-only mode
- Edit as Text : Will open the command file in an external editor (of the users choice)
- **Run Eficas**: Will open the command file in Eficas, an editor specially designed for creating and editing Code_Aster command files. This editor will take care of the very strict syntax Code_Aster requires, and is highly recommended for beginners for anything other than changing simple values in a command file.
- Right-click the Piston.comm file in the object browser and select Run Eficas
- In this introduction only 2 entries will be mentioned:
 - **DEFI MATERIAU** Define material:
 - ELAS Elastic material
 - The values entered earlier are present here: E-module, NU (Poissons ratio)
 - **AFFE_CHAR_MECA** : Assign boundary conditions and loads:
 - DDL_IMPO: The face group 'hold' is restricted in all directions, i.e.
 DX,DY,DZ = 0 (DDL: Degrés De Liberties = degrees of freedom, IMPOsed)
 - **PRESS_REP** : A pressure of 300 x 10³ Pascals is imposed on the '**Press'** face group
- To change a group under e.g. **PRES_REP**, simply click **GROUP_MA**, use the right hand arrow to move the name into the Value (Valeur) field and clear it. Either type the name of the group, *or* select the face group in the **Piston mesh** in the **MESH** module and press the button I've marked in the following picture curiously the icon image is gone in my version. Use the left hand arrow to enter the new name, confirm with 'Valider' a new value/name should now be present under **PRESS_REP** → **GROUP_MA**



An introduction to Salome & Code _Aster: a simple linear statics analysis of a piston J.Cugnoni, 2005 / C.Andersen, 2010 for CAELinux.com

Saisir Valeur	
Valeur(s) actuelle(s)	Valeur
Press	Image: Select a group from Salomé - mesh module Parametres Importer
	Valider

Post processing

- Click the Post-Pro icon to switch to the Post-Pro module 🔯
- Expand the Post-Pro tree in the object browser until it looks like this:
- Displacement / Deformation:

٠

- Under **RESU_DEPL_**, right-click **0**, and select **Deformed Shape**
- Enter a Scale factor of 300, tick Magnitude Coloring and press Apply and Close
- Now expand 0,_, right-click Def.Shape and select Show Only

Deformed Shape	
Deformed Shape Scalar Bar Input	
Scale Factor: 300	\$



- Von Mises stresses:
 - Under RESU_EQUI_NOUE_SIGM, right-click 0, and select Scalar Map
 - In the pull-down menu Scalar Mode, select VMIS.- and click OK
 - Again, expand 0,_, right-click ScalarMap and select Show Only

 Scalar Bar Properties 	9
Scalar Bar Input	
Scalar range:	
Scalar Mode [1] VMIS, - 🚽 🗖 Logarithmic	
Gauss Metric Average	
⊙ Use field range ⊖ Use imposed range	
Min: -550196 Max: 1.50348e+07	
Colors and labels	
Nb. of colors: 64 🔷 Nb. of labels: 5 🔷	
Orientation	
○ Vertical	
Origin	
X: 0.1 🔷 Y: 0.01 🗢	
(Dimensions (in % of the size of view)	
Width: 0.8 Height: 0.08 \$\$	
Text properties Bar properties	
Preview Hide scalar bar	
Show distribution Save as default	
<u>O</u> k <u>C</u> ancel <u>H</u> elp	



Congratulation , you have finished this first introduction to Salomé & Code_Aster.

Don't forget to visit our website for more information:

www.caelinux.com

J.Cugnoni, CAElinux.com, 2005

Updated to comply with SaloméMECA 2010.2 By C.Andersen, for CAELinux.com, 2010