

EUROPLEXUS

Version de Production 2024.0

Rapport de vérification et validation

Principe de vérification/validation de la Version de Production	2
Dates de référence	2
Méthode d'analyse des résultats des tests de validation	2
Indication de gravité pour les non-conformités dans les résultats des tests de qualification	3
Remarques sur l'exécution et la présentation des tests	3
Exécution séquentielle des tests de validation du code EUROPLEXUS	4
Récapitulatif du passage des tests : Exécution séquentielle	4
Exécution parallèle MPI des tests de validation du code EUROPLEXUS	6
Ajustement des jeux de données pour l'exécution parallèle	6
Récapitulatif du passage des tests pour 4 processus MPI	6
Détails de l'exécution de chaque test	8

Principe de vérification/validation de la Version de Production

La Version de Production du code EUROPLEXUS (EPX dans la suite du texte) est construite sur la base de la Version de Développement du code, élaborée selon un processus continu dans le cadre du Consortium EUROPLEXUS, impliquant les copropriétaires du code, le Commissariat à l'Énergie Atomique et aux Énergies Alternatives (CEA) et le Centre Commun de Recherche de la Commission Européenne (CCR), et les Partenaires Majeurs disposant d'un accès complet au code source, à savoir Électricité de France (EDF), l'Office National d'Études et de Recherches Aérospatiales (ONERA) et SAFRAN. Ce document présente d'abord la validation de la version séquentielle du code. Les résultats des tests effectués avec la version parallèle à mémoire distribuée sont reportés à la fin du document.

Tout comme la Version de Développement, la vérification principale de la Version de Production est assurée à l'aide d'une série de tests élémentaires de non-régression (**1979 tests**), fournis avec la Version et dont l'exécution sans erreur est impérative et ne fait pas l'objet du présent document.

La vérification étendue et la validation de la Version de Production sont assurées de manière systématique à partir de tests spécifiques, dits *de validation*, également disponibles sur la forge Tuleap servant de support à la Version de Développement. Le présent rapport est consacré à l'analyse de l'exécution de ces tests et peut être éventuellement accompagné de rapports de validation supplémentaires spécifiques à un ou plusieurs domaines d'application de la version, fondés sur des tests, confidentiels ou non, proposés par les industriels concernés.

Les tests mettent en œuvre les fonctionnalités du code en calcul de structure, en dynamique des fluides et en interaction fluide-structure dans des configurations physiques représentatives de son utilisation industrielle.


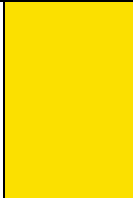


Dates de référence

Date de stabilisation de la Version de Production 2024.0	Version de Développement du 26 octobre 2023 (tag v2024.0)
Précédente Version de Production	Version de Production 2023.0, construite le 24 octobre 2022

Méthode d'analyse des résultats des tests de validation

Sauf mention particulière, chaque test est instrumenté à l'aide de la procédure interne d'EPX nommée QUALIFICATION. Elle permet de confronter la valeur pour une ou plusieurs variable(s) donnée(s) du modèle, nodale(s) ou élémentaire(s), obtenue(s) en cours ou à la fin de simulation, à une valeur de référence moyennant une tolérance. Ces valeurs peuvent venir d'une référence analytique, expérimentale, ou d'une exécution passée du programme. Le périmètre de validation est spécifié dans la description accompagnant chaque jeu de données. Certains tests (marqués N/A dans le tableau) n'ont pas de procédure de qualification et seule la bonne exécution du test à son terme est vérifiée.

Indication de gravité pour les non-conformités dans les résultats des tests de qualification

Indice de gravité	Code couleur	Signification
0		Toutes les valeurs testées dans la procédure QUALIFICATION sont conformes à la référence compte tenu de la tolérance fournie.
1		Des valeurs testées diffèrent de la référence avec un écart supérieur à la tolérance. L'analyse détaillée du test montre un déroulement correct du calcul et suggère une correction justifiée des valeurs de référence ou de la tolérance associée. Les nouvelles valeurs remplaceront les anciennes dans les jeux de données concernés à l'issue de la présente campagne de validation.
2		Des valeurs testées diffèrent de la référence avec un écart supérieur à la tolérance. L'analyse détaillée du test montre un déroulement erroné du calcul.
3		Le calcul ne va pas à son terme en raison d'une erreur d'exécution.

Remarques sur l'exécution et la présentation des tests

1. Certains tests relativement anciens peuvent demander des modifications acceptables de paramètres d'environnement du calcul avant de s'exécuter correctement, pour prendre en compte des chemins erronés vers les fichiers de maillage ou de sortie, des mots-clés devenus obsolètes ou des valeurs de dimensionnement insuffisantes (cf. Manuel Utilisateur). Ces modifications sont signalées le cas échéant, mais ne constituent pas des cas de dysfonctionnement significatif du code.
2. Pour certains tests, des illustrations des résultats de la simulation sont fournies. Leur objectif est purement qualitatif.
3. Pour la version 2024.0 d'EUROPLEXUS, tous les résultats des tests ont un indice de gravité 0.

Exécution séquentielle des tests de validation du code EUROPLEXUS

Récapitulatif du passage des tests : Exécution séquentielle

Nom du test	Nb valeurs de qualif.	Nb valeurs OK	Nb valeurs no OK	Indice (après interprétation)
vl_cea_auto_contact	3	3	0	0
vl_cea_coude_elsa	5	5	0	0
vl_cea_coude_pression	6	6	0	0
vl_cea_dadc_arrachement	5	5	0	0
vl_cea_debcrit	25	25	0	0
vl_cea_Essai44_Aquitainell_3D	66	66	0	0
vl_cea_exh2_cellules	2	2	0	0
vl_cea_jwls_2d	10	10	0	0
vl_cea_mara2	16	16	0	0
vl_cea_phi2as_treillis	3	3	0	0
vl_cea_plaq_perf	6	6	0	0
vl_cea_reac_nah2	12	12	0	0
vl_cea_rur3_air	2	2	0	0
vl_cea_sph_plaque	1	1	0	0
vl_cea_tchoc_123d	68	68	0	0
vl_cea_te_dissym	26	26	0	0
vl_cea_tubm_grfs	7	7	0	0
vl_cea_visco	2	2	0	0
vl_edf_Alu_tube_impact	1	1	0	0
vl_edf_amort_rayleigh	4	4	0	0
vl_edf_arlq1_1id	4	4	0	0
vl_edf_arlq2_2id	4	4	0	0
vl_edf_arlq3_1ic	4	4	0	0
vl_edf_arlq4_3ib	4	4	0	0
vl_edf_dalle_ponti_desm	4	4	0	0
vl_edf_eldi_feshell	6	6	0	0
vl_edf_eldi_gran_sandpile_mpi_med	1	1	0	0
vl_edf_glrc_beam	4	4	0	0
vl_edf_impact_2bars	4	4	0	0
vl_edf_meppenII-12_glrc	2	2	0	0
vl_edf_meppenII-2_glrc	2	2	0	0
vl_edf_perfo_alum_plate	1	1	0	0
vl_edf_shell_beam_pinb_cont	1	1	0	0
vl_edf_vtt_missile_impact	1	1	0	0
vl_edf_1d3d_simpson	30	30	0	0
vl_edf_1d3d_simpson_2	30	30	0	0
vl_edf_Essai44_Aquitainell_1D	35	35	0	0

<i>vl_edf_Essai45_Aquitainell_1D</i>	35	35	0	0
<i>vl_edf_cwhtf_1d_V-V_RTHER</i>	12	12	0	0
<i>vl_edf_romander_coude_1d_vf_fp_e_101</i>	34	34	0	0
<i>vl_edf_romander_coude_1d_vf_fp_e_103</i>	34	34	0	0
<i>vl_edf_romander_droit_1d_vf_elas</i>	18	18	0	0
<i>vl_edf_pipe_whip_reid_t4_1d</i>	7	7	0	0
<i>vl_fra_1d_supercanon30</i>	35	35	0	0
<i>vl_fra_1d_supercanon50</i>	35	35	0	0
<i>vl_fra_1d_supercanon70</i>	35	35	0	0
<i>vl_fra_1d_supercanon100</i>	35	35	0	0
<i>vl_fra_lame_fluide</i>	20	20	0	0
<i>vl_fra_lame_fluide_opposition_phase</i>	20	20	0	0
<i>vl_fra_poutre_epaisse_coques</i>	18	18	0	0
<i>vl_jrc_cavi51</i>	2	2	0	0
<i>vl_jrc_cont01</i>	2	2	0	0
<i>vl_jrc_cont02</i>	1	1	0	0
<i>vl_jrc_honeycomb</i>	1	1	0	0
<i>vl_jrc_hopbarur</i>	6	6	0	0
<i>vl_jrc_inde10</i>	1	1	0	0
<i>vl_jrc_inde13</i>	N/A	N/A	N/A	0
<i>vl_jrc_reac05</i>	2	2	0	0
<i>vl_jrc_sona01</i>	3	3	0	0
<i>vl_jrc_stea05</i>	2	2	0	0
<i>vl_jrc_towe02</i>	2	2	0	0
<i>vl_jrc_woco2d</i>	8	8	0	0
<i>vl_jrc_woco3d</i>	8	8	0	0

Exécution parallèle MPI des tests de validation du code EUROPLEXUS

On donne dans cette section l'évaluation des tests de validation avec la version parallèle à mémoire distribuée d'EPX. Les tests de validation qui ne sont pas fonctionnels car demandant des formulations indisponibles dans ce contexte ne sont pas pris en compte dans cette procédure de validation.

Pour chaque test retenu, les jeux de données à passer avec la version parallèle d'EPX présentent le suffixe `_MPI` dans le nom du test.

Les résultats sont donnés pour la version Intel MPI sur 4 processus parallèles.

Ajustement des jeux de données pour l'exécution parallèle

Pour l'exécution parallèle des tests de validation il faut effectuer dans certains cas des modifications :

1. Remplacer le mot clé LIAISON par LINK ;
2. Remplacer le mot clé LIAI dans la directive DIME par LCOF ;
3. Séparer le calcul et le post-traitement dans deux jeux de données différents ;
4. Faire un jeu de données pour chaque calcul ;
5. Tourner un seul cas test à la fois.

Récapitulatif du passage des tests pour 4 processus MPI

Nom du test	Nb valeurs de qualif.	Nb valeurs OK	Nb valeurs no OK	Indice
<i>vl_cea_auto_contact</i>	2	2	0	0
<i>vl_cea_coude_pression</i>	6	6	0	0
<i>vl_cea_dadc_arrach</i>	5	5	0	0
<i>vl_cea_dadc_poutre</i>	N/A	N/A	N/A	0
<i>vl_cea_debcrit</i>	25	25	0	0
<i>vl_cea_jwls_2d</i>	10	10	0	0
<i>vl_cea_mara2</i>	16	16	0	0
<i>vl_cea_phi2as_treillis</i>	3	3	0	0
<i>vl_cea_rur3_air</i>	2	2	0	0
<i>vl_cea_tchoc_123d</i>	68	68	0	0
<i>vl_cea_visco</i>	2	2	0	0
<i>vl_edf_AA7075_ballistic_test_mpi</i>	1	1	0	0
<i>vl_edf_Al_u_tube_impact</i>	1	1	0	0
<i>vl_edf_amort_rayleigh</i>	4	4	0	0
<i>vl_edf_arlq1_1id</i>	4	4	0	0
<i>vl_edf_arlq2_2id</i>	4	4	0	0
<i>vl_edf_arlq3_1ic</i>	4	4	0	0
<i>vl_edf_arlq4_3ib</i>	4	4	0	0
<i>vl_edf_eldi_feshell</i>	6	6	0	0
<i>vl_edf_glrc_beam</i>	4	4	0	0
<i>vl_edf_impact_2bars</i>	4	4	0	0

<i>vl_edf_joint_gluing_mpi</i>	1	1	0	0
<i>vl_edf_meppenII-12_glrc</i>	2	2	0	0
<i>vl_edf_meppenII-2_glrc</i>	2	2	0	0
<i>vl_edf_perfo_alum_plate</i>	1	1	0	0
<i>vl_edf_shell_beam_pinb_cont</i>	1	1	0	0
<i>vl_edf_Ti4Al6v_ballistic_test_mpi</i>	1	1	0	0
<i>vl_edf_vtt_missile_impact</i>	1	1	0	0
<i>vl_edf_HDR_mod2_rap0.99</i>	17	17	0	0
<i>vl_edf_popi</i>	3	3	0	0
<i>vl_edf_pipe_whip_reid_t4_3d_stru</i>	10	10	0	0
<i>vl_edf_pipe_whip_reid_t4_3d_ifs</i>	14	14	0	0
<i>vl_edf_vtt-x1_glrc</i>	2	2	0	0
<i>vl_edf_vtt-x2_glrc</i>	2	2	0	0
<i>vl_edf_vtt-x3_glrc</i>	2	2	0	0
<i>vl_jrc_hopbarur</i>	6	6	0	0
<i>vl_jrc_inde10</i>	1	1	0	0
<i>vl_jrc_inde13</i>	N/A	N/A	N/A	0

Détails de l'exécution de chaque test

VL_CEA_AUTO_CONTACT

Description

Titre :

Écrasement d'un cylindre sous impact avec auto-contact

Problème :

On cherche à analyser le comportement d'un cylindre en acier soumis à l'impact d'une masse tombant en chute libre. Les déformations sont telles que des plis se forment avec auto-contact des parois.

Des essais ont été réalisés au laboratoire CEA/DEN/DANS/DM2S/SEMT/DYN à cet effet.

Géométrie :

Le cylindre est droit, d'une hauteur de 200 mm et de 11.43 mm de diamètre. Son épaisseur est de 3.05 mm. Il est posé sur un massif indéformable et subit l'impact d'une masse de 300 Kg tombant en chute libre d'une hauteur de 7.35 m.

Maillage :

Il y a 42 éléments sur une circonférence et 33 éléments sur une génératrice, soit en tout 1386 éléments de type coque.

Le projectile est modélisé par un point matériel PMAT.

On propose deux modèles, l'un utilisant des éléments de coque massifs SHB8 (*vl_cea_contact_auto.epx*), l'autre des éléments coques quadrangulaires Q4GS (*vl_cea_contact_auto_shell.epx*).

Matériaux :

Le cylindre est en acier inox avec un comportement de type Von Mises isotrope VMIS ISOT. La limite élastique est de 250 MPa.

Conditions aux limites :

Les déplacements verticaux sont bloqués à la base du cylindre et pilotés par le déplacement du projectile en partie haute (directive IMPACT). Les faces internes et externes du cylindre sont susceptibles d'entrer en contact avec glissement (directive GLISSEMENT). Le traitement de l'auto-contact du cylindre modélisé avec des éléments quadrangulaires utilise la directive LINK DECO et l'option GLIS PENA SELF (auto-contact par pénalité pour éléments sans épaisseur).

Chargement :

Le système est au repos à l'instant initial et le projectile animé d'une vitesse initiale de 12 m/s juste avant l'impact.

Calcul :

Le calcul se poursuit jusqu'à 18 millisecondes.

Résultats :

A l'arrêt du calcul, on observe l'écrasement du cylindre et la vitesse de rebond du projectile.

Dans le cas avec éléments quadrangulaires, on observe un écrasement globalement identique du cylindre comparé au cas avec éléments volumiques SHB8, sauf au niveau du contact avec l'impacteur.

La surface de contact est un disque avec les éléments volumiques et un cercle avec les éléments surfaciques. Le contact bloque la rotation selon l'axe ortho-radial dans le premier cas, pas dans le second.

Commentaires :

1- L'exécution est 4 fois plus rapide pour le modèle avec éléments quadrangulaires.

2- La comparaison calcul-essai est favorable au calcul avec SHB8.

3- Le calcul avec coques quadrangulaires et auto-contact par pénalité est plus sensible aux perturbations numériques que le calcul avec coques SHB8.

Référence :

Les essais et les calculs sont décrits dans des rapports CEA, disponibles sur demande auprès du laboratoire CEA/DES/ISAS /DM2S/SEMT/DYN.

Exécution

Modifications du(des) jeu(x) de données :

- 1^{er} modèle (*vl_cea_contact_auto.epx*) : Aucune
- 2nd modèle (*vl_cea_contact_auto_shell.epx*) : 3 modifications détaillées ci-dessous ont été réalisées.
 - ➔ Le mot-clé « LIAI » devenu obsolète a été remplacé par le mot-clé « LCOF » dans la directive DIME.
 - ➔ Le mot-clé « LINK » devenu obsolète a été remplacé par le mot-clé « LINK COUP ».
 - ➔ Le groupe de mots-clés « IMPACT » a été placé dans le jeu de données après le groupe de mots-clés « GLIS ».

Remarques :

Le calcul « *vl_cea_contact_auto_shell.epx* » est uniquement exécuté en séquentiel.

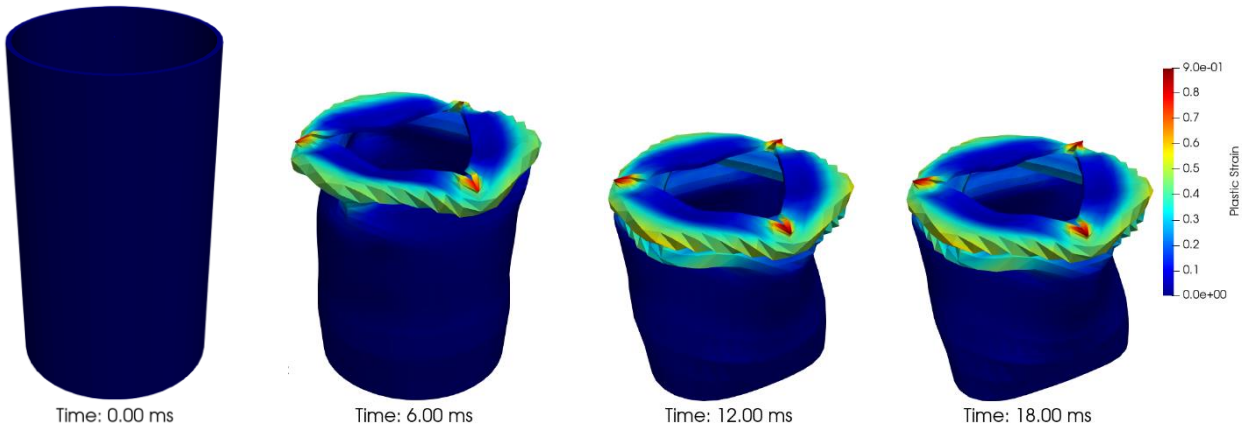
Qualifications :

OK : 2/2 pour le cas *vl_cea_auto_contact.epx*

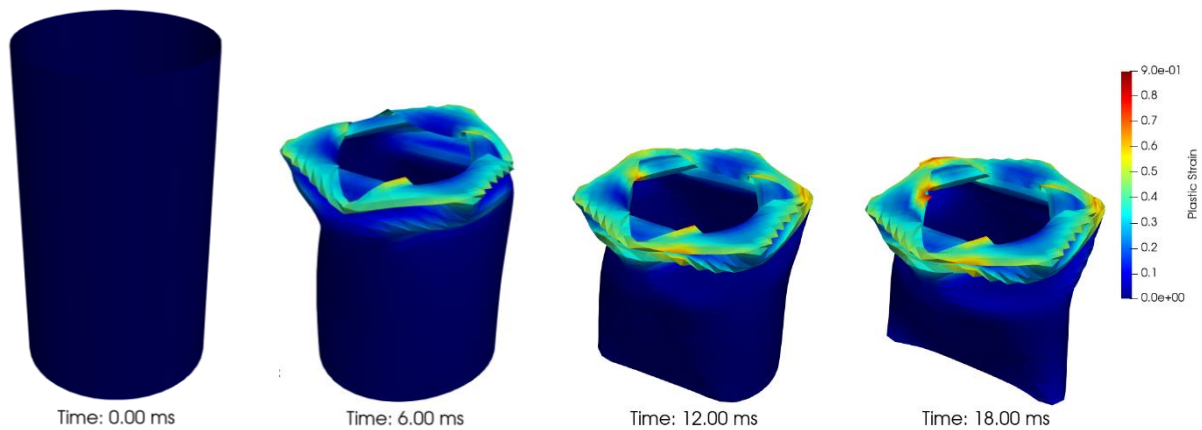
OK : 1/1 pour le cas *vl_cea_auto_contact_shell.epx*

Illustrations

Cas vl_cea_auto_contact.epx :



Cas vl_cea_auto_contact_shell.epx :



VL_CEA_COUDE_ELSA

Description

Titre :

Ouverture d'un coude sous l'action d'une force

Problème :

On cherche à vérifier le bon comportement des coudes modélisés en 1D au moyen d'éléments filaires TUYA. On considère ici un coude à 90° rempli de liquide et encastré à une extrémité. Sur l'autre, une force transverse est appliquée.

Maillage :

Le maillage est 1D filaire et ALE. La géométrie est imposée par le dispositif expérimental ELSA. Il y a 51 éléments de type TUYA et 2 éléments CLTU aux extrémités.

Matériaux :

Le matériau constituant la paroi est de type Von Mises isotrope VMIS ISOT, mais la structure reste élastique. Le fluide interne est un liquide peu compressible (matériau FLUI). Une pression est appliquée à l'encastrement (IMPE PIMP) et une condition de fond fermé est appliquée à l'autre extrémité (IMPE FOND).

Conditions aux limites :

A l'instant initial, l'ensemble est au repos et à la pression atmosphérique.

Chargement :

Une pression de 1 bar est maintenue sur le fluide à côté de l'encastrement et une force de 10 kN est appliquée à l'autre extrémité dans le plan du coude.

Calcul :

Un amortissement de relaxation est ajouté afin d'atteindre rapidement la solution statique. Le calcul se poursuit sur un temps de l'ordre de 3 périodes du mode le plus bas, soit 150 ms.

Résultats :

A l'arrêt du calcul, les résultats sont comparés avec la solution de référence donnée par CAST3M.

Référence :

EUROPLEXUS – Fiche de qualification "CIR_COUDE", M. Lepareux, rapport CEA DMT/SEMT/DYN/RT/2001-008/A.

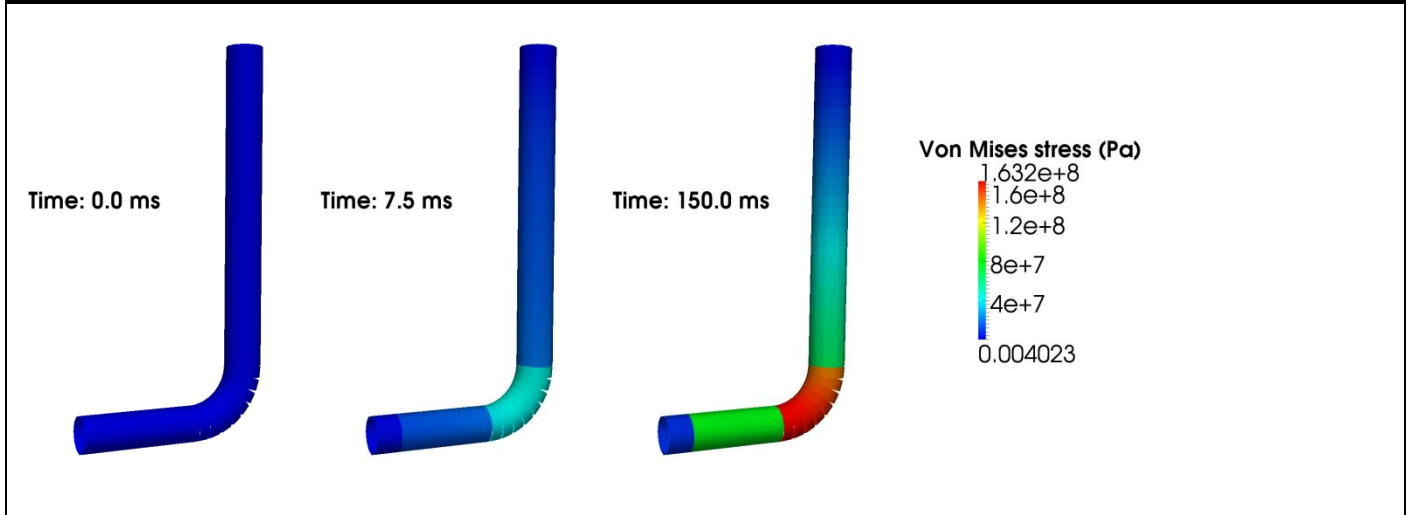
Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 5/5

Illustrations



VL_CEA_COUDE_PRESSION

Description*Titre :*

Ouverture d'un coude sous pression interne

Problème :

On cherche à vérifier le bon comportement du couplage fluide-structure dans le cas des tuyauteries. On considère ici un coude à 90° rempli de liquide encastré à une extrémité, libre et avec un fond fermé à l'autre.

Maillage :

Le maillage est 1D filaire et ALE. Le coude a un rayon de 4 m et la tuyauterie a une section circulaire de 80 cm de diamètre externe pour une épaisseur de 4 cm Il y a 32 éléments de type TUYA et 2 éléments CLTU aux extrémités.

Matériaux :

Le matériau constituant la paroi est de type élastique linéaire LINE. Le fluide interne est un liquide peu compressible (matériau FLUI). Une pression est appliquée à l'encastrement (IMPE PIMP) et une condition de fond fermé est appliquée à l'autre extrémité (IMPE FOND).

Conditions aux limites :

A l'instant initial, l'ensemble est au repos et à la pression atmosphérique.

Chargement :

Une pression de 80 bars est appliquée à l'instant initial sur le fluide du côté de l'encastrement.

Calcul :

Un amortissement de relaxation est ajouté afin d'atteindre rapidement la solution statique. Le calcul se poursuit sur un temps de l'ordre de 3 périodes du mode le plus bas, soit 200 ms.

Résultats :

A l'arrêt du calcul, les résultats sont comparés avec la solution analytique et la solution obtenue avec CAST3M.

Référence :

EUROPLEXUS – Fiche de qualification "CIR_COUDE", M. Lepareux, rapport CEA DMT/SEMT/DYN/RT/2001-008/A.

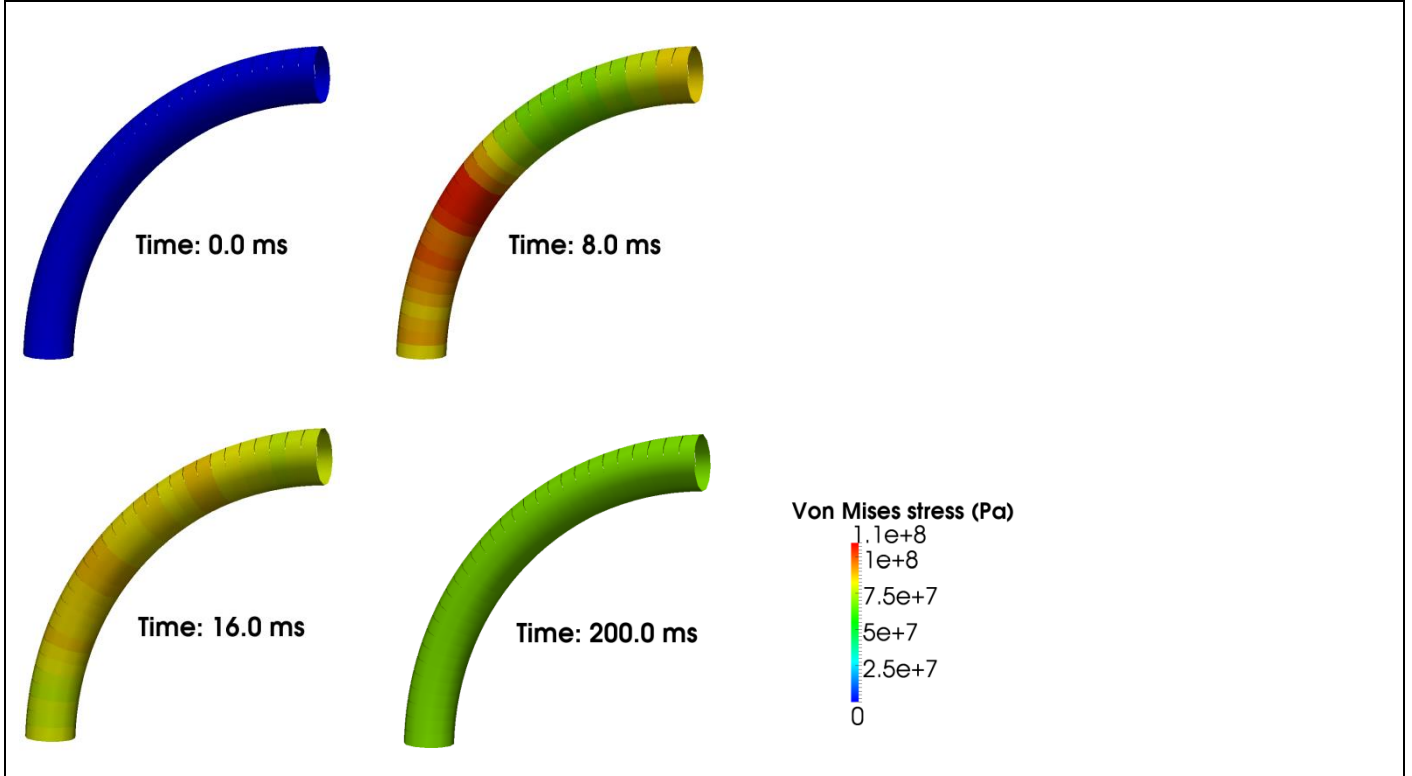
Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 6/6

Illustrations



VL_CEA_DADC_ARRACHEMENT

Description

Titre :

Test de modélisation d'un essai d'arrachement

Problème :

L'objet de ce test est de représenter simplement le comportement d'une liaison entre l'acier et le béton au cours d'un arrachement.

Maillage :

Le béton est représenté par une éprouvette cubique de 25cm de côté et est maillé en 3D. Deux volumes de matière sont retirés au centre afin d'avoir la longueur d'ancrage voulue. L'acier longitudinal a un diamètre de 12mm ; il est modélisé par des éléments 1D.

Matériaux : Le béton est représenté à l'aide du modèle DADC.

Chargement et conditions aux limites :

Un déplacement U suivant la direction Z est imposé sur le nœud de l'extrémité de l'acier, avec une vitesse de chargement de 0,25m/s. Les nœuds de la face gauche de l'éprouvette sont bloqués suivant la direction z afin de s'opposer à l'effort produit par le déplacement de l'acier.

Calcul :

Le calcul est conduit jusqu'à 0,022s.

Résultats :

A l'arrêt du calcul, les résultats sont comparés avec des valeurs de non-régression.

Référence :

Ce cas est décrit et commenté dans la thèse d'Armand Leroux, Modèle multiaxial d'endommagement anisotrope : Gestion numérique de la rupture et application à la ruine des structures en béton armé sous impacts, pp152-158.

Exécution

Le cas test est composé de deux jeux de données, *vl_cea_dadc_arrachement.epx* et *vl_cea_dadc_arrachement_post.epx*, pour séparer le calcul du post-traitement.

Modifications du jeu de données :

vl_cea_dadc_arrachement.epx : le calcul étant très long (plus de 40 heures en séquentiel), le jeu de données a été modifié en multipliant par un facteur 10 la vitesse de chargement du nœud de l'extrémité de la barre en acier. On obtient alors, en fin de calcul, le même déplacement qu'auparavant mais pour un temps final de 22 ms, ce qui réduit le temps de calcul d'un facteur 10.

vl_cea_dadc_arrachement.epx : les valeurs de référence des qualifications 1, 2, 3 et 5 ont été modifiées pour prendre en compte les modifications apportées au jeu de données, à savoir :

La valeur de référence de la qualification 1 est passée de 0.22 à 0.022 s (temps final)

La valeur de référence de la qualification 2 est passée de 511.1 à 568.0 J (travail des forces internes)

La valeur de référence de la qualification 3 est passée de 511.1 à 568.0 J (travail des forces externes)

La valeur de référence de la qualification 5 est passée de 9.710 104 à 10.744 104 N (force extérieure)

Remarques :

Après l'examen des évolutions de la force, on observe une augmentation de la valeur finale de près de 10% qui est due à la prise en compte de l'influence de la vitesse de déformation qui tend à rigidifier la structure (la vitesse de chargement étant 10 fois plus

importante). Les allures des différentes courbes sont globalement très proches et présentent peu voire pas d'oscillations parasites, contrairement à ce qui était observé dans les calculs précédents.

Qualifications :

OK : 5/5

Illustrations

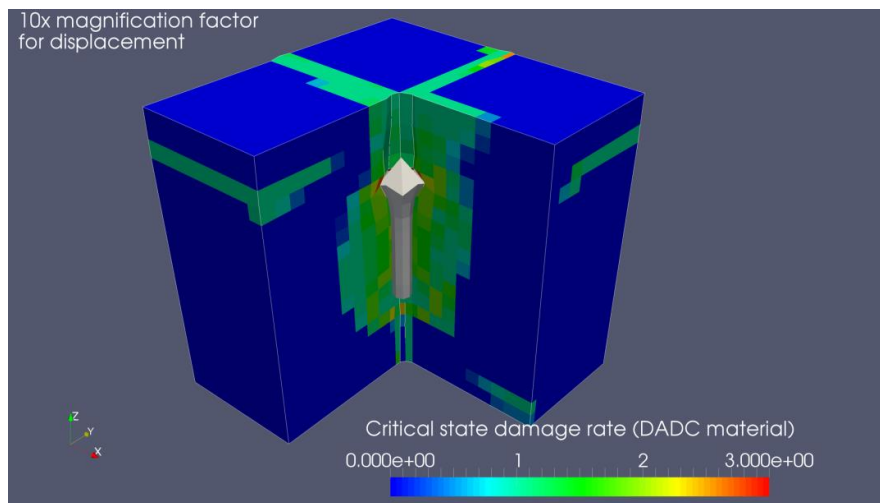


Figure 1 : t=11.4ms

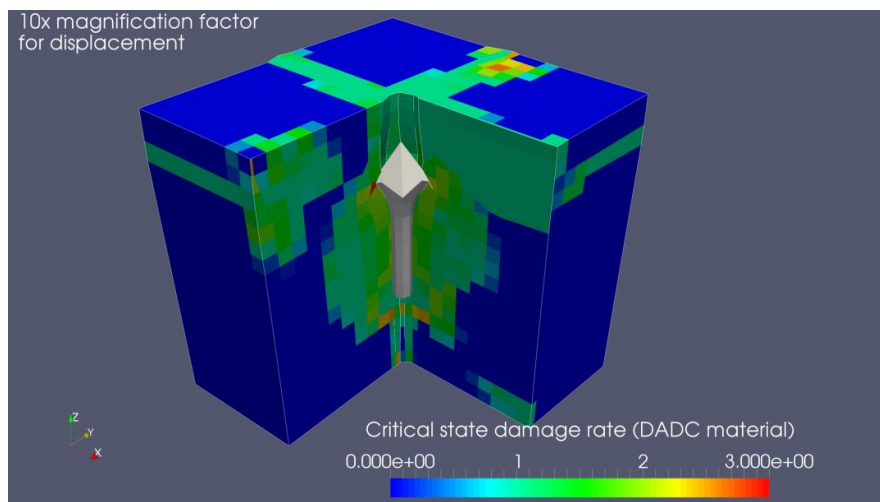


Figure 2 : t=17ms

Visualisation du champ d'endommagement dans le béton à deux instants de la simulation (somme des endommagements dans les trois directions principales). Interprétation du test à trouver dans la thèse précédemment citée.

VL_CEA_DADC_POUTRE

Description

Titre :

Impact sur la poutre P2-130

Problème :

Ce test modélise l'endommagement anisotrope d'une poutre en béton armé.

Maillage :

Le maillage de la poutre est composé de 26496 éléments hexaédriques. Les différentes armatures sont modélisées par des éléments 1D. Le projectile est modélisé par un solide indéformable de 311kg.

Matériaux :

Le béton est représenté avec le matériau DADC. L'impacteur est élastique linéaire et le tampon en nid d'abeille est représenté avec le matériau orthotrope de type BOIS.

Conditions aux limites :

Le projectile a une vitesse initiale de 5,425m/s, suivant la direction Y. Il impacte un cube de nid d'abeille.

Les nœuds de la poutre se trouvant sur les faces supérieure, face où repose le nid d'abeille, et inférieure, sont bloqués au niveau des appuis.

Calcul :

Le calcul modélise un transitoire de 35ms.

Résultats :

Les valeurs de qualification sont manquantes dans cette version du test pour l'analyse automatique du test. La validation est alors obtenue par comparaison avec les résultats obtenus dans la référence ci-dessous.

Référence :

Ce cas est décrit et commenté dans la thèse Armand Leroux, Modèle multiaxial d'endommagement anisotrope : Gestion numérique de la rupture et application à la ruine des structures en béton armé sous impacts, pp 140-152.

Exécution

Ce test est exécuté uniquement en MPI. La version séquentielle n'existe pas.

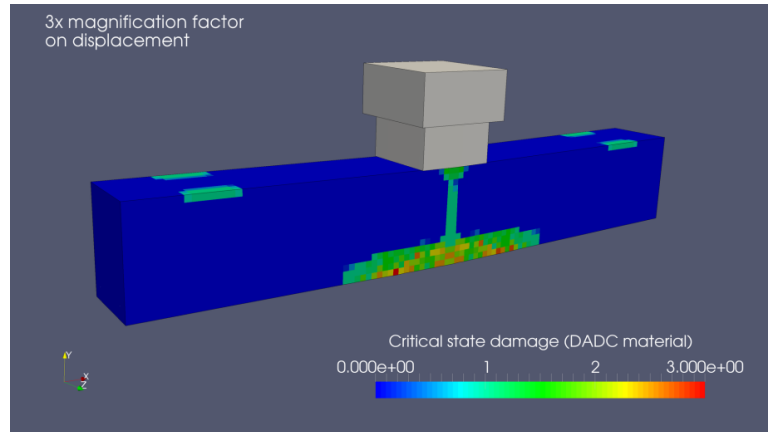
Le cas test est composé de deux jeux de données, vl_cea_dadc_poutre_mpi.epx et vl_cea_dadc_poutre_mpi_post.epx , pour séparer le calcul du post-traitement.

Modifications du(des) jeu(x) de données : aucune.

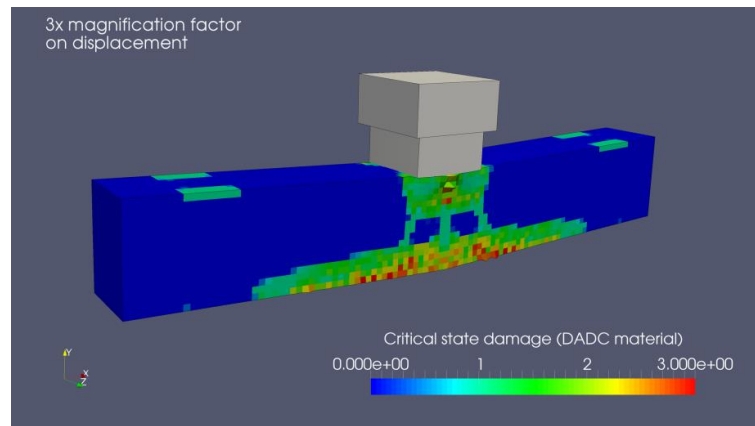
Qualifications :

Qualification manquante. Solution correcte par comparaison avec les résultats de la thèse de référence.

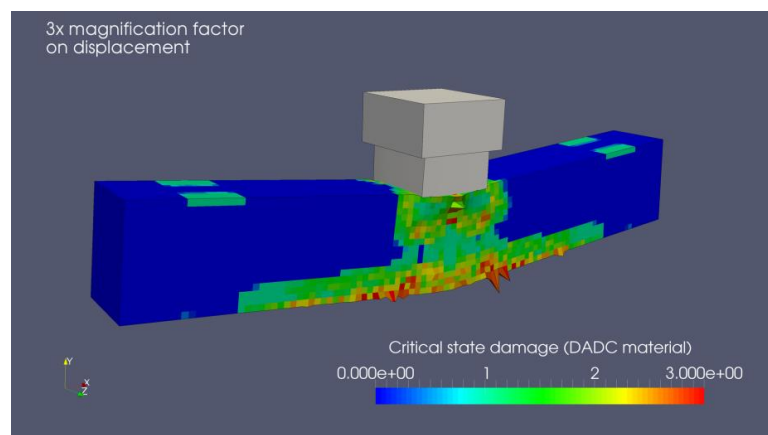
Illustrations



t=2ms



t=4ms



t=8ms

VL_CEA_DEBCRIT**Description***Titre :*

Tests de débit critique

Problème :

On cherche à vérifier le bon comportement des différents modèles de débit critique en gaz, liquide et eau diphasique.

Maillage :

Le maillage est 1D filaire et eulérien pour les 3 jeux de données (*vl_cea_debcrit.epx* pour le modèle avec eau diphasique, *vl_cea_debcrit_flu* pour le modèle avec liquide et *vl_cea_debcrit_gaz.epx* pour le modèle avec gaz).

Un réservoir de taille variable se vide à travers un tube de 1.5 m de longueur. Il y a 60 éléments de type TUBE et un élément CL1D en sortie pour appliquer le modèle de débit critique choisi.

Matériaux :

Le fluide est suivant le cas : un gaz parfait, un liquide peu compressible ou de l'eau en équilibre avec sa vapeur.

Conditions aux limites :

A l'instant initial, on ouvre une brèche de taille variable à la sortie.

Chargement :

Le système est au repos à l'instant initial et en équilibre.

Calcul :

Le calcul se poursuit jusqu'à atteindre un régime permanent.

Résultats :

A l'arrêt du calcul, les résultats sont comparés avec des résultats théoriques.

Référence :

Programme PLEXUS – Les modèles de débit critique, M. Lepareux, rapport CEA DRN/DMT 97.029.

Plexus – Fiche de qualification des modèles de débit critique "CIR-DCRI", G. Marchaud, M. Lepareux, rapport CEA/DRN/DMT/SEMT/DYN/RT/99.013/A.

Exécution*Modifications du(des) jeu(x) de données :*

Le jeu de données initial *vl_cea_debcrit.epx* présentait 8 tests enchainés par l'option SUITE. Pour rendre ce calcul disponible à la version parallèle, il a été divisé en 8 cas tests séparés et appelés : *vl_cea_debcrit_0n.epx* avec $n = 1 \dots 8$.

Pour les mêmes raisons, le jeu de données *vl_cea_debcrit_gaz.epx* a été divisé en 4 cas tests séparés et appelés : *vl_cea_debcrit_gaz_0n.epx* avec $n = 1 \dots 4$.

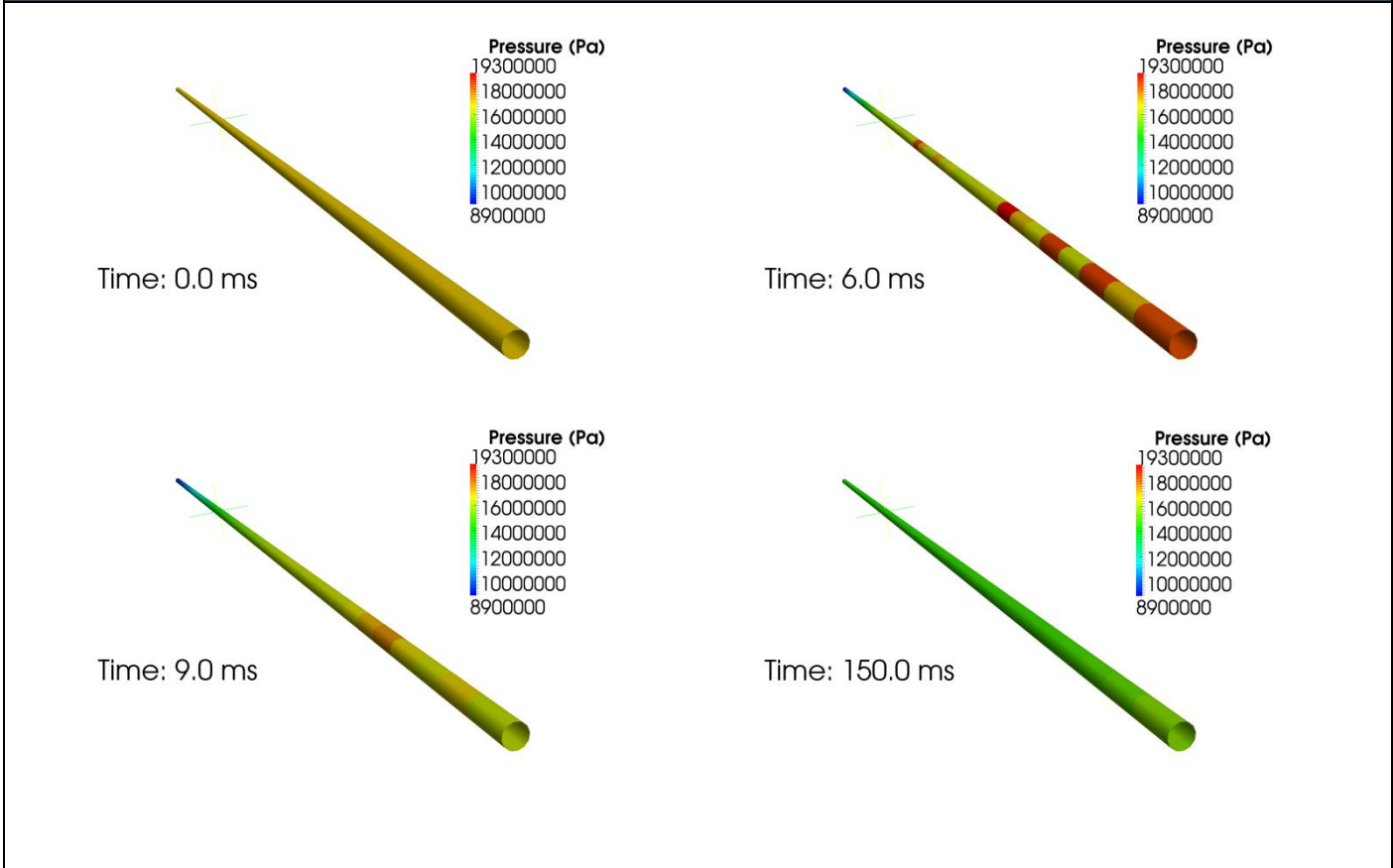
Qualifications :

OK : 19/19 pour les cas *vl_cea_debcrit_0n.epx*

OK : 1/1 pour le cas *vl_cea_debcrit_flu.epx*

OK : 5/5 pour les cas *vl_cea_debcrit_gaz_0n.epx*

Illustrations



VL_CEA_ESSAI44_AQUITAINEII_3D

Description*Titre :*

Analyse d'un essai de fouettement avec couplage 1D/3D : ESSAI Aquitaine II N°44

Problème :

Parmi les divers essais de fouettement et choc effectués dans le cadre du programme AQUITAINE II [1,3,4] nous avons retenu l'essai 44 qui est le mieux documenté. Le but de cet essai est l'obtention des efforts transmis lorsque la tuyauterie impacte une structure.

Maillage et conditions aux limites :

Les calculs ont été effectués en utilisant un maillage 1D avec interaction fluide-structure pour les parties droites flexibles de la tuyauterie (éléments de type "TUYA" : ligne AB et CD).

La partie de tuyauterie raccordant l'ensemble flexible au réservoir est supposée parfaitement rigide et composée d'éléments "TUBE", éléments de tuyauterie à 1 ddl fluide par nœud. Elle est raccordée à la ligne AB par l'intermédiaire d'un élément "BIFU" qui assure le couplage entre une tuyauterie rigide ne comportant qu'un degré de liberté fluide, et la partie flexible possédant 7 d.d.l. (6 pour la partie mécanique et 1 pour le fluide). Le coude est maillé avec des éléments de coque 3D (DKT3), le fluide interne par des éléments volumiques (PRISME). La butée rigide est représentée par un point matériel de masse très importante (100 tonnes). La tuyauterie est supposée encastrée au niveau du réservoir.

Le maillage comporte :

3 éléments « TUBE » à 1 d.d.l. par nœud (fluide seul)
21 éléments « TUYA » à 7 d.d.l. par nœud (6 mécanique et 1 fluide)
400 éléments de coques « DKT3 » à 6 d.d.l. par nœud (3 translations et 3 rotations)
660 éléments « PRISME » pour représenter le volume fluide dans le coude
132 éléments « CL3T » de conditions aux limites 3D
2 éléments de raccord « TUYM » qui permettent de raccorder les degrés de libertés fluide de la partie 1D (1 d.d.l. fluide) de la tuyauterie (élément « TUYA ») au maillage volumique du fluide (3 d.d.l. fluide)
1 élément « BIFU » à 2 nœuds qui assure la continuité des débits et de la pression entre les branches OA et AB de la tuyauterie
1 élément « CAVI » à 1 nœud et 1 d.d.l. (fluide) pour représenter le réservoir

Matériaux :

La loi de comportement est élastoplastique de type Von Mises isotrope. La tuyauterie est en acier AISI 316L. Ses caractéristiques mécaniques à 326° C sont :

- Module d'Young $E = 180 \text{ GPa}$
- coefficient de Poisson $\nu = 0.33$
- masse volumique $\rho = 7800 \text{ kg.m}^{-3}$
- limite élastique $\sigma_l = 155 \text{ MPa}$
- contrainte à rupture $\sigma_r = 475 \text{ MPa}$
- déformation à rupture $\epsilon_l = 25\%$

Le diamètre externe de la tuyauterie est de 88.9 mm et son épaisseur est de 7.62 mm. Pour le fluide nous avons utilisé le matériau eau équilibré (« EAU EQUIL ») qui fait l'hypothèse que les deux phases sont toujours en équilibre thermodynamique et permet de traiter l'eau et sa vapeur en mélange homogène. Son état est parfaitement déterminé à partir de la température et de la pression locale, par interpolation dans les tables de l'eau.

Chargement :

Prise en compte du couplage IFS.

Calcul :

Le calcul Europlexus est effectué en deux parties. Dans un premier temps nous avons recherché l'état d'équilibre de la tuyauterie soumise à une pression interne de 166 bar. Le premier calcul est fait en utilisant l'option d'amortissement quasi statique. Le calcul

est ensuite repris avec ouverture de la brèche jusqu'au temps final de 100 ms. Dans le second calcul la condition de fond fermé est remplacée par une condition de débit critique (modèle de Moody) avec un rapport de section de sortie sur section amont de 0.9.

Résultats :

A l'arrêt du calcul au temps $T = 100$ ms, les résultats sont conformes aux résultats obtenus expérimentalement. Voir [1] pour une analyse plus détaillée.

Référence :

1. P. Galon, M. Lepareux, Couplage 1D/3D : analyse d'un essai de fouettement au moyen d'EUROPLEXUS. Rapport DRN/DMT/SEMT/DYN/ RT/00.015/A
2. P. Caumette, J.L. Garcia, Experimental studies of PWR primary piping under LOCA conditions. Nuclear Engineering and Design 61 (1980) 197-208.
3. J.L. Garcia, P. Caumette, J.L. Huet, A. Martin, Study of pipe whip and impact. SMIRT 1981
4. A. Martin, E. Sermet, J.L. Garcia, AQUITAINE II complémentaire : Rapport final d'essais. Note technique DRE/STRE/LMA 84/362
5. J.L. Garcia, "AQUITAINE II - Interprétation" : analyses d'essais avec chocs sur structures rigides. Note technique DRE/STRE/LMA 84/373
6. A. Ricard, A. Millard, Fouettement des tuyauteries par le programme TEDEL. Rapport DMT/SMTS/BAMS/81-52
7. P. Galon, M. Lepareux, Comportement dynamique des tuyauteries. Analyse d'un essai de fouettement au moyen de PLEXUS. Rapport DMT/SEMT/DYN/RT/99-028

Exécution

Modifications du(des) jeu(x) de données : Aucune

Qualifications :

OK : 66/66

VL_CEA_EXH2_CELLULES

Description*Titre :*

Détonation H2 dans un tube avec 3 diaphragmes

Problème :

On cherche à suivre la progression d'une onde de choc dans un tube suite à la détonation d'un mélange d'air et d'hydrogène en présence de vapeur d'eau. Le tube supposé rigide est divisé en 4 compartiments carrés communiquant par des diaphragmes de section croissante.

Géométrie :

Le tube à 20 m de long et 5 m de côté. Les diaphragmes ont des ouvertures successives de 1.25 m, 2.50 m et 3.75 m et une épaisseur de 10 cm.

Maillage :

Le maillage est 2D plan. Il y a 16480 éléments CVL1 pour modéliser le mélange gazeux.

Matériaux :

Le matériau est de type EXVL sous 1 bar et 362.6 Kelvin avec une concentration de 26.7 % en H2 et 2.9 % en vapeur d'eau. L'ignition est faite dans un angle du premier compartiment où la température est portée à 3000 K.

Conditions aux limites :

Les bords du tube sont fixes.

Chargement :

Le mélange est au repos à l'instant initial.

Calcul :

Le calcul se poursuit jusqu'à 40 millisecondes.

Résultats :

A l'arrêt du calcul, on observe la pression au fond du tube et dans l'angle où a lieu l'ignition.

Références :

Ce calcul est un simple test de faisabilité.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque :

Des oscillations et des déséquilibres sont observés localement et de manière transitoire dans le champ de pression, ce qui correspond à un phénomène connu et accepté dans le programme, dit de *damier*, lié à la formulation des éléments.

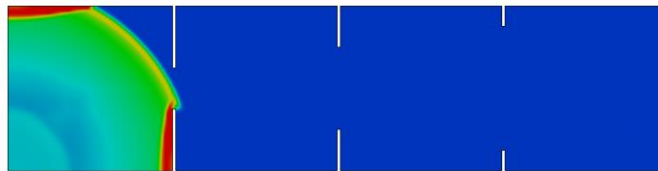
Qualifications :

OK : 2/2

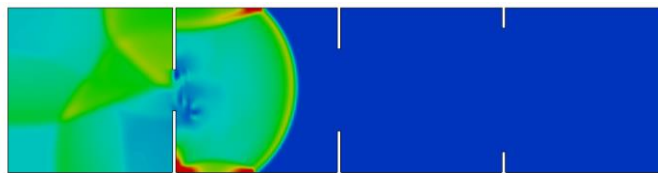
Illustrations



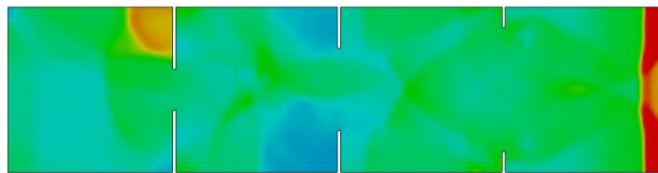
Time: 0.4 ms



Time: 3.2 ms

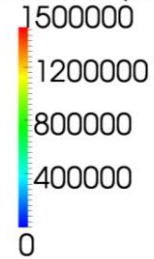


Time: 5.6 ms



Time: 13.2 ms

Pressure (Pa)



VL_CEA_JWLS_2D

Description

Titre :

Caisson rigide avec une charge de TNT au centre (modèle JWLS)

Problème :

On cherche à suivre l'explosion d'une charge de TNT dans un caisson rigide, en tenant compte de la propagation de la réaction dans la charge. On s'intéresse également aux réflexions de l'onde de choc sur les parois.

Géométrie :

Le caisson fait 156 cm de haut et 193 cm de diamètre. L'explosif est au centre et a la forme d'un cylindre de 284 mm de haut et 71 mm de diamètre. Le caisson est rempli d'air aux conditions normales. A l'instant initial, la détonation est initiée au milieu de la partie supérieure du TNT.

Maillage :

Le maillage est 2D axisymétrique. Il y a 9100 éléments CAR1 dont 400 pour le TNT. Le maillage reste fixe (calcul eulérien).

Matériaux :

L'explosif est du TNT dont les paramètres JWL sont $A=3.738$, $B=0.03747$, $R1=4.15$, $R2=0.90$, $\omega=0.35$ et $E0=0.0600$. Les pressions sont en 10^6 bars (cas de A, B, E0), les autres paramètres sont adimensionnels. La densité du TNT est 1.630 g/cm^3 . Pour la vitesse de détonation, on prend celle de Chapman-Jouguet (6930 m/s).

Attention : E0 est une énergie par unité de volume et a bien les dimensions d'une pression.

Pour l'air : la masse volumique est de 1.3 kg/m^3 .

Conditions aux limites :

Les parois sont fixes. On utilise la liaison FSR.

Chargement :

Le système est au repos à l'instant initial, quand le TNT est mis à feu dans sa partie supérieure.

Calcul :

Le calcul se poursuit jusqu'à 1.8 millisecondes, quand le TNT a disparu et que l'onde de choc s'est réfléchi à la paroi. Il faut noter que le pas de temps a été limité avec $DTMAX = 2.E-7$ et $KFLUX = 0.05$, en raison de la violence des phénomènes.

Résultats :

A l'arrêt du calcul, seule une analyse qualitative des résultats est proposée pour des raisons de confidentialité (voir ci-dessous) et les valeurs de référence sont des valeurs de non-régression.

Références :

La référence est confidentielle. Les données de l'explosif utilisé ont été remplacées par celles du TNT tirées de "JWL equation of state coefficients for high explosives" de E. Lee, M. Finger, W. Collins du Lawrence Livermore Laboratory (janv 1973).

Modification :

- 29 septembre 2009 :

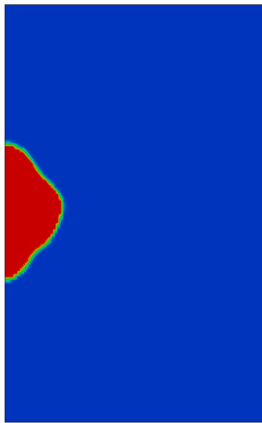
Les résultats sont différents pour la pression suite à la modification du calcul de l'amortissement quadratique et plus particulièrement du terme d'amortissement dans la direction circonférentielle en axisymétrique qui n'avait pas lieu d'être calculé (pas de choc dans cette direction).

Exécution

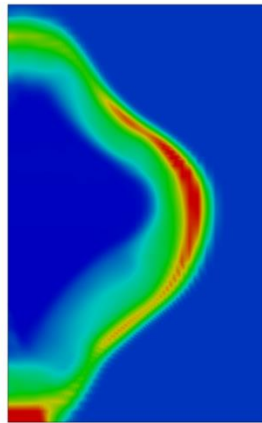
Modifications du(des) jeu(x) de données : aucune.

Remarque :

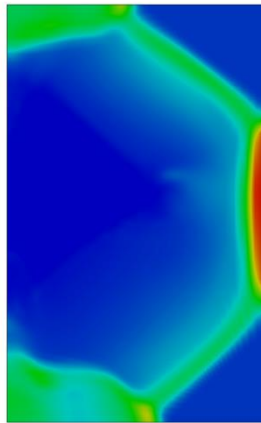
L'exécution des fichiers de post-traitement *vl_cea_jwls_2d_rend_p.epx* et *vl_cea_jwls_2d_rend_v.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :**OK : 10/10****Illustrations***Pression dans le fluide au cours du temps*

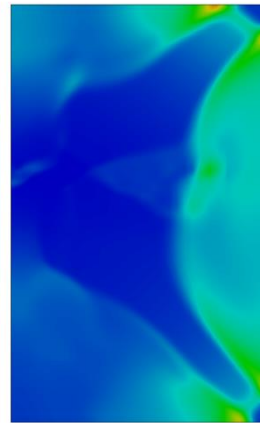
Time: 0.1 ms



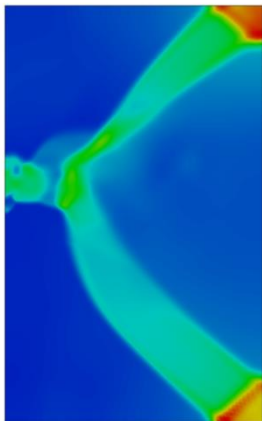
Time: 0.4 ms



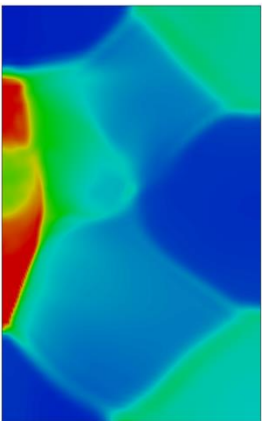
Time: 0.7 ms



Time: 1.1 ms



Time: 1.4 ms



Time: 1.8 ms

VL_CEA_MARA2

Description

Titre :

Essai MARA-2 : Simulation d'un Accident de Confinement (ADC)

Problème :

La maquette MARA est la reproduction d'un réacteur rapide à l'échelle 1/30. La cuve est partiellement remplie d'eau, surmontée par un ciel de gaz. A l'instant initial, une explosion se produit au centre de la cuve. On cherche à déterminer les déformations de la cuve et l'impulsion sur le couvercle due à l'impact du liquide.

Géométrie :

La maquette a la forme d'un caisson cylindrique de 70 cm de diamètre et de 55 cm de hauteur. Le couvercle est supposé parfaitement rigide. La virole est une coque mince. L'explosif est au centre.

Maillage :

Le maillage est 2D axisymétrique. Il y a 4455 éléments CAR1, 52 éléments TRIA pour l'eau et l'air, et 100 éléments COQU pour la cuve. Le calcul est fait en ALE, et commence avec une bulle de faible rayon à haute pression.

Matériaux :

L'explosif a été calibré pour reproduire au mieux l'accident envisagé. Les 3 zones (l'eau, l'air, la bulle) sont modélisées avec le matériau ADCR, où seules diffèrent les concentrations initiales des constituants et la pression totale. Le matériau pour la cuve est de type VMIS DYNA (Von Mises avec écrouissage dynamique), pour tenir compte des effets de la vitesse de déformation.

Conditions aux limites :

On utilise la liaison FSA pour assurer l'interaction fluide-structure.

Chargement :

Le système est au repos à l'instant initial, mais hors d'équilibre.

Calcul :

Le calcul se poursuit jusqu'à 25 ms, quand l'onde de choc s'est réfléchi sur les parois.

Résultats :

A l'arrêt du calcul, les résultats sont comparés avec les mesures relevées dans l'essai.

Référence :

EUROPLEXUS - Fiche de qualification : Essai MARA-2, P. Galon, rapport CEA DEN/SEMT/DYN/RT/02-008/A.

Exécution

Modifications du(des) jeu(x) de données : aucune.

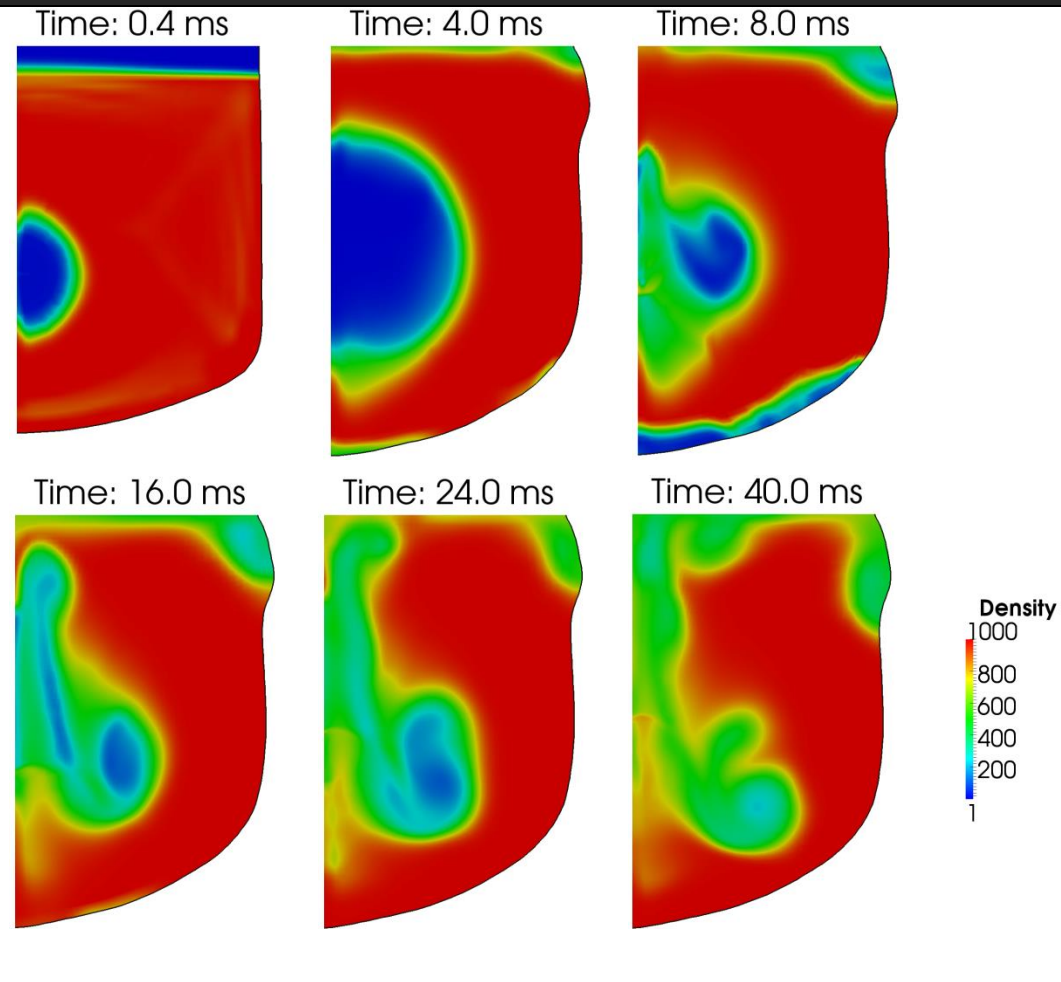
Le cas test est divisé en deux jeux de données vl_cea_mara2_calcul.epx et vl_cea_mara2_post.epx pour séparer le calcul du post-traitement.

Remarques :

L'exécution des fichiers de post-traitement vl_cea_mara2_rend_p.epx et vl_cea_mara2_rend_v.epx est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

OK : 16/16

Illustrations

VL_CEA_PHI2AS_TREILLIS

Description

Titre :

Test 1 PHI2AS : Treillis impacté par une masse

Problème :

Un treillis composé de poutres à section carrée est impacté par une masse ponctuelle. On cherche l'instant et la valeur de la flèche maximale ainsi que l'instant et la vitesse du rebond du projectile.

Géométrie :

Le treillis a la forme d'un "H" où la barre horizontale est prolongée d'une longueur égale à gauche et à droite. Tous les segments ont la même longueur $L=90$ mm. Le côté de la section carrée des poutres est de 6 mm. L'ensemble est dans le plan xOy . Le point O est au centre du treillis et les points B et C de part et d'autre de O. Le projectile de 3 kg arrive en O suivant Oz à la vitesse de 10 m/s. Les extrémités du treillis sont encastées. La pesanteur est négligée.

Maillage :

On ne maillle que la moitié du treillis en éléments de poutres POUT. Le projectile est représenté par un point matériel PMAT. Il y a 72 points et 70 éléments POUT.

Matériaux :

Le matériau du treillis est élastoplastique : $\rho = 7800$ kg/m³, $E = 2E11$ Pa, $\nu = 0.3$, limite élastique $3.2E8$ Pa, $H = 1.3465E8$ Pa (écrouissage constant, défini par une table de deux points, voir la documentation d'EUROPLEXUS).

Conditions aux limites :

Blocages aux extrémités du treillis et conditions de symétrie.

Chargement :

Le système est au repos à l'instant initial.

Calcul :

Le calcul se poursuit jusqu'à 10 millisecondes.

Résultats :

A l'arrêt du calcul, les résultats sont en bon accord avec ceux obtenus par d'autres codes dans le cadre du *benchmark* IPSI.

Références :

Forum IPSI pour la Formation et l'Information en Analyse de Structures (Phi2AS) : Codes explicites pour les collisions et chocs structuraux, Volume XIX - numéro 1 et 1bis (mars 1995).

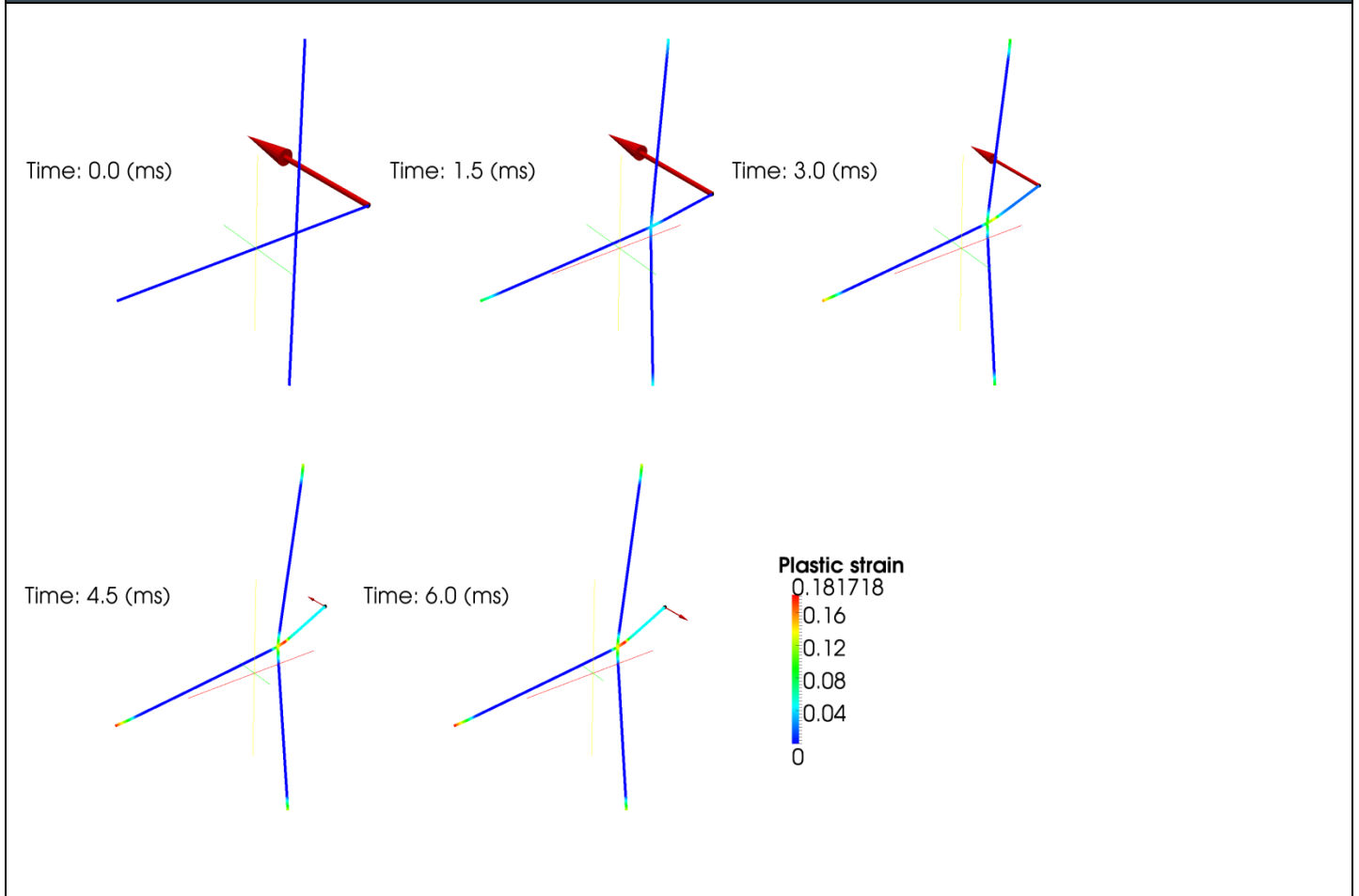
Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 3/3

Illustrations



VL_CEA_PLAQ_PERF

Description*Titre :*

Tuyau en écoulement avec une plaque perforée déformable

Problème :

On cherche à vérifier le bon comportement du couplage fluide-structure pour une plaque perforée traversée par un écoulement. La plaque perforée est encastrée dans une portion de tuyauterie, elle aussi déformable, et entraîne une perte de charge de part et d'autre.

Géométrie :

La plaque est au milieu d'une tuyauterie de 80 cm de long et 40 cm de diamètre. La plaque et les parois de la tuyauterie ont 5 mm d'épaisseur. Afin de faire des comparaisons, une deuxième plaque identique à la plaque centrale est placée à côté de la tuyauterie et est soumise à la même différence de pression.

Maillage :

Le maillage est 3D et ALE. Les structures sont des coques minces. On ne maille qu'un quart de la tuyauterie. Il y a 4998 éléments CUBE pour le fluide (5915 nœuds) et 770 éléments Q4GR (848 nœuds) pour les coques. Des éléments de conditions aux limites CL3D sont placés aux extrémités du domaine fluide.

Matériaux :

Le fluide est de l'eau initialement sous 1 bar, les parois et les plaques sont en acier avec un comportement linéaire. Une pression est imposée à l'entrée suivant une rampe qui passe de 1 bar à 3.4 bars en 3 ms. Des conditions absorbantes sont placées en sortie du fluide. L'impédance de la plaque est de 0.5 (voir le Manuel Utilisateur) introduite via un modèle GRFS (GRille avec iFS).

Un amortissement de relaxation est ajouté sur les 2 plaques pour converger plus rapidement sur la solution en régime permanent.

Conditions aux limites :

En plus des conditions de symétrie, le couplage fluide-structure sur les parois et au niveau de la plaque centrale est pris en compte. La plaque témoin est encastrée.

Chargement :

Le système est au repos à l'instant initial, et en équilibre.

La plaque témoin est soumise à une rampe de pression de 0.8 bar, solution du problème.

Calcul :

Le calcul se poursuit jusqu'à 100 ms.

Résultats :

A l'arrêt du calcul, on retrouve bien la solution analytique.

Pour plus de précision on compare avec un calcul CASTEM en grands déplacements (voir *.dgibi dans l'archive vl_cea_plaq_perf_q7.zip fournie).

Références :

Solution : $V(\text{fluide}) = (P - P_{\text{Pref}}) / (\rho * c * (1 + \alpha)) = 1.066667 \text{ m/s}$

donc : $DP(\text{plaque}) = \alpha * \rho * c * V = 0.8 \text{ bar}$

et : $Dx(\text{centre}) = 0.96077 \text{ mm}$ (petites déformations)

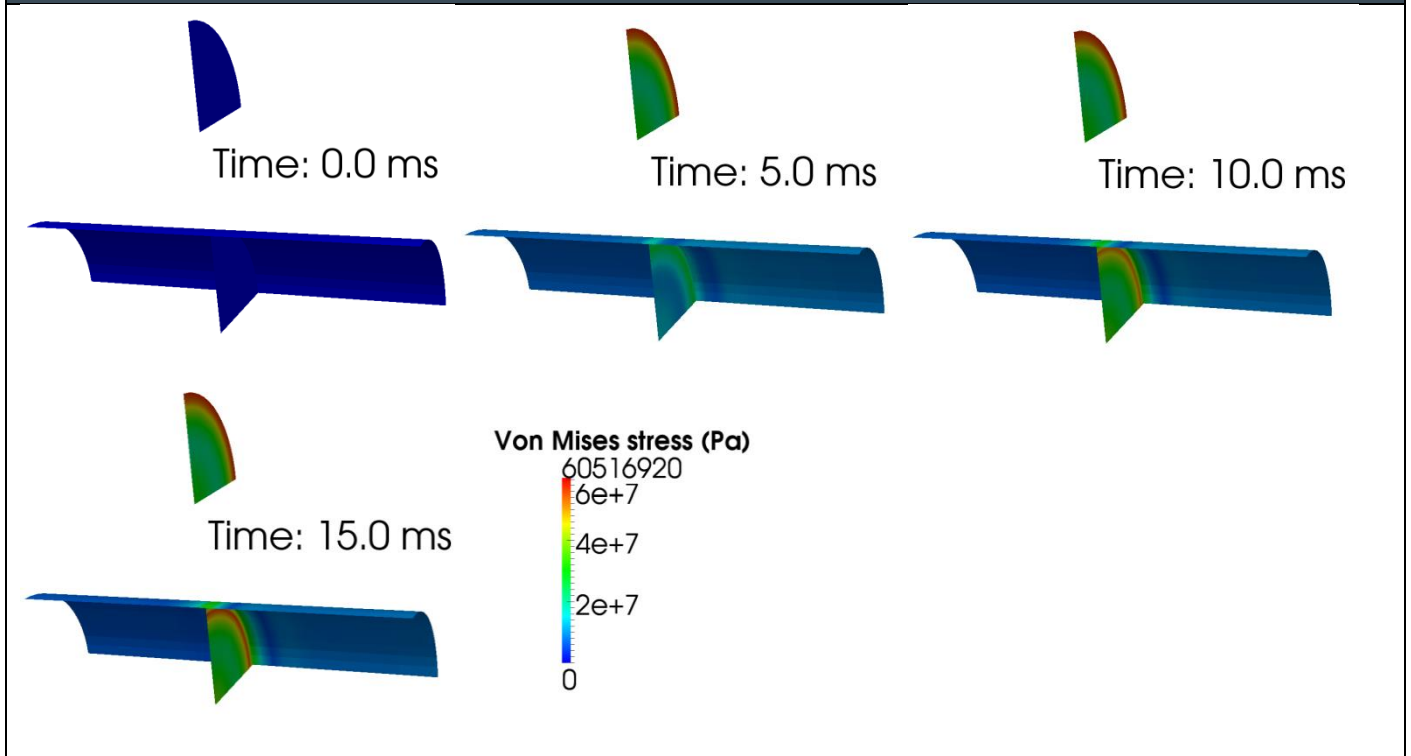
En fait un peu moins pour la plaque isolée (grandes déformations) et un peu plus pour la plaque centrale (encastrement imparfait).

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 6/6

Illustrations

VL_CEA_REAC_NAH2

Description*Titre :*

Réaction sodium-eau dans une tuyauterie

Problème :

Il s'agit d'analyser des essais réalisés en 1968-69 à Cadarache et qui ont permis de qualifier les modèles utilisés dans les codes. Une brèche est créée sur une tuyauterie d'eau sous pression et de l'eau est injectée dans une tuyauterie concentrique remplie de sodium. Quatre essais sont modélisés, numérotés de SR1 à SR4.

Géométrie :

L'installation comporte 2 tubes concentriques. Le sodium circule dans le tube externe et l'eau dans le tube interne. Ces circuits sont alimentés par des réservoirs pressurisés. A l'instant initial, le tube interne est sectionné et une réaction sodium-eau se produit.

Maillage :

Le maillage est filaire. Il est le même pour les 4 cas. Il y a 1386 éléments TUBE pour les 2 circuits.

Matériaux :

Les propriétés thermodynamiques de l'eau sont calculées au moyen des tables internes au code. Le modèle retenu est du type homogène équilibré (matériau EAU). Le sodium est modélisé au moyen du matériau RSEA. Un modèle de débit critique est utilisé pour l'injection d'eau, et des membranes de sécurité limitent la pression du circuit de sodium. Le circuit d'eau est alimenté par un réservoir sous pression.

Conditions aux limites :

Les parois sont supposées rigides, le calcul est eulérien.

Chargement :

Le système est au repos à l'instant initial, quand commence l'injection d'eau.

Calcul :

Le calcul se poursuit jusqu'à 100 ms, quand un régime stationnaire apparaît.

Résultats :

A l'arrêt du calcul, les résultats de référence sont conformes aux essais.

Références :

EUROPLEXUS - Compléments de validation du modèle de réaction sodium-eau, M. Lepareux, F. Bliard, rapport CEA/DEN/DANS/DM2S/SEMT/DYN/RT/01-029/A.

Exécution

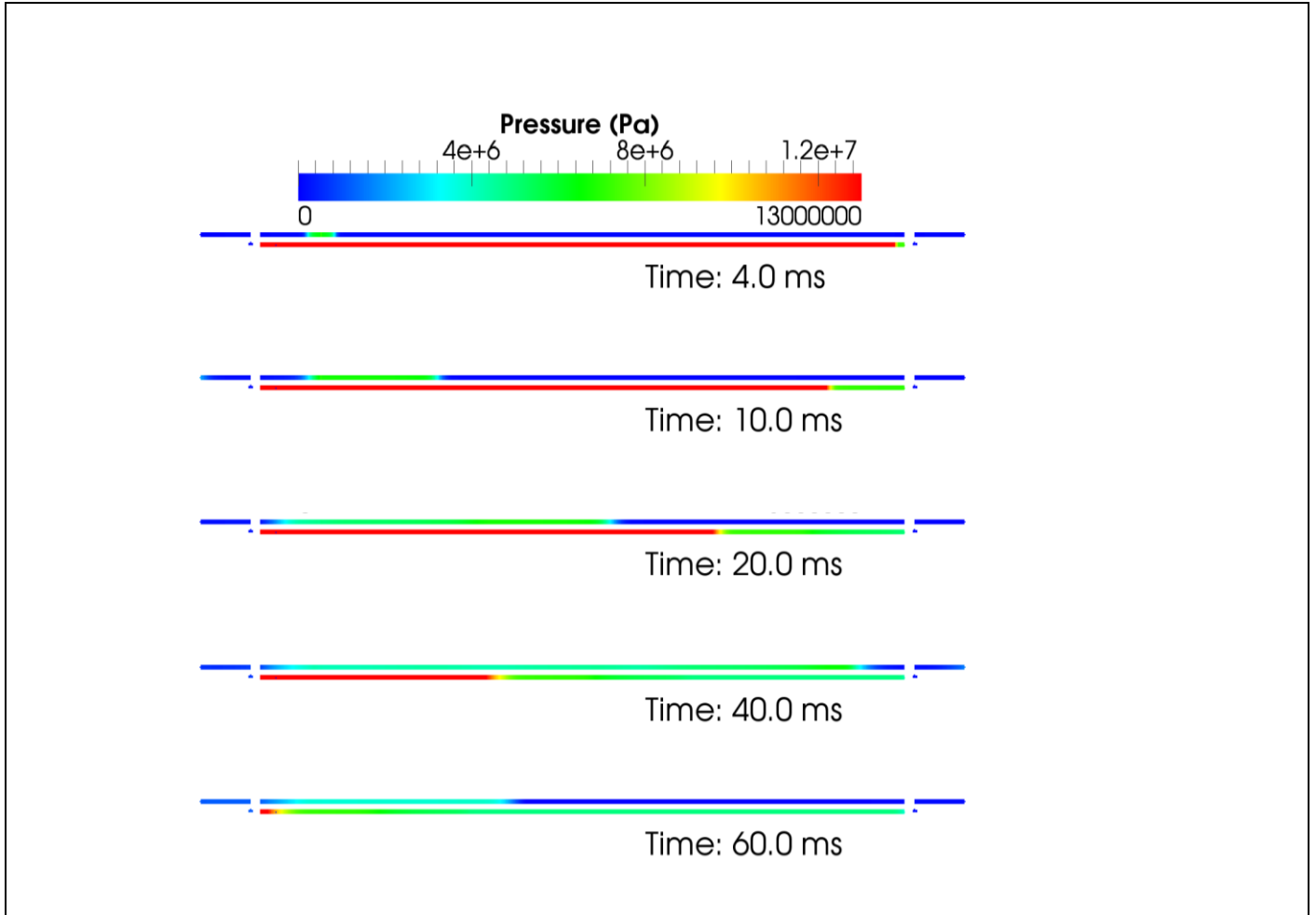
Modifications du jeu de données : aucune.

Remarque :

Le déroulement du calcul présente de nombreux messages ATTENTION pour des non-convergences dans les itérations internes du matériau EAU. C'est une situation habituelle avec ce matériau qui ne détériore pas les résultats.

Qualifications :

OK : 12/12



VL_CEA_RUR3_AIR

Description

Titre :

Essais RUR-3 : Tuyauterie impactée par un dièdre de 60°

Problème :

Une tuyauterie en acier inox posée sur un plan horizontal est impactée par une masse formant un dièdre de 60° tombant en chute libre. On cherche à déterminer l'écrasement du tuyau et la force d'impact. Ces essais ont été réalisés au CEA/CESTA en 1985.

Géométrie :

La tuyauterie a une longueur de 700 mm, un diamètre externe de 139.7 mm et une épaisseur de 2 mm. Elle est ouverte aux 2 extrémités et maintenue sur le massif par des pions verticaux.

Pour l'essai 3-1 une masse de 30 kg tombe de 3 m.

Pour l'essai 3-3 une masse de 60 kg tombe de 2.25 m.

Maillage :

On ne maillie que le quart du tuyau en éléments triangulaires DKT3. Deux points matériels PMAT permettent de modéliser le massif rigide et la masse tombante. Deux jeux de données sont proposés : *vl_cea_rur31_air.epx* pour l'essai 3-1 et *vl_cea_rur33_air.epx* pour l'essai 3-3.

Matériaux :

Le matériau du tuyau est élastoplastique : $\rho = 7800 \text{ kg/m}^3$, $E = 179\text{E}9 \text{ Pa}$, $\nu = 0.3$, limite élastique = $174\text{E}6 \text{ Pa}$. La courbe de traction a été déterminée par des essais spécifiques.

Conditions aux limites :

En plus des conditions de symétrie, le point matériel représentant le massif est bloqué.

Chargement :

Le système est au repos à l'instant initial.

Calcul :

Le calcul se poursuit jusqu'à 30 ms pour l'essai RUR 3-1 et 40 ms pour l'essai RUR 3-3.

Résultats :

A l'arrêt du calcul, les résultats de référence sont conformes aux essais.

Références :

Programme PLEXUS - Calcul d'écrasement d'une tuyauterie R.U.R. en air, M. Lepareux, J.L. Lieutenant, rapport CEA DEDR/DEMT 86.458.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque :

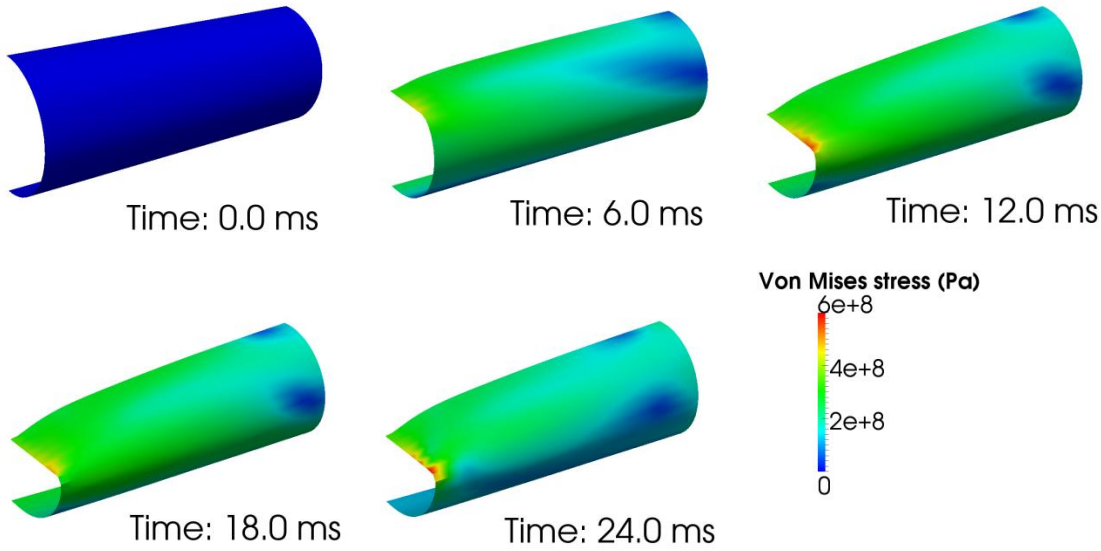
Un message ATTENTION est obtenu concernant des degrés de liberté faisant l'objet de plusieurs liaisons sans activation de l'option de calcul couplé LAGC. Cela ne perturbe pas la solution.

Qualifications :

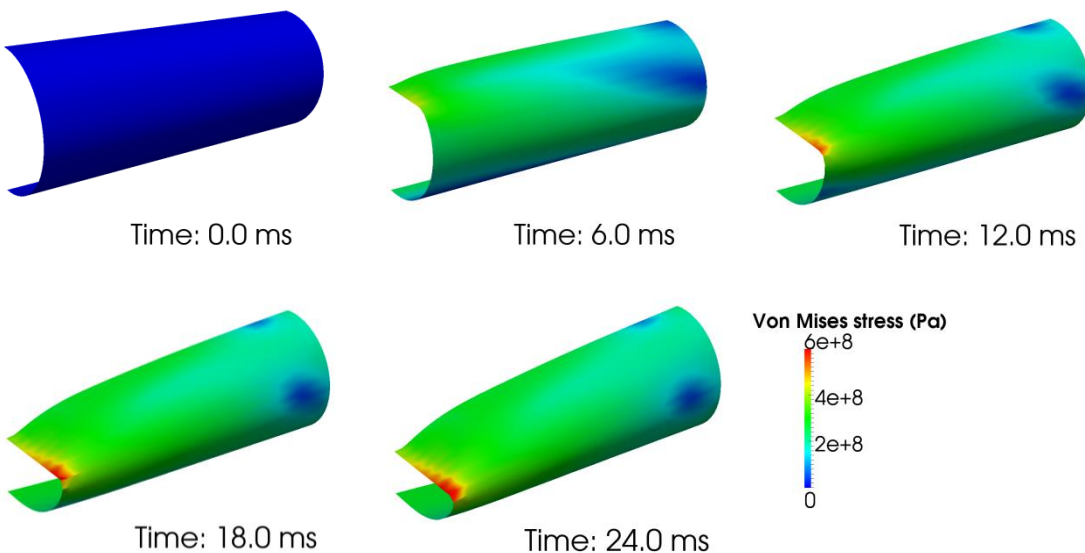
OK : 1/1 pour le cas *vl_cea_rur31_air.epx* - **OK : 1/1** pour le cas *vl_cea_rur33_air.epx*

Illustrations

Cas vl_cea_rur31_air.epx :



Cas vl_cea_rur33_air.epx :



VL_CEA_SPH_PLAQUE

Description*Titre :*

Impact d'un paquet d'eau sur une plaque carrée

Problème :

On cherche à analyser le comportement d'une plaque carrée en acier soumise à l'impact d'un paquet d'eau arrivant à grande vitesse sous un angle de 45 degrés. L'eau est modélisée au moyen de billes selon la méthode SPH.

Géométrie :

La plaque est un carré de 467 mm de côté et de 15 mm d'épaisseur. Le paquet d'eau est initialement contenu dans un prisme de base carrée de 14 cm de côté et de 40 cm de long. Il arrive sous une incidence de 45 degrés à la vitesse de 226 m/s.

Maillage :

Il y a 1369 éléments Q4GR pour modéliser la plaque et 10368 éléments de type BILLE pour l'eau. Les billes ont un diamètre de 1 cm.

Matériaux :

La plaque est en acier avec un comportement de type Von Mises isotrope VMIS ISOT. La limite élastique est de 570 MPa.

Conditions aux limites :

Les bords de la plaque sont encastres.

Chargement :

La plaque est au repos à l'instant initial et toutes les billes ont une vitesse de 226 m/s.

Calcul :

Le calcul se poursuit jusqu'à 5 ms.

Résultats :

A l'arrêt du calcul, on observe la dispersion des billes et la déformation résiduelle de la plaque.

Références :

Ce calcul est un simple test de faisabilité.

Exécution

Modifications du(des) jeu(x) de données : aucune.

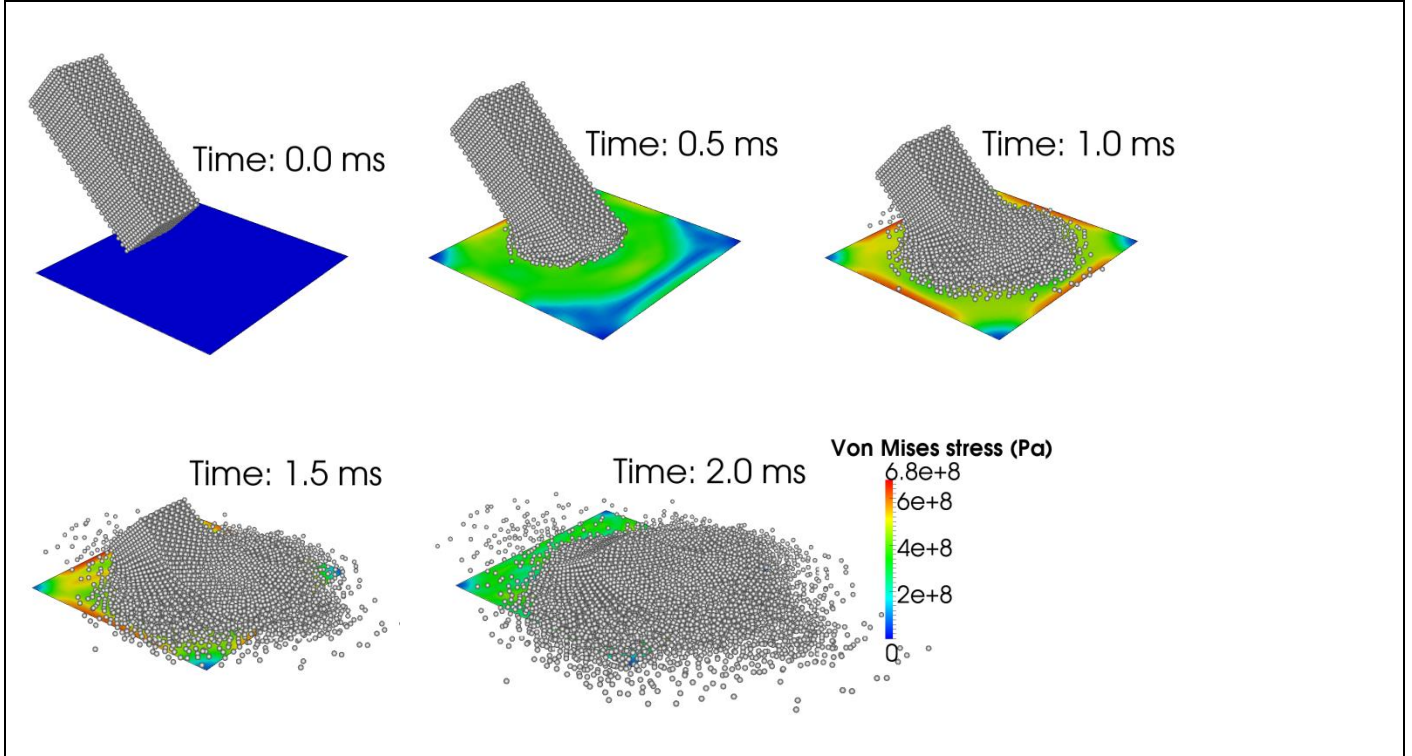
Remarque :

L'exécution du fichier de post-traitement `vl_cea_sph_plaque_rend.epx` est ignorée, car il est destiné à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Il n'apporte aucune information de qualification supplémentaire.

Qualifications :

OK : 1/1

Illustrations



VL_CEA_TCHOC_123D

Description

Titre :

Tube à choc en gaz parfait (1D, 2D, 3D) et en gaz de Van der Waals (1D)

Problème :

On cherche à vérifier le bon comportement des éléments 1D filaires (TUBE), pour un gaz parfait et un gaz de Van der Waals (1D), ainsi que pour les éléments 2D (CAR1, TRIA) et 3D (CUBE, PRIS). De plus, en 2D, on vérifie également les éléments de type volumes finis VFCC (Q4VF et T3VF).

On considère un tube à choc de 1 m de long séparé en 2 zones égales, contenant de l'air ou de l'argon. Une zone est à la pression normale (BP) et l'autre à haute pression (HP). Les 2 zones sont à la même température. A l'instant initial, la cloison séparant les 2 zones disparaît.

Maillage :

Le tube à choc est maillé avec 400 éléments de type TUBE (1D). En 2D, il y a 600 éléments égaux (1200 pour les triangles) et 2 couches en hauteur. En 3D, il y a 400 éléments égaux (800 pour les prismes) et 1 seule couche. Cinq jeux de données sont proposés : *vl_cea_tchoc_1d_01.epx*, *vl_cea_tchoc_1d_02.epx*, *vl_cea_tchoc_2d_01.epx*, *vl_cea_tchoc_2d_02.epx* et *vl_cea_tchoc_3d.epx*.

Matériaux :

Le gaz est de l'air ou de l'argon, les deux compartiments sont à la même température proche des conditions normales. Le gaz à haute pression est à 10 bars et celui à basse pression à 1 bar.

Conditions aux limites :

Les parois sont fixes.

Chargement :

Le système est au repos à l'instant initial, mais hors d'équilibre.

Calcul :

Le calcul se poursuit jusqu'à 0.9 ms, avant que l'onde de choc n'atteigne l'extrémité du tube.

Résultats :

A l'arrêt du calcul, les résultats de référence sont corrects, et proches de la solution analytique pour le cas en gaz parfait.

Cette solution analytique est calculée avec CAST3M (voir *vl_cea_tchoc_1d.dgibi* dans l'archive *vl_cea_tchoc_123d.zip* fournie) en même temps que les points des courbes théoriques de référence.

Références :

Europlexus - Fiche de qualification CIR_TCHOC_GP, M. Lepareux, rapport CEA DMT/SEMT/DYN/RT/2000-029/A.

Exécution

Modifications du(des) jeu(x) de données : les éléments de type CVL1 et TVL1 sont remplacés par les éléments Q4VF et T3VF (nouvelle formulation pour les volumes finis de type VFCC) dans le jeu de données *vl_cea_tchoc_2d_02.epx*

Qualifications :

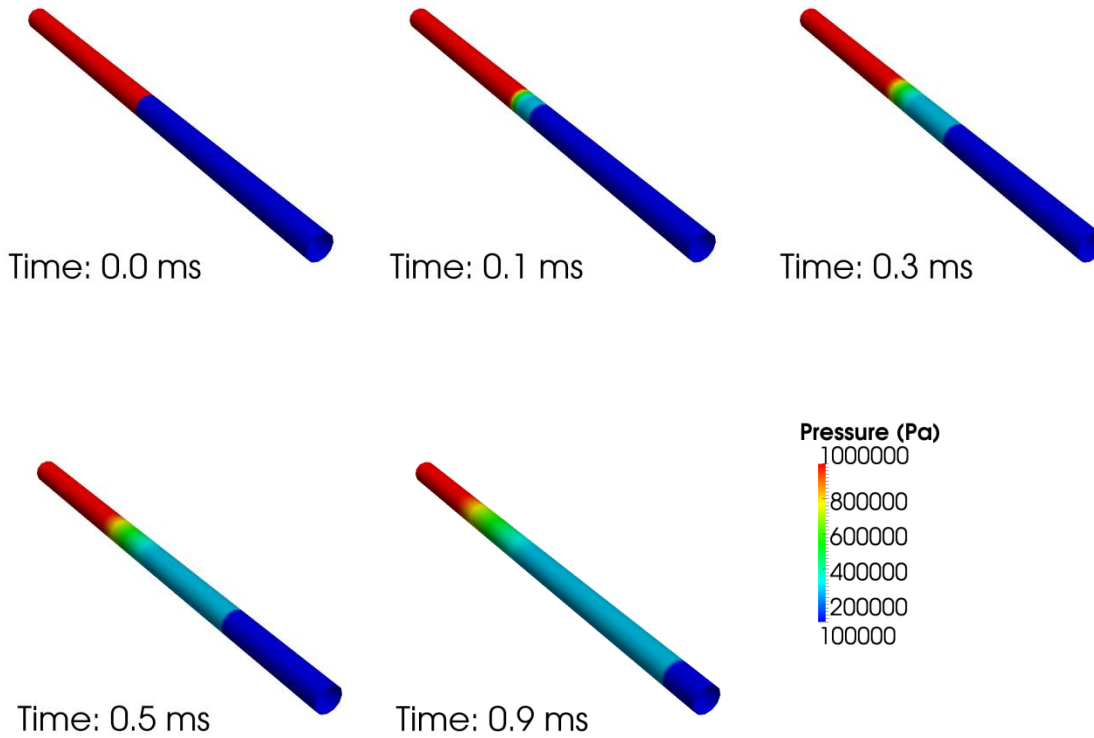
OK : 9/9 pour le cas *vl_cea_tchoc_1d_01.epx* - **OK : 9/9** pour le cas *vl_cea_tchoc_1d_02.epx*

OK : 18/18 pour le cas *vl_cea_tchoc_2d_01.epx* - **OK : 14/14** pour le cas *vl_cea_tchoc_2d_02.epx*

OK : 18/18 pour le cas *vl_cea_tchoc_3d.epx*

Illustrations

Cas vl_cea_tchoc_1d.epx :



VL_CEA_TE_DISSYM

Description

Titre :

Réactions sur les branches d'un TE dissymétrique sous pression interne.

Problème :

On cherche à vérifier le bon comportement des bifurcations 1D en liaison avec les éléments de tuyauteries filaires. On considère ici un "T" à angle droit dans le plan xOy, encastré aux 3 extrémités, rempli de liquide sous pression.

On observe le déplacement du T et les réactions aux encastremets.

Maillage :

Les 3 branches sont droites et ont une longueur de 5 m. La branche 1 suivant Ox a un diamètre externe de 457 mm, la branche 2 suivant Ox a un diamètre externe de 508 mm et la branche 3 suivant Oy a un diamètre externe de 457 mm. Les épaisseurs des tuyauteries sont identiques (7 mm). Les branches se rejoignent en un TE maillé par un élément BIFU. Le maillage comprend 50 éléments TUYA par branche.

Matériaux :

La paroi est en acier avec un comportement élastique. Le fluide interne est de l'eau initialement sous 1 bar. Une pression imposée de 40 bars est appliquée au fluide interne.

Conditions aux limites :

L'extrémité des branches 1 et 2 est bloquée (structure et fluide). L'extrémité de la branche 3 est bloquée pour la structure uniquement.

Chargement :

La pression interne passe de 0 à 40 bars à l'instant initial puis reste constante. Afin d'atteindre la solution statique, un amortissement de relaxation a été ajouté (OPTI AMOR QUASI STATIQUE 14.2 0.9).

Calcul :

On s'intéresse à l'évolution du système pendant les 150 premières ms. Pour cela, on trace les déplacements du TE ainsi que les forces et moments aux 3 extrémités bloquées.

Deux calculs sont réalisés :

- a) sans prendre en compte le poids propre (pas de pesanteur) : vl_cea_te_dissym_01.epx
- b) avec le poids propre : vl_cea_te_dissym_02.epx

Résultats :

Les résultats sont comparés avec ceux de CAST3M dans la même configuration. Les deux codes donnent des résultats pratiquement identiques, aux différences de modélisation près.

Référence :

Europlexus - Fiche de qualification "cir_te_dissym", M. Lepareux, rapport CEA SEMT/DYN/RT/2001-010/A.

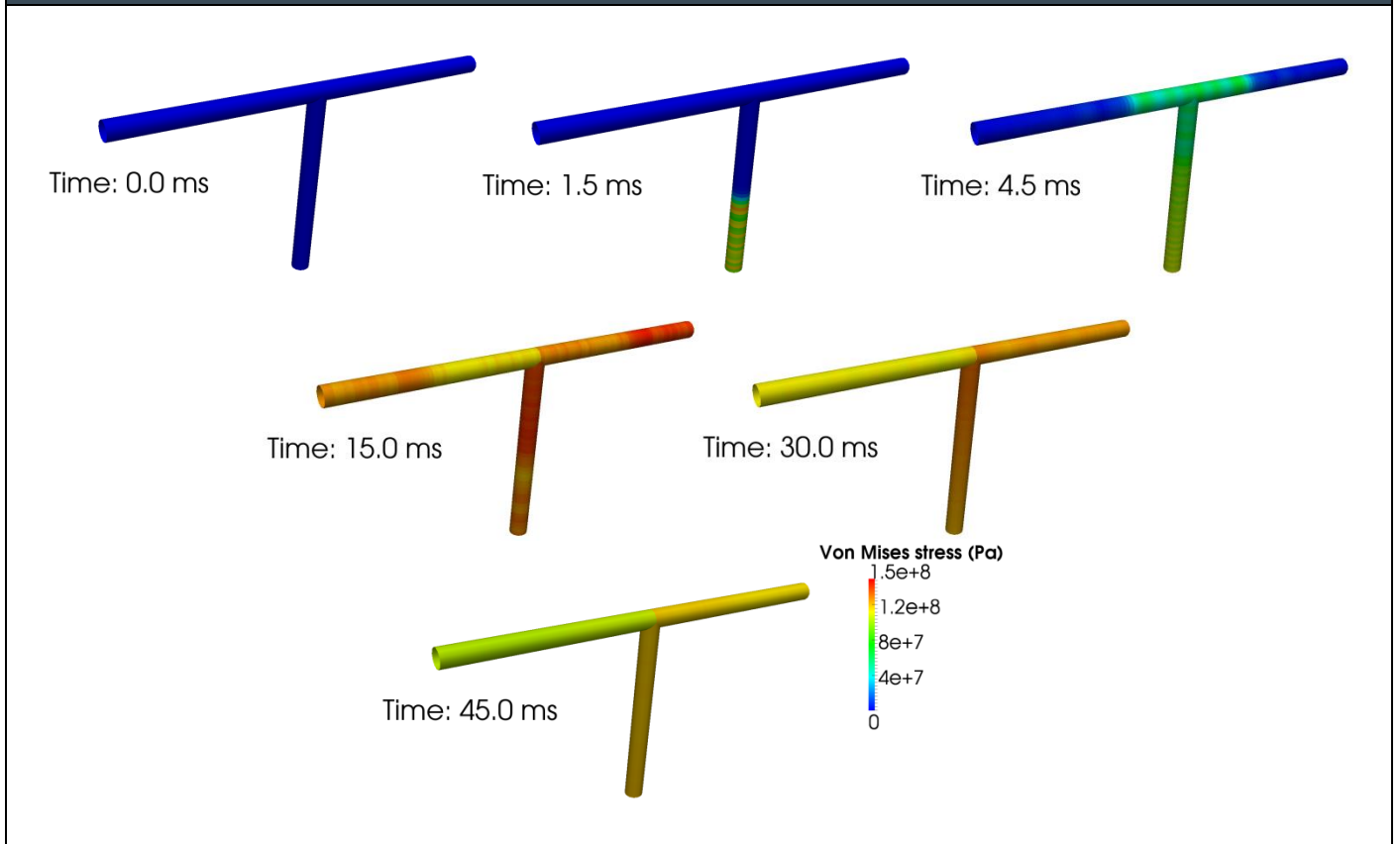
Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 26/26

Illustrations



VL_CEA_TUBM_GRFS

Description*Titre :*

Raccord Filaire-Massif avec une plaque perforée déformable

Problème :

On cherche à vérifier le bon comportement des raccords de maillage en filaire (1D) et massif (3D) ainsi que du couplage fluide structure pour une plaque perforée traversée par un écoulement. La comparaison est faite avec un cas filaire pur, un cas filaire avec un raccord BIFU et un cas mixte filaire massif avec 2 raccords TUBM.

Géométrie :

Le tube est droit et composé de 3 segments de 50 cm. Le diamètre est de 15.97 cm pour avoir une section de 0.02 m². La plaque est au milieu de la section centrale. Elle a 2 mm d'épaisseur et est encastrée sur sa circonférence. Le taux de perçage est tel que le coefficient d'impédance est $\alpha=0.06$ (voir le Manuel Utilisateur).

Maillage :

Il y a 3 maillages différents pour un seul calcul :

a/ tout est filaire avec 21 éléments de type TUBE,

b/ tout est filaire et la section centrale est raccordée avec 2 BIFU,

c/ la section centrale est en 3D, dont on ne maillie que le quart, et est raccordée avec 2 TUBM aux 2 autres sections qui restent filaires.

La plaque est modélisée avec un élément CL1D (matériau IMPE GRIL) dans les 2 premiers cas et avec un ensemble de coques DKT3 et d'éléments CL3D (matériau IMPE GRFS) dans le dernier cas. Il y a 4116 éléments CUBE pour le fluide (4901 nœuds) pour le 3D.

Les parois sont fixes et non représentées. Le calcul est ALE. Des éléments de conditions aux limites CL1D sont placés aux extrémités dans les 3 cas.

Matériaux :

Le fluide est de l'eau initialement sous 10 bars, la plaque est en acier avec un comportement linéaire. La pression de 10 bars est maintenue à l'entrée, et un diaphragme est placé à la sortie, qui s'ouvre sur la pression externe de 1 bar en 10 ms. Le diaphragme a un coefficient de perte de charge $K=100$.

Conditions aux limites :

En plus des conditions de symétrie, il y a un couplage fluide structure sur la plaque centrale.

Chargement :

Le système est au repos à l'instant initial, et en équilibre.

Calcul :

Le calcul se poursuit jusqu'à 200 ms.

Résultats :

A l'arrêt du calcul, la solution de référence reproduit bien la solution analytique pour les 3 configurations. On compare également avec un calcul CAST3M (voir *.dgibi dans l'archive *vl_cea_tubm_grfs_t7.zip* fournie).

Référence :

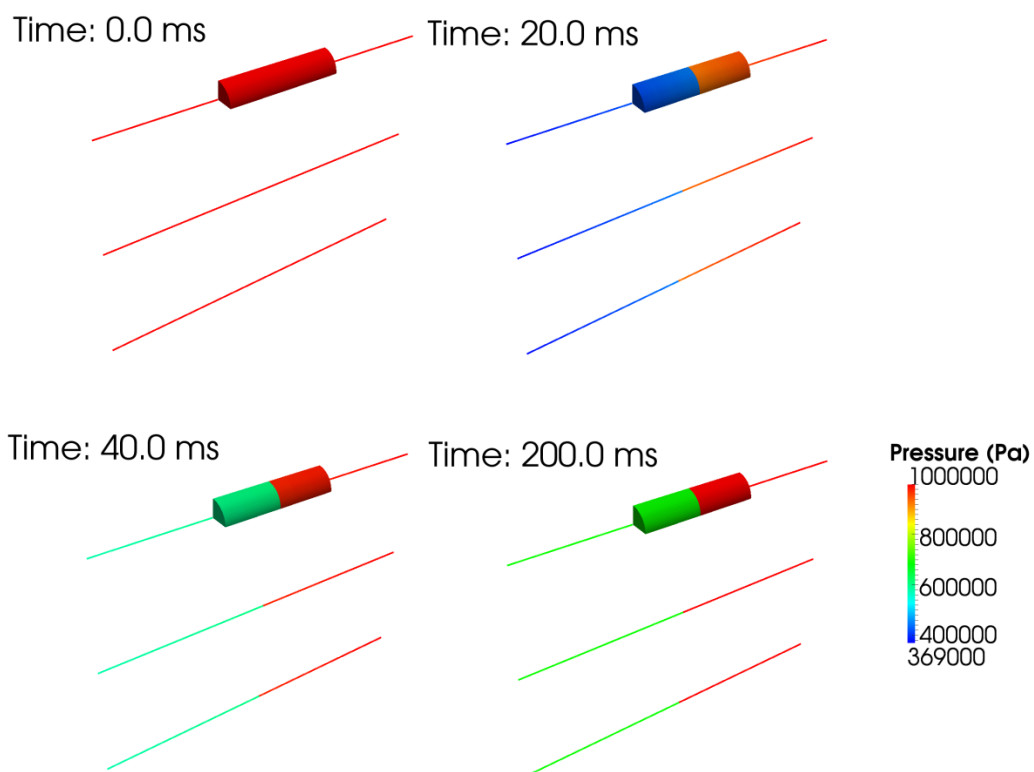
La solution est donnée dans *.lst, et les formules dans *.dgibi (cf. archive *vl_cea_tubm_grfs_t7.zip*).

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 7/7

Illustrations

VL_CEA_VISCO

Description*Titre :*

Écoulement laminaire d'un liquide entre 2 plans parallèles

Problème :

On cherche à vérifier le bon comportement de la viscosité pour un liquide. Pour cela on impose un gradient de pression dans un tube formé par deux plans parallèles.

Maillage :

Le maillage est 2D plan et eulérien. Les dimensions du tube sont de 10 mm de long et 8 mm de hauteur. On ne maille qu'un demi-tube avec 6 éléments dans une demi-hauteur. Il y a 112 nœuds et 90 éléments CAR1, ainsi que 12 éléments CL2D pour les conditions aux limites.

Matériau :

Le fluide est de l'eau sous 1 bar. A l'entrée, la pression imposée est de 1.1 bar appliquée en 1 ms selon une rampe, et reste à 1 bar en sortie. La viscosité a été fortement augmentée pour atteindre rapidement un régime permanent : $\mu = 0.4$ (eau : $\mu = 1E-3$).

Conditions aux limites :

Les vitesses sont bloquées à la paroi et les conditions de symétrie sont appliquées sur l'axe.

Chargement :

Le système est au repos à l'instant initial, et en équilibre.

Calcul :

Le calcul se poursuit jusqu'à 150 ms.

Résultats :

A l'arrêt du calcul, la solution de référence présente un profil parabolique pour les vitesses, et les résultats sont en bon accord avec la solution analytique.

Référence :

Solution : $V(\text{axe}) = -0.5 * (h * h / \mu) * (dP/dX)$.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque :

L'exécution du fichier de post-traitement *vl_cea_visco_rend_v.epx* est ignorée, car il est destiné à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version testée.

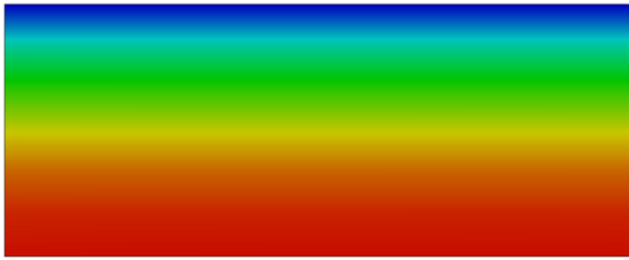
Qualifications :

OK : 2/2

Illustrations

Champs de vitesse stationnaire atteint :

Time: 150.0 ms



Velocity (m/s)



VL_EDF_AA7075_BALLISTIC_TEST_MPI

Description*Title:*

Ballistic impact test for 7075-T651 aluminum alloy thin plate.

Problem:

A high strain rate material behavior of 7075-T651 aluminum alloy extensively used for aircraft metallic profiles is modeled here with EUROPLEXUS and results are compared against ballistic impact tests conducted by the Korea Advanced Institute of Science and Technology [1].

Test description:

Ballistic tests were performed with a pneumatic gun projecting an impactor composed of a 36 mm diameter-15 mm length steel cylinder and a 40 mm diameter-80 mm length polycarbonate cylinder, into a 7075-T651 aluminum alloy square plate target. Velocity of the impactor is determined thanks to a high-speed camera Phantom V711. Results are recorded at 20 000 fps for resolution of 1088 by 400. The plate thickness is 5 mm and the plate side size is 400 mm. The plate is clamped on its perimeter.

Mesh:

The plate is discretized with Q4GS (quadrangle) and T3GS (triangle) shell elements. The smallest size of shell elements is found in the contact area and equal to 1 mm. The impactor is meshed with CUBE elements.

Materials:

Only the steel part of the impactor is modeled. The polycarbonate part is taken into account as a modification of the density turning the steel density from 7200 kg/m³ to 15719 kg/m³. An elastic behavior LINE is applied to the steel impactor. The aluminum alloy 7075 behavior of the plate is modeled with a Johnson-Cook law (VMJC) of EUROPLEXUS. For both shell elements Q4GS and T3GS considered here, the element thickness variation is activated through the LMST option. Plate elements erosion driven by the Johnson-Cook failure criterion is activated in all calculations.

Boundary conditions:

The four sides of the plate are clamped. The contact between the plate and the impactor is represented with LINK GLIS master/slave surface method. The COPT contact option allows to take into account the thickness of the plate.

Initial conditions:

Initial velocity 187.4 m/s is applied to the impactor.

Calculation:

The calculation is performed up to 0.1 ms.

Results:

The residual velocity is compared to the experimental value.

Reference :

[1] Jae-Wook Jung, Sang Lee Eon, and Jung-Wuk Hong. Experimental and Numerical Investigations of High-Speed Projectile Impacts on 7075-T651 Aluminum Plates. Materials, 2019. URL: <http://www.sciencedirect.com/science/article/pii/S0749641902000190>, doi:10.3390/ma12172736.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque : Le test est exécuté uniquement en MPI.

Qualifications : **OK : 1/1**

VL_EDF_ALU_TUBE_IMPACT

Description*Title:*

Axial impact of a 2024-T3 aluminium tube on a rigid steel slab.

Problem:

This case simulates an experiment [1] where an aluminium A2024-T3 tube impacts on a rigid steel slab.

Test description:

A tube-like cylindrical shell with an attached steel mass was launched against a massive steel slab. Despite the high impact velocity, the tubes were not damaged during impact, only a slight axisymmetric buckling mode was observed.

Mesh:

The slab and the mass are discretized with CUBE elements whereas the cylindrical tube is meshed with Q4GS shells. To simulate different interactions, two contact zones are defined: between the tube's bottom and the slab, and between the tube's upper end and the mass. A 2 mm element mean size is specified for the meshing.

The steel cylinder is modelled here as a rectangular slab of limited dimensions (100mm x 100mm x 4mm). To ensure that the slab model is rigid, the nodes situated on the face opposite to the impact are blocked. The length of the mass is chosen to keep the total mass of 127 g.

Materials:

For the steel slab and the steel attached mass the LINE elastic material is used. For the cylindrical shell, a Johnson-Cook material model (VMJC) is employed.

Boundary conditions:

The lower face of the steel slab is blocked (LINK BLOQ). Contact conditions (LINK GLISS) are prescribed between the slab and the lower end of the tube and between the upper end of the tube and the mass.

Initial conditions:

An initial vertical velocity of -117.35 m/s is imposed to the tube and the mass.

Calculation:

The calculation is performed up to 0.5 ms.

Results:

Numerical and experimental results are compared for the tube shortening and the impact duration.

Reference :

[1] A.L. Florence and J.N. Goodier, Dynamic Plastic Buckling of Cylindrical Shells in Sustained Axial Compressive Flow, Journal of Applied Mechanics, 1, vol.35, 1968, pp. 80-86.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque :

This calculation has been performed with (vl_edf_Alu_tube_impact_mpi.epx) and without MPI (vl_edf_Alu_tube_impact.epx).

Qualifications :

OK : 1/1

VL_EDF_AMORT_RAYLEIGH

Description*Title:*

Plate subjected to an impulse-type and step-type loading with Rayleigh's damping.

Problem:

Two calculations have been performed:

- ➔ Plate subjected to an impulse-type loading with Rayleigh's damping (vl_edf_amort_rayleigh_impv.epx)
- ➔ Plate subjected to a step-type loading with Rayleigh's damping (vl_edf_amort_rayleigh_step.epx)

This test case validates the Rayleigh's damping. A square plate 50m large and 2m thick clamped on its four edges is loaded by a uniform pressure applied to a square area of 20m side at the center of the plate. The bending response is studied.

Mesh:

The mesh is composed of 3-node T3GS thick shell elements.

Material:

Linear elastic material law is used.

Boundary conditions:

The plate is clamped on its four edges.

Loading:

A uniform pressure is applied to a square area of 20m side at the center of the plate. An impulse-type loading is considered with 1MPa peak value.

Damping:

5% Rayleigh's damping in the range from 5 to 40 Hz is applied.

Calculation:

The calculation is performed up to 1.0 s.

Results:

The reference solution is obtained with Code_Aster (DYNA_LINE_TRAN).

Qualifications in vl_edf_amort_rayleigh_impv.epx:

At $t=1.517E-1s$, the vertical displacement at the center of the plate is verified.

At $t= 8.46E-2s$, the vertical velocity at the center of the plate is verified.

Qualifications in vl_edf_amort_rayleigh_step.epx:

At $t=9.67E-2s$, the vertical displacement at the center of the plate is verified.

At $t=1.528E-1s$, the vertical velocity at the center of the plate is verified.

Reference:

This test case is completely described in EDF R&D/AMA technical report HT62/06/008 intitled "Introduction de l'amortissement de Rayleigh dans le code de dynamique rapide EUROPLEXUS" [Introduction of Rayleigh damping in EUROPLEXUS fast dynamics software].

Exécution

Modifications du(des) jeu(x) de données :

Aucune.

Qualifications :

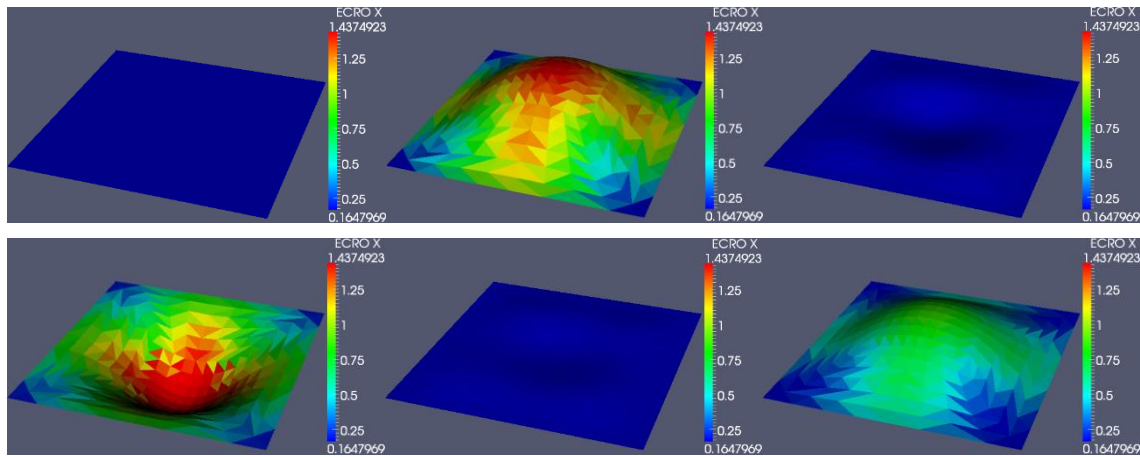
OK : 2/2 for vl_edf_amort_rayleigh_impv.epx

OK : 2/2 for vl_edf_amort_rayleigh_step.epx

Illustrations

Cas du test vl_edf_amort_rayleigh_imp_u.epx:

Champ de pression dans la plaque (0.05 s entre chaque image, déplacement x 30 000)



VL_EDF_ARLQ1_1ID

Description

Titre :

Flexion d'une dalle encastrée-libre avec un collage Arlequin

Configuration :

```
          3D          plaque | F(t)
/-----          ----- v
/|                    -----
/-----
          <-->
          zone de
          collage
          Arlequin
```

Maillage :

Deux jeux de données sont proposés :

Le jeu de données "vl_edf_arlq1_1id_cub8.epx" utilise des CUB8 pour la partie 3D volumique et Q4GS pour la partie plaque.
Le jeu de données "vl_edf_arlq1_1id_cube.epx" utilise des CUBE pour la partie 3D volumique et Q4GS pour la partie plaque.

Matériau :

Le matériau est élastique linéaire de type acier.

Liaisons :

La moitié gauche de la dalle est modélisée en 3D volumique alors que la moitié droite est modélisée en plaques. Le collage Arlequin est prescrit sur la zone de recouvrement de deux modèles à l'aide du mot-clé ARLQ ROTA de la directive LINK COUP. Le collage se fait sur une zone de recouvrement rectiligne.

Chargement :

La dalle est chargée par une force linéique verticale appliquée à l'extrémité libre de la dalle.

Calcul :

Le calcul se poursuit jusqu'à 140 ms.

Résultat :

A l'instant final du calcul on vérifie la solution Arlequin (flèche à l'extrémité libre) par rapport à la solution du modèle tout plaque.

Référence :

Ce test est complètement décrit dans la référence H-T61-2015-04524-FR.

Exécution

Modifications du(des) jeu(x) de données :

Aucune.

Qualifications :

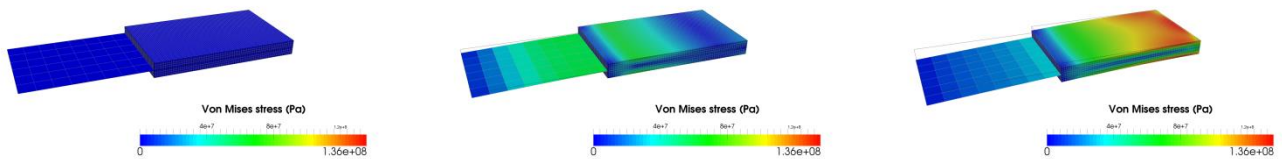
OK : 2/2 pour test vl_edf_arlq1_1id_cub8.epx

OK : 2/2 pour test vl_edf_arlq1_1id_cube.epx

Illustrations

Cas du test vl_edf_arlq1_1id_cub8.epx :

Déformée et contrainte de Von Mises dans le modèle tous les 0.07 s (déplacement amplifié d'un facteur 5)

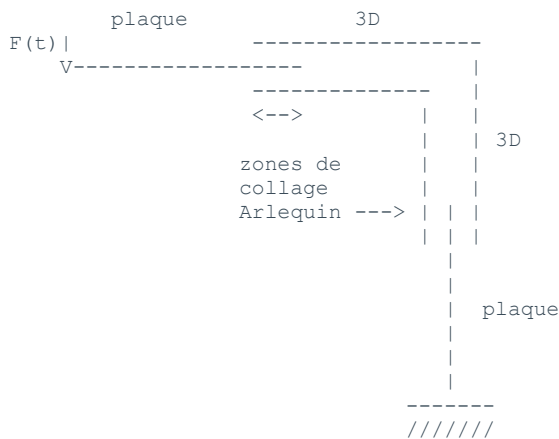


VL_EDF_ARLQ2_2ID

Description

Titre :
Flexion d'un coude encastré sur un bord avec un collage Arlequin

Configuration :



Maillage :

Deux jeux de données sont proposés :
Le jeu de données "vl_edf_arlq2_2id_cub8.epx" utilise des CUB8 pour la partie 3D volumique et Q4GS pour la partie plaque.
Le jeu de données "vl_edf_arlq2_2id_cube.epx" utilise des CUBE pour la partie 3D volumique et Q4GS pour la partie plaque.

Matériau :

Le matériau est élastique linéaire de type acier.

Liaisons :

La partie du coude dans l'angle est modélisée en 3D volumique alors que les deux extrémités sont modélisées en plaques. Le collage Arlequin est prescrit sur deux zones de recouvrement à l'aide du mot-clé ARLQ de la directive LINK COUP. Le collage se fait sur deux zones de recouvrement rectilignes.

Chargement :

La structure est chargée par une force linéique verticale appliquée à l'extrémité libre du coude.

Calcul :

Le calcul se poursuit jusqu'à 140 ms.

Résultat :

A l'instant final du calcul on vérifie la solution Arlequin (flèche à l'extrémité libre) par rapport à la solution du modèle tout plaque.

Référence :

Ce test est complètement décrit dans la référence H-T61-2015-04524-FR.

Exécution

Modifications du(des) jeu(x) de données : Aucune.

Qualifications :

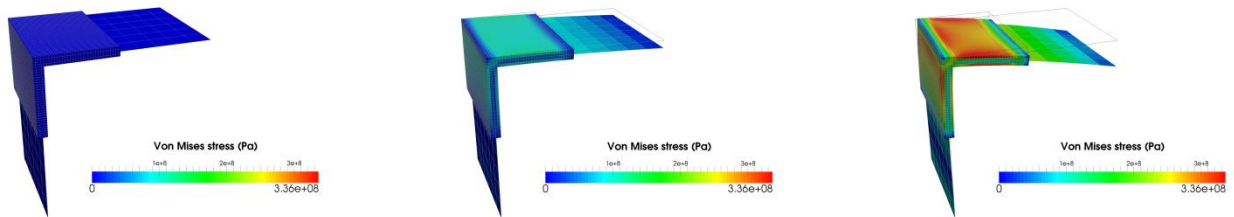
OK : 2/2 pour test vl_edf_arlq2_2id_cub8.epx

OK : 2/2 pour test vl_edf_arlq2_2id_cube.epx

Illustrations

Cas du test vl_edf_arlq2_2id_cub8.epx :

Déformée et contrainte de Von Mises dans le modèle tous les 0.07 s (déplacement amplifié d'un facteur 3)



VL_EDF_ARLQ3_1IC

Description

Titre :

Dôme encastré soumis à une pression avec un collage Arlequin

Configuration :

Il s'agit d'une calotte sphérique encastrée sur son pourtour et chargée par une pression sur sa partie centrale.

Maillage :

Deux jeux de données sont proposés :

Le jeu de données "vl_edf_arlq3_1ic_cub8.epx" utilise des CUB8 pour la partie 3D volumique (partie centrale) et Q4GS pour la partie périphérique modélisée en plaque.

Le jeu de données "vl_edf_arlq3_1ic_cube.epx" utilise des CUBE pour la partie 3D volumique (partie centrale) et Q4GS pour la partie périphérique modélisée en plaque.

Matériau :

Le matériau est élastique linéaire de type acier.

Liaisons :

Le collage Arlequin est prescrit sur une zone de recouvrement en ligne courbe fermée. Les deux modèles sont liés à l'aide du mot-clé ARLQ de la directive LINK COUP.

Chargement :

Le dôme est chargé par une pression appliquée sur la partie centrale autour de l'axe de symétrie du dôme.

Calcul :

Le calcul se poursuit jusqu'à 50 ms.

Résultat :

A l'instant final du calcul on vérifie la solution Arlequin (flèche au point situé sur l'axe vertical) par rapport à la solution du modèle tout plaque.

Référence :

Ce test est complètement décrit dans la référence H-T61-2015-04524-FR.

Exécution

Modifications du(des) jeu(x) de données :

Aucune.

Qualifications :

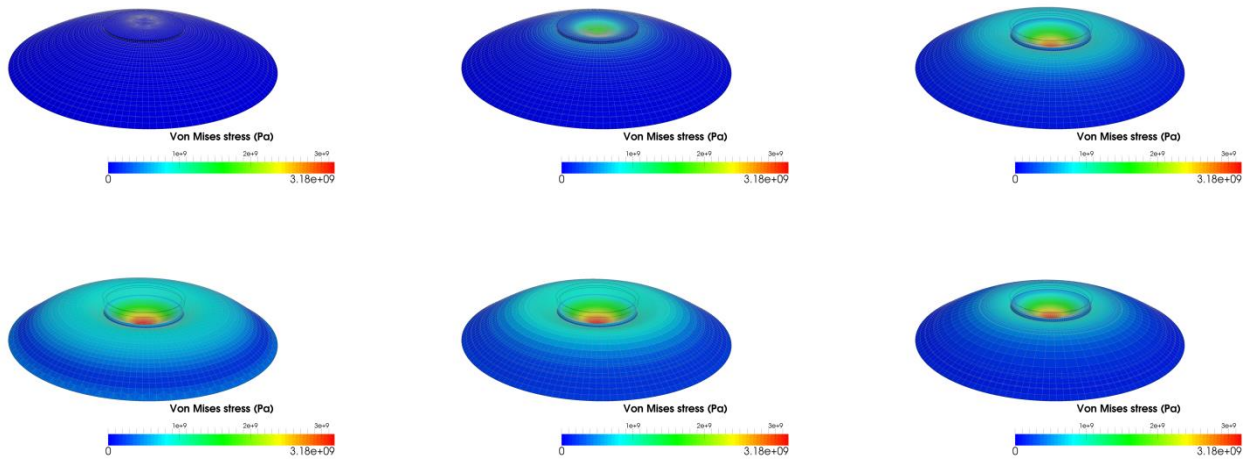
OK : 2/2 pour test vl_edf_arlq3_1ic_cub8.epx

OK : 2/2 pour test vl_edf_arlq3_1ic_cube.epx

Illustrations

Cas du test vl_edf_arlq3_1ic_cub8.epx :

Déformée et contrainte de Von Mises dans le modèle tous les 0.01 s (déplacement amplifié d'un facteur 10)



VL_EDF_ARLQ4_3IB

Description

Titre :

Bâtiment soumis à une pression avec un collage Arlequin

Configuration :

Le bâtiment de forme cubique encastré sur sa base est soumis à une pression appliquée sur trois plans de la zone 3D.

Maillage :

Deux jeux de données sont proposés :

Le jeu de données "vl_edf_arlq4_3ib_cub8.epx" utilise des CUB8 pour la partie 3D volumique et Q4GS pour la partie plaque.

Le jeu de données "vl_edf_arlq4_3ib_cube.epx" utilise des CUBE pour la partie 3D volumique et Q4GS pour la partie plaque.

Matériau :

Le matériau est élastique linéaire de type acier.

Liaisons :

Un des angles du cube est modélisé en 3D volumique alors que le reste est modélisé en plaques. Le collage Arlequin est prescrit sur une ligne brisée passant par trois murs du bâtiment. Le collage est spécifié à l'aide du mot-clé ARLQ de la directive LINK COUP. Les éléments situés sur les arêtes du cube sont exclus du collage.

Chargement :

Une pression est appliquée sur trois plans de la zone 3D.

Calcul :

Le calcul se poursuit jusqu'à 140 ms.

Résultat :

A l'instant final du calcul on vérifie la solution Arlequin (flèche à l'extrémité libre) par rapport à la solution du modèle tout plaque.

Référence :

Ce test est complètement décrit dans la référence H-T61-2015-04524-FR.

Exécution

Modifications du(des) jeu(x) de données :

Aucune.

Qualifications :

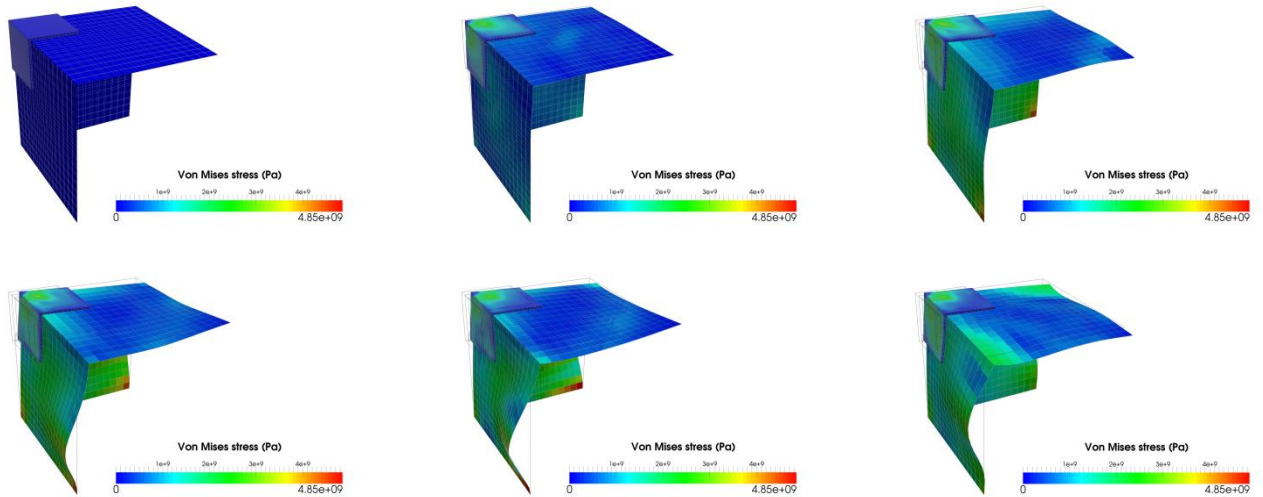
OK : 2/2 pour test vl_edf_arlq4_3ib_cub8.epx

OK : 2/2 pour test vl_edf_arlq4_3ib_cube.epx

Illustrations

Cas du test vl_edf_arlq4_3ib_cub8.epx :

Déformée et contrainte de Von Mises dans le modèle tous les 0.025 s (déplacement amplifié d'un facteur 2)



VL_EDF_DALLE_PONTI_DESM

Description

Title:

Reinforced concrete plate under blast loading

Problem:

A reinforced concrete circular plate (radius is 1m and thickness is 0.1m) is subjected to blast loading. A high and fast pressure is applied to the upper face (the peak is 0.32MPa). The symmetry of the problem enables us to model one quarter of the plate only. The maximum of displacement arises at $t=9.3\text{ms}$, and till the end time at $t=30\text{ms}$ one can observe oscillations and damping.

Mesh:

The model is 3D and uses 4320 CUB8 finite elements for concrete and 712 BR3D finite elements for metallic bars.

Materials:

For concrete, a nonlinear material DESM is used with the following elastic parameters: $\text{RO}=2300 \text{ kg/m}^3$, $E=3.1\text{E}10 \text{ Pa}$, $\nu=0.2$. For reinforcement, a nonlinear material VMIS ISOT is used with the following elastic parameter: $\text{RO}=7800 \text{ kg/m}^3$, $E=2.0777\text{E}11 \text{ Pa}$, $\nu=0.3$ and the initial yield stress is $450.0\text{E}6 \text{ Pa}$.

Boundary conditions:

One line of nodes is blocked along the z axis on the upper face and on the bottom face of the plate. Besides, axisymmetric constraint is imposed on two lateral faces by blocking degrees of freedom along x and y axis.

Initial conditions:

Nil.

Loading:

The blast loading consists in applying a pressure on the upper face of the plate.

Calculation:

The calculation is performed up to 30 ms.

Results:

Displacement, velocity and acceleration of the center of the plate on the bottom face are recorded as non-regression results at an intermediate time and final time.

Référence:

Comportement au souffle des structures en béton armé. Analyse expérimentale et modélisation, C. Pontirolli, thèse de doctorat, 1995.

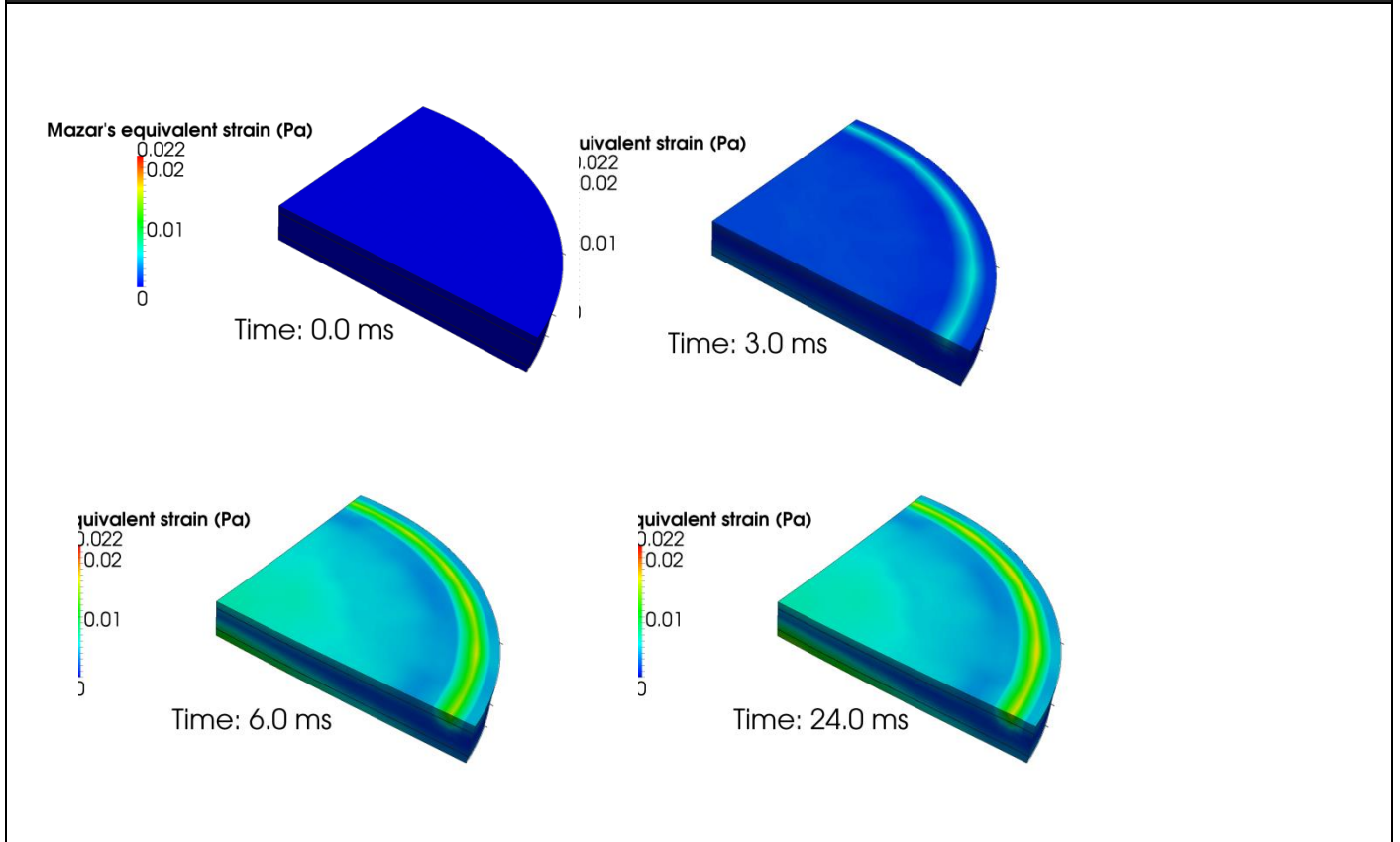
Exécution

Modifications du(des) jeu(x) de données : compte tenu d'un écart lié à de petites vitesses (10^{-1} m/s) et de la légère dispersion observée sur quelques calculs, la tolérance portant sur la deuxième valeur de validation (intermediate time) a été légèrement augmentée, passant de 0.1 à 0.2.

Qualifications :

OK : 4/4

Illustrations



VL_EDF_ELDI_FESHELL

Description

Titre :

Impact d'une barre sur une plaque encastrée sur son pourtour

Problème :

On considère l'impact normal d'un projectile modélisé avec les éléments discrets (ELDI) sur une plaque carrée modélisée avec les éléments finis coque (Q4GS). Le projectile a la forme d'une barre rectiligne uniforme de longueur 0,5 m et de section carrée de 0,25 m de côté. La cible est constituée d'une plaque métallique carrée de 1 m de côté et d'une épaisseur de 0,2 m.

Maillage :

La plaque est maillée en coques avec 1600 éléments quadrangulaires. Le projectile est discrétisé avec 2856 éléments discrets.

Matériaux :

Le comportement des matériaux est élastique linéaire. On utilise le matériau BL3S pour le projectile et le matériau LINE pour la cible.

Conditions aux limites :

La plaque est encastrée sur son pourtour. Le contact est modélisé via la méthode PINBALL.

Conditions initiales :

Jeu initial : 0,03 m.

Vitesse initiale du projectile : -20 m/s.

Résultats :

On compare les déplacements des différents points avec des valeurs d'un autre calcul EUROPLEXUS où le projectile est modélisé avec les éléments finis volumiques CUB8 et le traitement du contact par la méthode GLIS.

Références :

Modélisation du contact via la méthode PINBALL entre éléments discrets et éléments finis, J. Rousseau, 4e rapport d'avancement de thèse, 01/12/2008.

Exécution

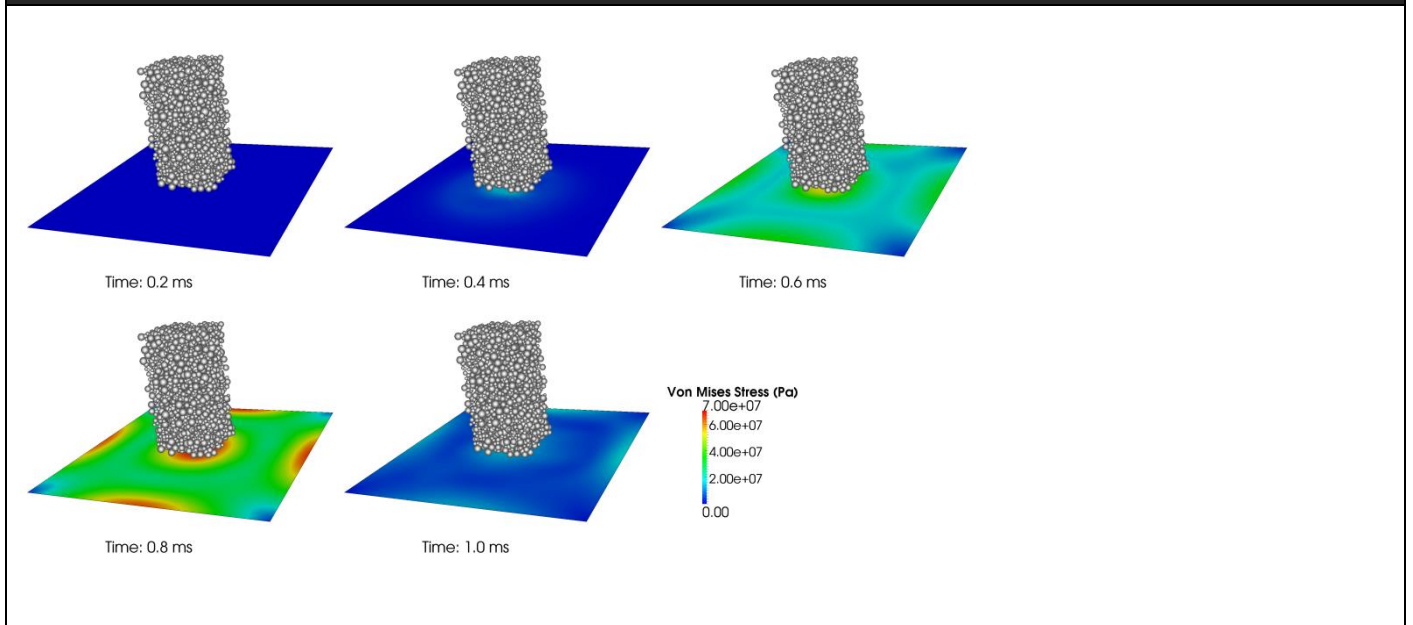
Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK :6/6

Les valeurs de qualifications portent sur le déplacement selon X de trois nœuds référencés AC, AHC et BC, à deux instants de simulation différents, avec des tolérances respectives différentes.

Illustrations



VL_EDF_ELDI_GRAN_SANDPILE_MPI_MED

Description

Titre :

Formation d'un tas de billes.

Problème :

Ce cas test vise à vérifier le comportement collectif des éléments discrets disponibles dans EUROPLEXUS (EPX) lorsque seules les interactions par contact sont considérées. La solution de référence se trouve dans un article [1].

Description du test :

L'essai est effectué dans une boîte rectangulaire avec une plaque au milieu dont la partie centrale est immobile et les extrémités sont amovibles. La profondeur de la boîte fait $4d$ où d est le diamètre d'une particule qui vaut 2 mm. La première étape consiste à remplir le haut de la boîte avec 2000 particules. Une fois le tassement terminé, on retire les deux côtés amovibles de la plaque, ce qui provoque un mouvement des particules vers la partie inférieure de la boîte. Les particules qui restent sur la plaque fixe forment un tas. On peut ensuite mesurer l'angle de repos de ce tas.

Maillage :

Les parois de la boîte sont modélisées par la méthode des éléments finis de type CUBE tandis que les particules sont modélisées par la méthode des éléments discrets de type ELDI.

Matériaux :

Les matériaux sont modélisés par les mots clés suivants : LINE et BL3S (BETON).

Les liens entre les particules sont de type CONT.

Les interactions entre une particule et la paroi sont de type LINK COUP GLIS.

Conditions aux limites :

Le mots clés « LINK BLOQ » est utilisé pour modéliser les conditions aux limites.

Chargement :

Le mot clé « GRAVITE » est utilisé.

Conditions initiales :

Le mot clé « INTE VITE » est utilisé.

Calcul :

Le calcul est mené jusqu'à 5 s avec un pas de temps de 10 microsecondes. On fait disparaître les plaques amovibles à $t = 1$ s en utilisant l'option « FANTOME ».

Résultats :

En exploitant les positions finales des billes, on mesure l'angle de dépôt et le compare avec la valeur expérimentale, soit 34 degrés.

Référence :

[1] ZHOU, Y., XU, B., YU, A. & ZULLI, P., 2002. An experimental and numerical study of the angle of repose of coarse spheres. Powder technology, 125 (45-54).

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque : Ce test est uniquement exécuté en séquentiel.

Qualifications : **OK :1/1**

VL_EDF_GLRC_BEAM

Description

Title:

Alternate quasi-static bending of a RC beam (large cycle) with GLRC DAMA model.

References:

This test case is completely described in the technical report [1], where EUROPLEXUS solution is compared with a solution by Code_Aster with multi-fiber beam model and the La Borderie non-linear damage model for concrete and elastic-plastic material for steel.

Object:

This quasi-static test case validates the GLRC DAMA model in cyclic bending. The tested structure is a 5m long RC beam with a symmetric steel reinforcement composed of two 32mm diameter bars for the upper and the lower layers.

Mesh:

The mesh is composed of 3-node T3GS thick shell elements (MED format).

Material:

GLRC DAMA (elasto-plasticity and damage) material law is used.

Boundary conditions:

The beam is simply supported at its two ends.

Loading:

The beam is subjected to an alternate displacement imposed at the beam center.

Two calculations have been performed:

- ➔ "vl_edf_glrc_beam_small_cycle.epx" using small cycle: A small cycle is considered involving a concrete cracking only while steel reinforcement remains elastic.
- ➔ "vl_edf_glrc_beam_large_cycle.epx" using large cycle: A large cycle is considered involving both concrete cracking and plastic deformations of steel reinforcement bars.

The load is imposed slowly and a mass proportional damping is used in order to reproduce a quasi-static bending condition.

Calculation:

The "vl_edf_glrc_beam_small_cycle.epx" calculation is performed up to 1.0 s.

The "vl_edf_glrc_beam_large_cycle.epx" calculation is performed up to 2.0 s.

Results – Qualifications:

The XX components of stress and curvature at beam center are compared with reference values (non-regression).

Reference:

[1] P. Koechlin, S. Moulin, Modèle de comportement global des plaques en béton armé sous chargement dynamique de flexion. Loi GLRC améliorée. Modélisation de la fissuration par endommagement, HT-62/02/021/A, 2002.

Exécution

Modifications du(des) jeu(x) de données :

Aucune.

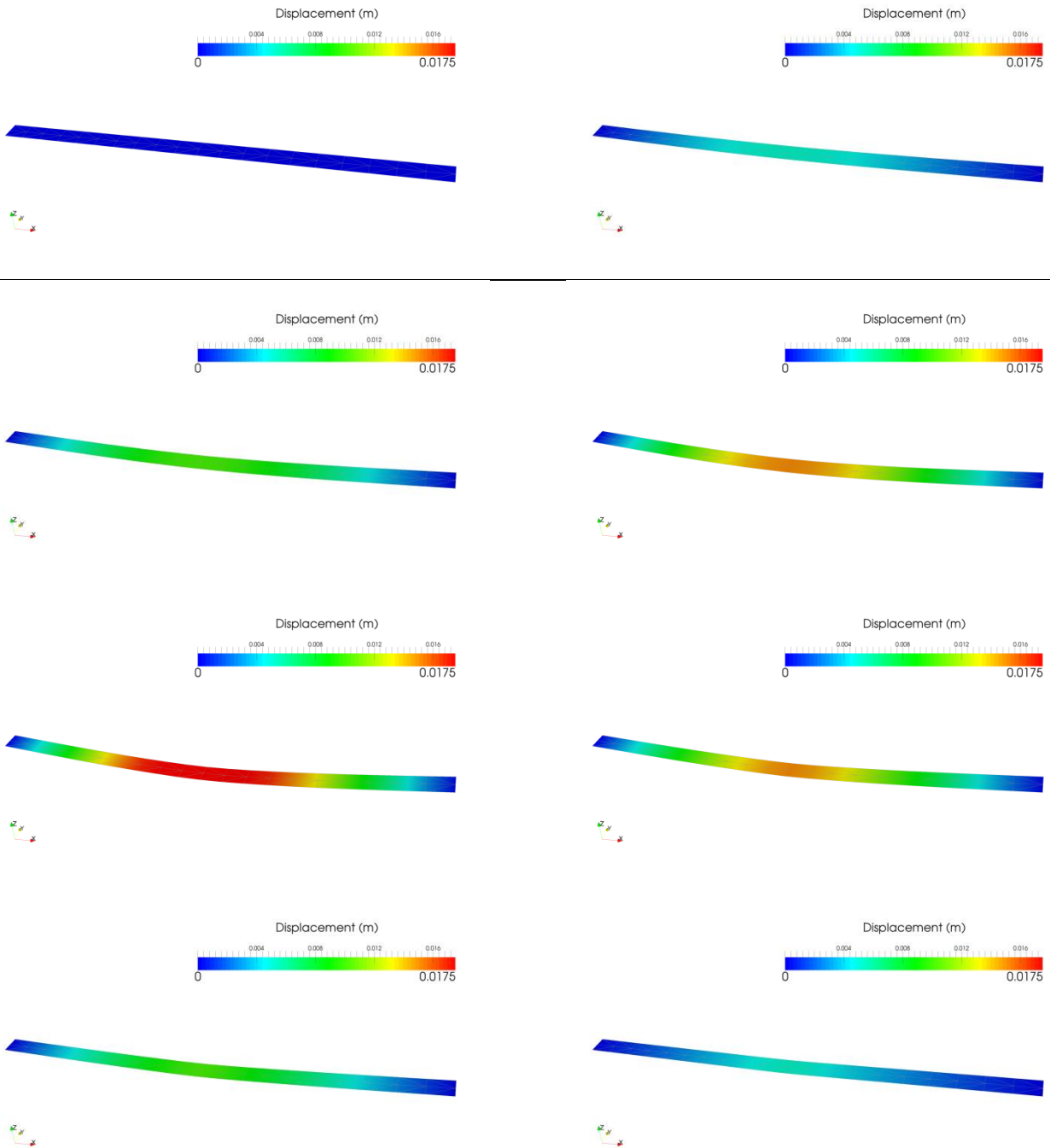
Qualifications :

OK : 2/2 pour vl_edf_glrc_beam_small_cycle.epx

OK : 2/2 pour vl_edf_glrc_beam_large_cycle.epx

Illustrations

Cas : vl_edf_glrc_beam_large_cycle.epx. Déplacement dans la poutre toutes les 0.1s (déformée amplifiée d'un facteur 10)



VL_EDF_IMPACT_2BARS

Description*Title:*

Axial impact of two identical bars

Problem:

Two identical bars (geometry and material) of square cross section 2x2 cm² and 1 m length move one to another with the same initial velocity $V_0=1$ m/s then collide. After a phase of contact, considered as elastic and whose duration depends on the length of the two bars and the speed of sound, the bars separate and move away in opposite directions with the velocity V_0 .

Mesh:

The model is 3D and uses 100 CUB8 finite elements (50 for each bar).

Materials:

For both bars, a linear elastic material is used with the following parameters: $E=2.E11$ Pa, $\nu=0$ (no Poisson effect for elastic wave propagation in the bars), $\rho=7800$ kg/m³.

Boundary Conditions:

Initially the bars are free of constraints. Contact is modeled by using GLIS directive.

Initial Conditions:

Initial gap: 1E-4 m.

Initial velocity: -1 m/s for the bar 1 and 1 m/s for the bar 2.

Loading:

Nil.

Calculation:

The calculation is performed up to 0.6 ms.

Results:

Displacements are compared with an analytical solution at an intermediate time and final time.

Reference:

Vibrations des structures. Ecole d'été d'analyse numérique, R.J.Gibert, CEA-EDF-INRIA, EYROLLES, 1988.

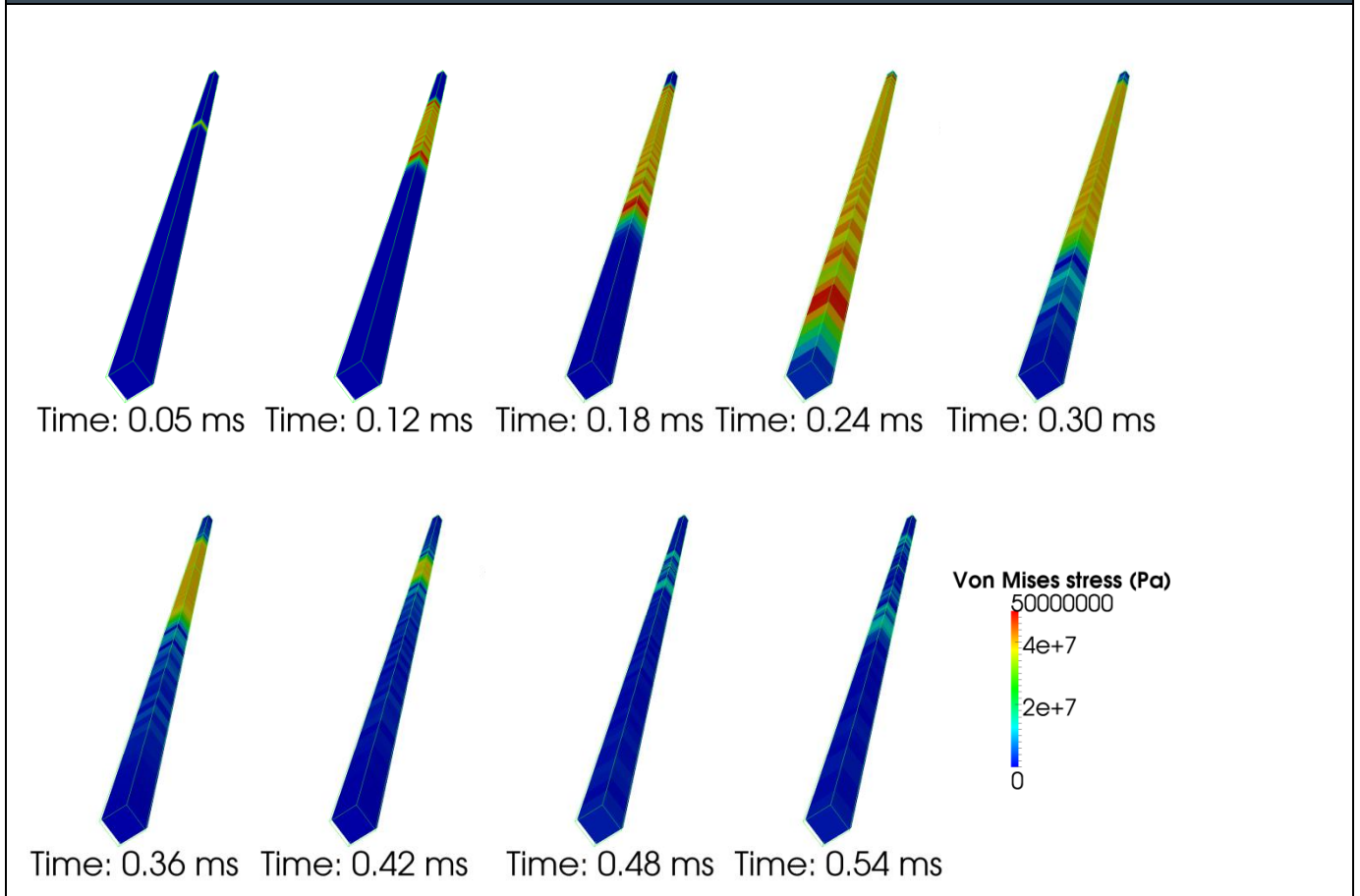
Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 4/4

Illustrations



VL_EDF_JOINT_GLUING_MPI

Description*Title:*

Gluing parts of an aluminum alloy assembly

Problem:

Two U-shaped aluminum alloys profiles, assembled together with 6 riveted joints, were tested under a traction loading conditions by the Norwegian University of Science and Technology [1]. This test is simulated with Europlexus using the GLUE method to model the global effect of the joints.

Test description:

Two U profiles (one horizontal and one vertical) made of 6063 T4 aluminum alloy are assembled with 6 rivets. The thickness is equal to 2 mm. Diameter of the rivets is equal to 10 mm. The assembly is tested on a traction test facility. A vertical displacement at the rate of 2.5 mm/s is applied at the top surface of the vertical U profile. The horizontal U profile is 40 mm long, clamped at its two extremities.

Mesh:

The Two U profiles are discretized with ``Q4GS`` (quadrangle) and ``T3GS`` (triangle) shell elements. The average size of the mesh is 5 mm.

Three loading areas are defined in the mesh:

- an area on the horizontal profile to be clamped,
- an area on the vertical profile to apply the displacement loading,
- an area in both profiles corresponding to the 6 rivets.

Materials:

The 6063 aluminum alloy has an anisotropic behavior. To model this particular behavior, we use two isotropic laws ``VM23``. Thus, according to the loading direction of the profiles, we consider that:

- The first one is equivalent to the transverse law for 6063 aa applied to the vertical profile,
- the second one is equivalent to the normal law for 6063 aa applied to the horizontal profile.

The rupture strain is equal to 20% for both laws.

Boundary and initial Conditions:

Three loading areas are defined in the model:

- the first area, on the horizontal profile, is clamped with the ``LINK BLOQ`` directive;
- the second area, situated on the vertical profile, a constant traction velocity of 10 m/s is applied with the ``LINK DEPL`` directive. This velocity is higher than the 2.5 mm/s velocity of the test in order to speed up the calculation;
- on the third area, both profiles are glued together to model the 6 rivets link using the ``LINK GLUE`` method

Calculation:

The calculation is performed up to 2 ms.

Results:

Comparison between experimental and numerical forces (resultant for all rivets) is made. It can be noted that numerical results are globally in a good agreement with experimental ones.

Reference :

[1] N.-H. Hoang, A.-G. Hanssen, M. Langseth, and R. Porcaro. Structural behaviour of aluminum self-piercing riveted joints: An experimental and numerical investigation. International Journal of Solids and Structures, 49(23): 3211-3223, 2012. URL: <http://www.sciencedirect.com/science/article/pii/S0020768312002673>.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque : Ce calcul est exécuté uniquement en MPI.

Qualifications :

OK : 1/1

VL_EDF_MEPPENII-12_GLRC

Description*Title:*

MEPPEN II-2 impact test studied with GLRC DAMA material

Problem:

The MEPPEN tests are viewed as real scale impact tests because the RC slabs are representative of existing NPP structures and the missile can be assimilated to a small aircraft. The steel tube-like projectile has a length of 6m and a diameter of 60cm. Its mass is around 1000kg and the impact velocity is 241.5 m/s.

The target is a RC slab of total dimensions 6.5mx6mx0.7m with

- rear face reinforcement ratio of 40.1 cm²/m²
- front face reinforcement ratio of 20.4 cm²/m²
- shear reinforcement ratio of 52.3 cm²/m².

Mesh:

The finite element mesh is composed of quadrangular shells (Q4GS)

Materials:

Global non-linear GLRC DAMA material law is used.

Boundary and initial Conditions:

The slab is simply supported by a rigid line-type supports restraining out-of-plane displacements of the slab. Because of the symmetry, only a quarter of the slab is modelled and symmetry conditions are prescribed on two symmetry planes.

Loading:

An impact force is uniformly applied to a disc of 0.4m radius at the center of the slab. The Riera's method is used to define the impact force evolution trend.

Damping:

No structural damping is added.

Calculation:

The calculation is performed up to 100 ms.

Results:

At the time $t=1.15E-2$ s, a resulting shear force is compared with the applied force divided by the factor $2*PI*R*EP$.

At the final time $t=2.E-2$ s, the displacement of point W9 in the impact direction is compared with experimental value.

Reference:

This validation test is fully described in the EDF R&D/AMA report HT62-2010-02533-EN entitled "EUROPLEXUS V&V file for APC studies".

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 2/2

VL_EDF_MEPPENII-2_GLRC

Description*Title:*

MEPPEN II-2 impact test studied with GLRC DAMA material

Problem:

The MEPPEN tests are viewed as real scale impact tests because the RC slabs are representative of existing NPP structures and the missile can be assimilated to a small aircraft. The steel tube-like projectile has a length of 6m and a diameter of 60cm. Its mass is around 1000kg and the impact velocity is 172.2 m/s.

The target is a RC slab of total dimensions 6.5mx6mx0.7m with

- rear face reinforcement ratio of 53.6 cm²/m²
- front face reinforcement ratio of 27.3 cm²/m²
- shear reinforcement ratio of 24.6 cm²/m².

Mesh:

The finite element mesh is composed of quadrangular shells (Q4GS)

Materials:

Global non-linear GLRC DAMA material law is used.

Boundary and initial Conditions:

The slab is simply supported by a rigid line-type supports restraining out-of-plane displacements of the slab. Because of the symmetry, only a quarter of the slab is modelled and symmetry conditions are prescribed on two symmetry planes.

Loading:

An impact force is uniformly applied to a disc of 0.4m radius at the center of the slab. The Riera's method is used to define the impact force evolution trend.

Damping:

No structural damping is added.

Calculation:

The calculation is performed up to 100 ms.

Results:

At the time $t=1.75E-2$ s, a resulting shear force is compared with the applied force divided by the factor $2*PI*R*EP$.

At the time $t=1.45E-2$ s, the displacement of point W5 in the impact direction is compared with experimental value.

Reference:

This validation test is fully described in the EDF R&D/AMA report HT62-2010-02533-EN entitled "EUROPLEXUS V&V file for APC studies".

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 2/2

VL_EDF_PERFO_ALUM_PLATE

Description*Title:*

Ballistic impact test for 2024-T3/T351 aluminum thin plate

Problem:

This ballistic test was performed with a pneumatic gun projecting a 12.7 mm diameter chrome 52100 alloy steel sphere onto a 2024-T3/T351 aluminum target plate of 1.5875 mm thickness, cut to 304.8 by 304.8 mm size and attached to the 25.4 mm wide support frame, thus leaving a 254 by 254 mm free target area. For this test, the ballistic limit is found about 122 m/s [1]. For the impact velocity of 200 m/s, the residual velocity of the sphere is 160 m/s.

Mesh:

The plate is discretized with T3GS plate elements. Fully fixed boundary conditions are imposed on the outer boundary of the plate. A fully 3D meshed sphere (TETR) with the projectile-to-plate contact modeled through the GLIS directive is used.

Materials:

To describe the response of 2024-T3/T351 aluminum plate, a Johnson-Cook material model (VMJC) of EUROPLEXUS is employed with the LLNL-1 set of parameters suggested in [2]. The element thickness variation is activated through the LMST option [3].

Plate elements erosion driven by Johnson-Cook failure criterion is activated.

The LINE elastic material is used for the steel sphere.

Contact between the sphere and the plate is detected by LINK COUP GLIS directive with the master surface defined by the sphere elements. Contact constraints are solved by the Lagrange multipliers method. Friction coefficient $m=0.3$ is used.

Boundary and initial Conditions:

The plate boundary is fixed.

An initial vertical velocity of 200 m/s is applied to the sphere.

Calculation:

The calculation is performed up to 0.2 ms.

Results:

One checks the residual velocity of the sphere after perforation.

References:

[1] Kelly, S. and Johnson, G., Statistical testing of aircraft materials for transport airplane rotor burst fragment shielding, Federal Aviation Administration, FAA Rep. No. DOT/FAA/AR-06/9, 2006.

[2] M. Buyuk, S. Kan and M.J. Loikkanen, Explicit Finite-Element Analysis of 2024-T3/T351 Aluminum Material under Impact Loading for Airplane Engine Containment and Fragment Shielding Journal of Aerospace Engineering, Vol. 22, Nu. 3, 287-295, 2009.

[3] F. Casadei, V. Aune G. Valsamos and M. Larcher, Accounting for large membrane strains in Q4GS and T3GS elements in EUROPLEXUS, Technical Report EUR 27836 EN, 2016.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 1/1

VL_EDF_SHELL_BEAM_PINB_CONT

Description

Title:

Impact of a rigid shell-type missile on a steel reinforcement grid.

Problem:

This test presents a simulation with EUROPLEXUS of a verification case where a rigid aluminum missile is launched against a steel grid representing a reinforcement grid of a reinforced concrete slab. This case permits to verify the right behavior of the pinball method to treat the contact between shell and beam elements, which cannot be represented with GLIS method since it is not possible to define an outer normal on a beam element.

Moreover, this case compares the results obtained with the Lagrange multipliers method and the penalty method to estimate the contact force and to verify the correct behavior of the model when erosion is activated. For the final simulation of an aircraft crash on a reinforced concrete structure with EUROPLEXUS. The code must be able to correctly simulate the contacts among different structural components during the impact, in particular shell/beam contact.

Case description

In this simulation, a cylindrical thin-wall missile with spherical cap nose is launched against a grid composed of two series of beams interconnected at their cross points. This case focuses on the detection of the contact between a thin shell and a web of beams and aims at checking the correct behavior of the grid during the impact. To facilitate the verification, the missile is considered rigid.

Mesh:

The grid is discretized with 50 mm length POUT elements and the missile is meshed with 73 mm edge size Q4GS elements. Steel reinforcement beams are interconnected at crossing points by using common nodes.

Materials:

The missile is considered as rigid with the AA-2024/T3 characteristics.
For the grid, an isotropic Von Mises material law (VMIS ISOT) is used.
The contact method 'LINK COUP PINB' is used.

Boundary and initial Conditions:

The missile is launched with a 70 m/s initial velocity. Both ends of all reinforcement beams are blocked (LINK BLOQ).

Calculation:

The calculation is performed up to 20 ms.

Results:

In the calculation, pinballs located in elements are used and defined with DMIN 0.009. It implies that the diameter of the higher hierarchical pinball is bigger than 0.009.

One verifies the contact detection between a thin shell and a web of beams.

This case can be run in serial and parallel but it is mandatory to use LINK COUP option.

References:

None

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 1/1

VL_EDF_TI4AL6V BALLISTIC TEST MPI

Description

Title:

Ballistic impact test on Ti-4Al-6V thin plate

Problem:

The Titanium alloy Ti-4Al-6V is used in aircraft engine pylon. A validation test for this material is run here with EUROPLEXUS by studying perforation of a thin Ti-4Al-6V plate by a spherical impactor. Numerical results are compared with those of ballistic impact tests conducted by the Ballistic Research Laboratories [1].

Test description

High velocity impact tests involve a 3.16 mm diameter steel sphere perforating a thin 1.32 mm Ti-4Al-6V plate for a variety of initial velocities from 500 m/s to 1500 m/s. Experimental results are given in [1], p. 156.

Mesh:

The Ti-4Al-6V plate is discretized with Q4GS (quadrangle) shell elements. Three meshes for the plate are built. One with a size of 0.4 mm, one with the size of 0.2 mm and the thinner with a size of 0.1 mm. The spherical impactor is meshed with tetrahedral elements TETR.

Materials:

The steel behavior of the spherical impactor and Ti-4Al-6V plate is modeled with a Johnson-Cook material law (VMJC). Plate elements erosion driven by a Johnson-Cook failure criterion is activated in all presented calculations.

Boundary and initial Conditions:

Initial velocities, from 500 m/s to 1500 m/s, are applied to the impactor with the INIT VITE directive. The contact between the plate and the impactor is considered with the LINK GLIS master/slave surface method. Because of the high velocity of the impactor (> 500 m/s), the influence of the friction is negligible.

Calculation:

The calculation is performed up to 0.01 ms.

Results:

The residual velocity after perforation is tested and compared to the experimental value.

Reference:

[1] Zvi Rosenberg and Erez Dekel. Terminal Ballistics, Second Edition. Springer, 2012. ISBN 978-981-10-0393-6. doi:10.1007/978-981-10-0395-0.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en MPI

Qualifications :

OK : 1/1

VL_EDF_VTT_MISSILE_IMPACT

Description*Title:*

Axial impact of an EN 1.4432 steel missile on a rigid steel slab

Problem:

This test simulates the calibration VTT test [1] where a thin-walled steel missile is launched on a rigid steel plate. The ability to simulate a thin-walled projectile crash with a coarse mesh and erosion activated is studied here.

Test description

This experiment is a preliminary test whose purpose is the characterization of the missile for the IRIS 2010 VTT test. During the experiment, the missile is launched against a rigid steel plate at a 102.21 m/s velocity.

Mesh:

The slab is discretized with CUBE elements whereas the missile is meshed with Q4GS shells. To reduce the complexity, the rear part is meshed with 25 mm thick shell elements with a density recalibrate to 18650 kg/m3. This part of the missile is well preserved during the impact, so this representation does not influence the result.

Materials:

Materials involved: VMJC, LINE

Boundary and initial Conditions:

The lower face of the steel slab is blocked (LINK BLOQ). Contact conditions (LINK GLIS) are prescribed between the slab and the missile. For the 5 mm mesh-size, self-contact is activated to simulate the wrinkling during impact. An initial normal to the slab velocity of 102.21 m/s is imposed to the missile.

Calculation:

The calculation is performed up to 20 ms.

Results:

The experimental and simulation results are compared as to the final length of the projectile.

Reference:

[1] IRIS_2010, Improving Robustness Assessment Methodologies for Structures Impacted by Missiles, NEA/CSNI/R(2011)8, Final Report, 2012.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 1/1

VL_EDF_1D3D_SIMPSON

Description*Title:*

Simpson's water-hammer experiments - monophasic case

Problem:

The Simpson experiments are considered to assess the ability of EUROPLEXUS to model a water hammer triggered by the rapid closure of a valve in a steady-state flow. In addition, after wave reflection, cavitation can occur at the valve and vapor can appear. These tests validate the correct modeling of initial hydrostatic pressure gradients, gravity effects, wave propagation and reflection at a pressurized tank boundary, modeling of valve closure, steam bubble generation and collapse. By varying the initial flow velocity, cavitation level and the strength of the generated water- and condensation-induced water-hammer events are strongly modified.

Geometry:

The experimental setup consists of a water circuit including two tanks, an upstream and a downstream tank, at different levels. The pressure difference in the two tanks produces a water flow opposite to the sense of gravity in a 36-m long straight pipe (inner diameter of 19 mm). A steady flow is initially produced as water is allowed to circulate back to the upstream tank through a return line in a closed circuit.

The initial temperature in the piping is ambient and measured to be 23.85 °C. The pressure of the up-stream tank depends on the case in study and varies between 3.1 and 3.5 bar. The flow velocity is thus adjusted for the different cases to cover a range of 0.2 - 1.1 m/s. At a given instant, a ball valve located at a 36 m distance from the upstream tank is actuated, resulting in a quasi-instantaneous closure of the piping at this position. This triggers a compression wave (P+) from the valve towards the reservoir, in the opposite direction of the steady flow. The underlying physics and resulting effects are shown to depend essentially on the initial flow velocity.

Mesh:

The pipe is modeled by 1D elements TYVF from $x = 0$ m to $x = 15$ m, then by 3D elements CUVF for the fluid coupled with 2D elements Q4GS for the pipe from $x = 15$ m to $x = 21$ m and finally by 1D elements TYVF from $x = 21$ m to $x = 36$ m. The links between the 1D meshes and the 3D mesh are performed using CL3D for 3D faces connected with 1D elements and TUYM for the 1D-3D junctions. Note that the influence of the pipe elasticity on the effective speed of pressure waves (reduced speed of sound, sometimes called Allievi effect) is not taken into account here. It is a pure Eulerian mesh where the motion of nodes and the deformation of elements are not possible. Elastic stresses and deformations of beams and quads are computed but not applied to the mesh.

Materials:

The material used to model water is the FLUID material with a density $\rho_0 = 997.407$ kg/m³ and a speed of sound $c_0 = 1280$ m/s. The value used for the speed of sound matches the experimental data and takes into account the fluid compressibility and the pipe elasticity. In addition, the minimum pressure in the fluid is set to the vapor pressure of water at 23.85 °C i.e. $p_{\min} = 0.03$ bar. The case considered here is a monophasic case with $p_0 = 3.469$ bar and $u_0 = 0.239$ m/s. The tube material is standard aluminum with a density of $\rho = 7850$ kg/m³, a Young modulus of $E = 75$ GPa and a Poisson coefficient of $\nu = 0.3$. The tube thickness is 1.6 mm.

Boundary Conditions:

The valve closure is assumed to be instantaneous. The pressure vessel is modeled using a CAVF element.

Results:

The computation is performed with the Courant number $C = 0.8$. The numerical results obtained with the 1D/3D approach are compared to the experimental data. Good agreement is obtained for the pressure history at different locations along the pipe.

References:

- A.R. Simpson, Large Water Hammer Pressures due to Column Separation in Sloping Pipes (Transient Cavitation), Ph.D. thesis, The University of Michigan, 1986

- A.R. Simpson, E.B. Wylie, Large water-hammer pressures for column separation in pipelines, ASCE J. Hydraul. Eng., 117 (10) (1991), pp. 1310-1316
- F. Daude, A.S. Tijsseling, P. Galon, Numerical investigations of water-hammer with column-separation induced by vaporous cavitation using a one-dimensional Finite-Volume approach, J. Fluids Struct., 83 (2018), pp. 91-118
- F. Daude, T. Douillet-Grellier, P.Galon, Europlexus verification and validation (V&V) report: thermal-hydraulic fluid transients, 3125-1721-2020-03020-FR Technical Report, EDF R&D Paris Saclay, 2020.

Exécution

Modifications du(des) jeu(x) de données : En 2023, correction du chargement lié à la gravité.

Qualifications :
OK : 30/30

VL_EDF_1D3D_SIMPSON_2

Description*Title:*

Simpson's water-hammer experiments with moderate cavitation

Problem:

The Simpson experiments are considered to assess the ability of EUROPLEXUS to model a water hammer triggered by the rapid closure of a valve in a steady-state flow. In addition, after wave reflection, cavitation can occur at the valve and vapor can appear. These tests validate the correct modeling of initial hydrostatic pressure gradients, gravity effects, wave propagation and reflection at a pressurized tank boundary, modeling of valve closure, steam bubble generation and collapse. By varying the initial flow velocity, cavitation level and the strength of the generated water- and condensation-induced water-hammer events are strongly modified.

Geometry:

The experimental setup consists of a water circuit including two tanks, an upstream and a downstream tank, at different levels. The pressure difference in the two tanks produces a water flow opposite to the sense of gravity in a 36-m long straight pipe (inner diameter of 19 mm). A steady flow is initially produced as water is allowed to circulate back to the upstream tank through a return line in a closed circuit.

The initial temperature in the piping is ambient and measured to be 23.85 °C. The pressure of the up-stream tank depends on the case in study and varies between 3.1 and 3.5 bar. The flow velocity is thus adjusted for the different cases to cover a range of 0.2 - 1.1 m/s. At a given instant, a ball valve located at a 36 m distance from the upstream tank is actuated, resulting in a quasi-instantaneous closure of the piping at this position. This triggers a compression wave (P+) from the valve towards the reservoir, in the opposite direction of the steady flow. The underlying physics and resulting effects are shown to depend essentially on the initial flow velocity.

Mesh:

The pipe is modeled by 1D elements TYVF from $x = 0$ m to $x = 15$ m, then by 3D elements CUVF for the fluid coupled with 2D elements Q4GS for the pipe from $x = 15$ m to $x = 21$ m and finally by 1D elements TYVF from $x = 21$ m to $x = 36$ m. The links between the 1D meshes and the 3D mesh are performed using CL3D for 3D faces connected with 1D elements and TUYM for the 1D-3D junctions. Note that the influence of the pipe elasticity on the effective speed of pressure waves (reduced speed of sound, sometimes called Allievi effect) is not taken into account here. It is a pure Eulerian mesh where the motion of nodes and the deformation of elements are not possible. Elastic stresses and deformations of beams and quads are computed but not applied to the mesh.

Materials:

The material used to model water is the FLUID material with a density $\rho_0 = 997.407$ kg/m³ and a speed of sound $c_0 = 1280$ m/s. The value used for the speed of sound matches the experimental data and takes into account the fluid compressibility and the pipe elasticity. In addition, the minimum pressure in the fluid is set to the vapor pressure of water at 23.85 °C i.e. $p_{\min} = 0.03$ bar. A case with moderate cavitation is considered here ($p_0 = 3.281$ bar and $u_0 = 0.401$ m/s). The tube material is standard aluminum with a density of $\rho = 7850$ kg/m³, a Young modulus of $E = 75$ GPa and a Poisson coefficient of $\nu = 0.3$. The tube thickness is 1.6 mm.

Boundary Conditions:

The valve closure is assumed to be instantaneous. The pressure vessel is modeled using a CAVF element.

Results:

The computation is performed with the Courant number $C = 0.8$. The numerical results obtained with the 1D/3D approach are compared to the experimental data. Good agreement is obtained for the pressure history at different locations along the pipe.

References:

- A.R. Simpson, Large Water Hammer Pressures due to Column Separation in Sloping Pipes (Transient Cavitation), Ph.D. thesis, The University of Michigan, 1986

- A.R. Simpson, E.B. Wylie, Large water-hammer pressures for column separation in pipelines, ASCE J. Hydraul. Eng., 117 (10) (1991), pp. 1310-1316
- F. Daude, A.S. Tijsseling, P. Galon, Numerical investigations of water-hammer with column-separation induced by vaporous cavitation using a one-dimensional Finite-Volume approach, J. Fluids Struct., 83 (2018), pp. 91-118
- F. Daude, T. Douillet-Grellier, P.Galon, Europlexus verification and validation (V&V) report : thermal-hydraulic fluid transients, 3125-1721-2020-03020-FR Technical Report, EDF R&D Paris Saclay, 2020.

Exécution

Modifications du(des) jeu(x) de données : En 2023, correction du chargement lié à la gravité.

Qualifications :
OK : 30/30

VL_EDF_HDR_MOD2_RAP0.99_MPI

Description*Title:*

HDR blowdown experiment V32

Problem:

A series of blowdown experiments have been performed under varying conditions in the early 1980's in Germany [1,2]. These experiments have been carried out in order to study the FSI effects during a sudden depressurization in a full-scale geometry by using realistic conditions. These tests aim at verifying the ability of the 1-D/3-D FSI coupling FV/FE approach to reproduce the mechanical response of the reactor pressure vessel internals and of the core barrel due to the rapid depressurization following a water line pipe break.

Geometry:

The test facility (see [1-3]) consists of a reactor pressure vessel with internals, several pipes connected to the vessel filled with water and a blowdown nozzle. The vessel internals include a flexible core barrel with a mass ring attached to its bottom. The mass ring represents in a simplified way the inertial effects of the internal structures which are removed in the present experiments. The size of the HDR reactor vessel, 11 m high and 3 m in diameter, is very similar to a typical pressurized water reactor (PWR). The core barrel is 7.6 m high with a diameter of 2.66 m and a thickness of 2.3 cm. The weight of the mass ring is 13 500 kg. The lower end of the core barrel is free and its upper end is rigidly clamped. Finally, the break nozzle is 1.37 m long with an inner diameter $d = 0.2$ m.

Mesh:

Some geometric simplifications have been done (see [3] for more details). The core barrel is modelled using 17200 4-node quadrilateral Reissner-Mindlin shell elements with 6 degrees of freedom per node and 20 Gauss integration points (4 in the plane, 5 through the thickness). The mass ring is represented by 14400 8-node brick solid elements with 3 degrees of freedom per node and 6 Gauss integration points. Concerning the fluid domain, the downcomer, the lower plenum, the core region and the part of the nozzle connected to the vessel are discretized using 731328 3-D cells. The other part of the break nozzle is discretized using 16 1-D cells and a 1-D/3-D fluid coupling is used at the interface between the 1-D and the 3-D regions of the fluid computational domain.

Materials:

A linear elastic constitutive law is used to represent the material behavior of the flexible core barrel and the mass ring with the following characteristics: a Young's modulus $E = 175 \times 10^3$ MPa, a Poisson's ratio $\nu = 0.295$ for both the core barrel and the mass ring and a density $\rho_s = 7790$ kg.m⁻³ for the core barrel and $\rho_s = 10612$ kg.m⁻³ for the mass ring. The density value of the mass ring is chosen to correspond to its weight. The fluid material EAU is used here. It treats liquid water and its vapor as a homogeneous mixture: the liquid and vapor phases are defined to be at the same pressure and temperature.

Boundary Conditions:

The upper end of the core barrel is rigidly clamped. In this experiment V32, the water at rest filling the test facility is at an initial pressure of $p = 110$ bar. The initial temperature in the break nozzle and in the downcomer is $T = 240^\circ\text{C}$. The initial temperature in the lower plenum is $T = 235^\circ\text{C}$. The initial core temperature varied axially from $T = 283^\circ\text{C}$ at the lower core barrel end to $T = 308^\circ\text{C}$ at the upper core. Initial subcooling between the downcomer and the nozzle area is quite large, i.e. $\Delta T = 78^\circ\text{C}$, in the experiment which increased the loads on the core barrel. The free end of the blowdown nozzle is closed by a membrane which is suddenly broken at the beginning of the experiments with a short break opening time (about 1-2 ms). In the present computation, a break opening time of 1 ms is assumed and the Moody's critical flow model [4] is used at the free end of the blowdown nozzle. As a consequence, a strong depressurization wave is generated at the break, then propagates through the blowdown nozzle and reaches the reactor pressure vessel. Afterwards, the pressure wave propagates inside the downcomer between the reactor pressure vessel and the core barrel. Then, starting from the lower end of the core barrel, the pressure wave travels upward inside the core barrel. Due to the high variation of the pressure inside the reactor pressure vessel induced by the propagation of the depressurization wave, strong mechanical loads are applied to the core barrel producing displacements and deformations of the structure.

Results:

The computation is performed with the Courant number $C = 0.8$. Second-order accuracy is used in the FV approximation. The numerical results obtained with the 1-D/3-D FSI numerical approach are compared with the experimental data for test V32 reported in [1,2]. A good agreement between the numerical results and the experimental data is observed: the depressurization phase is well retrieved in the computations as the pressure wave propagates through the downcomer as well as the recompression phase due to the core barrel motion. We can observe that the pressure waves travel in the downcomer with a velocity significantly smaller than the speed of sound because of the flexibility of the core barrel. The overall behavior of the core barrel motion is qualitatively well retrieved in the present computation. However, we can notice that the amplitude of the radial displacement is slightly underestimated in the numerical results. As the pressure loads received by the core barrel are well retrieved in the computation, this tends to show that this discrepancy is due to the structural modeling (boundary conditions, area where the fluid force is applied, mass ring and core barrel dimensions etc...).

References:

- [1] L. Wolf. Design report for the HDR-RPV-1 blowdown experiments V31.2, V32, V33 and V34 with specifications for the pretest computation. Technical Report HDR Safety Program, PDHR Report No. 3/243/81, KFZ, 1981.
- [2] L. Wolf. Experimental results of coupled fluid-structure interactions during blowdown of the HDR-vessel and comparisons with pre- and post-test predictions. Nuclear Engineering and Design, 70:269–308, 1982.
- [3] Europlexus verification et validation (V&V) report : thermal-hydraulic fluid transients, 3125-1721-2020-03020-FR Technical Report, EDF R&D Paris Saclay, 2020, F. Daude, T. Douillet-Grellier, P.Galon.
- [4] F. J. Moody. Maximum flow rate of a single component two-phase mixture. Journal of Heat Transfer, 87:134–141, 1965.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en MPI

Qualifications :

OK : 17/17

VL_EDF_CWHTF_1D_V-V_RTHER

Description*Title:*

CWHTF 290601 experiment: 1D Finite Volume Modelling (with FSI)

Problem:

The Cold Water-Hammer Test-Facility (CWHTF) was built in Forschungszentrum Rossendorf. The experiments aim to obtain fluid-structure interaction effects such as dynamic 3-D stresses in the walls during the water-hammer transient. High internal fluid pressure and significant pipe stress variations are measured. The test 290601 is here considered to assess the capability of the 1-D FSI FV/FE approach to predict both fluid pressure and structural stress waves in the same coupled computation.

Mesh:

The considered experimental test facility consists of a pressure vessel, a pipe line with horizontal and vertical straight sections, two 90-deg elbows with a radius of curvature of 306 mm and a fast acting valve in the horizontal section. The total length of the pipeline is $L = 3.3$ m; the inner pipe diameter is about $d = 207$ mm and the wall thickness $e = 6$ mm. The vertical pipe region is terminated by a lid flange with a weight of 19.25 kg which acts as a bouncing plate. The mesh is composed of 79 TYVF elements, 2 APPU elements, 1 PMAT element, 1 CLTU and 1 CAVF element.

Materials:

The stainless steel pipe of the CWHTF is supposed to have a linear elastic behavior. The pipe is filled with Water ('EAU' material).

Boundary Conditions:

The vessel at the inlet of the pipeline is modeled using a tank boundary condition. The free bouncing plate corresponds to a moving wall boundary condition associated to a weight of 19.25 kg.

Loading:

At the left side of the valve, the thermo-dynamical state of the sub-cooled water is given by $p = 1$ bar and $T = 21$ °C whereas the right side of the valve is filled with a saturated liquid water, of which initial state is given by $p = 29$ mbar and $T = 21$ °C. The top section of the vertical pipe is filled with steam at $p = 29$ mbar.

Results:

The evolution during 0.6 s of the coupled pipe-fluid system is studied. Good agreement is observed between the numerical results and the experimental data.

References:

- Fluid-Structure Interaction Investigations for Pipelines. Technical Report FZR-393, Forschungszentrum Rossendorf, 2003, E. Altstadt, H. Carl, and R. Weiss.
- Europlexus verification and validation (V&V) report : thermal-hydraulic fluid transients, 3125-1721-2020-03020-FR Technical Report, EDF R&D Paris Saclay, 2020, F. Daude, T. Douillet-Grellier, P.Galon.
- Eléments de validation du code Europlexus pour la simulation des phénomènes transitoires rapides dans les tuyauteries, 6125-1721-2021-01699-EN Technical Report, EDF R&D Paris Saclay, 2021, F. Daude.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 12/12

VL_EDF_ESSAI44_AQUITAINEII_1D

Description*Title:*

Aquitaine pipe whipping experiment (Aquitaine II experiment: test 44)

Problem:

This experiment is characterized by the presence of an elbow impacting a stiff structure. The tests were performed in CEA at the AQUITAINE facility [2] which has been designed to investigate the mechanical consequences of a breach opening on a primary pipe segment.

These tests aim at verifying the ability of the 1-D FSI coupling FV/FE approach to reproduce the pipe motion and the impact force during such a severe transient. During the experiments, pipe breaks are initiated with an explosive cord leading to a breach opening time less than one millisecond. Due to the difference between the internal pressure and the atmospheric conditions, the fluid is discharged out of the pipe. The induced blowdown thrust force acts on the pipe which causes the dynamic pipe motion. Then, during the pipe movement, the elbow impacts on the rigid obstacle leading to the local plastic crush of the pipe cross-section.

Geometry:

The facility consists of a vessel (which has a capacity of 0.25 m³), a 0.43-m-long horizontal pipe, a 90-deg elbow of 0.135 m radius of curvature and a 0.2-m-long vertical pipe (see [2,1] for more details). The pipes (with an inner diameter of $d = 88.9$ mm and a wall thickness of $\delta = 7.62$ mm) are made of 316L stainless steel similar to A106 grade B. The initial gap between the pipe and the target is $h = 186$ mm.

Mesh:

3 cells are used for the fluid domain corresponding to the flanged part of the horizontal pipe, 14 FSI pipe elements are used for the horizontal part of the flexible pipe, 7 FSI elements for the elbow and 7 FSI elements for the vertical part of the pipe.

Materials:

As in [3,9], a bi-linear constitutive law is used and is based on the following mechanical characteristics: Young's modulus $E = 180 \times 10^3$ MPa, Poisson's ratio $\nu = 0.33$, density $\rho_0 = 7800$ kg/m³, yield stress $\sigma_Y = 155$ MPa, yield strain $\epsilon_Y = 0.086$ %, ultimate stress $\sigma_r = 475$ MPa and deformation at rupture $\epsilon_r = 25$ %.

Boundary Conditions – Loading:

The free end of the vertical pipe is initially closed by a rigid membrane separating the internal volume of the pipe from the external atmosphere. The pipe is rigidly connected to the vessel by means of a flange. The tank and the pipe are filled with hot liquid water at rest, at pressure $p = 166$ bar and at temperature $T = 326^\circ\text{C}$. In the present computation, an instantaneous break of the membrane is assumed and the Moody's critical flow model [10] is used at the free end of the vertical pipe. The vessel at the inlet of the pipe is modeled using a tank boundary condition.

Results:

The computation is performed with the Courant number $C = 0.8$. The numerical results obtained with the 1-D FSI numerical approach are first compared with the experimental data for test 44. The pipe motion is well reproduced in the simulation.

In contrast, the impact load is strongly overestimated. This is due to the reduced kinematics of the beam element, i.e. the variation of the cross-section is not considered. This was also observed in the previous study [11,3]. In order to circumvent this problem, 3-D elbow modeling could be applied as in [11,3] where the impact zone is modeled using shell elements that enable to naturally take into account the local crushing of the pipe.

References:

[1] Europlexus verification et validation (V&V) report : thermal-hydraulic fluid transients, 3125-1721-2020-03020-FR Technical Report, EDF R&D Paris Saclay, 2020, F. Daude, T. Douillet-Grellier, P.Galon.

[2] J. L. Garcia, P. Chouard, and E. Sermet. Experimental studies of pipe impact on rigid restraints and concrete slabs. Nuclear Engineering and Design, 77:357–368, 1984.

- [3] S. Potapov and P. Galon. Modelling of Aquitaine II pipe whipping test with the plexus fast dynamics code. Nuclear Engineering and Design, 235(17-19):2045–2054, 2005.
- [4] P. Caumette, J.L. Garcia, Experimental studies of PWR primary piping under LOCA conditions. Nuclear Engineering and Design 61 (1980) 197-208.
- [5] J.L. Garcia, P. Caumette, J.L. Huet, A. Martin, Study of pipe whip and impact. SMIRT 1981
- [6] A. Martin, E. Sermet, J.L. Garcia, AQUITAINE II complémentaire : Rapport final d'essais. Note technique DRE/STRE/LMA 84/362
- [7] J.L. Garcia, "AQUITAINE II - Interprétation" : analyses d'essais avec chocs sur structures rigides. Note technique DRE/STRE/LMA 84/373
- [8] A. Ricard, A. Millard, Fouettement des tuyauteries par le programme TEDEL. Rapport DDMT/SMTS/BAMS/81-52
- [9] P. Galon, M. Lepareux, Comportement dynamique des tuyauteries. Analyse d'un essai de fouettement au moyen de PLEXUS. Rapport DDMT/SEMT/DYN/RT/99-028, 1999.
- [10] F. J. Moody. Maximum flow rate of a single component two-phase mixture. Journal of Heat Transfer, 87:134–141, 1965.
- [11] P. Galon, M. Lepareux, Couplage 1D/3D: analyse d'un essai de fouettement au moyen de EUROPLEXUS. Rapport DRN/DMT/SEMT/DYN/ RT/00.015/A

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 35/35

VL_EDF_ESSAI45_AQUITAINEII_1D

Description*Title:*

Aquitaine pipe whipping experiment (Aquitaine II experiment: test 45)

Problem:

This experiment is characterized by the presence of an elbow impacting a stiff structure. The tests were performed in CEA at the AQUITAINE facility [2] which has been designed to investigate the mechanical consequences of a breach opening on a primary pipe segment.

These tests aim at verifying the ability of the 1-D FSI coupling FV/FE approach to reproduce the pipe motion and the impact force during such a severe transient. During the experiments, pipe breaks are initiated with an explosive cord leading to a breach opening time less than one millisecond. Due to the difference between the internal pressure and the atmospheric conditions, the fluid is discharged out of the pipe. The induced blowdown thrust force acts on the pipe which causes the dynamic pipe motion. Then, during the pipe movement, the elbow impacts on the rigid obstacle leading to the local plastic crush of the pipe cross-section.

Geometry:

The facility consists of a vessel (which has a capacity of 0.25 m³), a 0.43-m-long horizontal pipe, a 90-deg elbow of 0.135 m radius of curvature and a 0.2-m-long vertical pipe (see [2,1] for more details). The pipes (with an inner diameter of $d = 88.9$ mm and a wall thickness of $\delta = 7.62$ mm) are made of 316L stainless steel similar to A106 grade B. The initial gap between the pipe and the target is $h = 267$ mm.

Mesh:

3 cells are used for the fluid domain corresponding to the flanged part of the horizontal pipe, 14 FSI pipe elements are used for the horizontal part of the flexible pipe, 7 FSI elements for the elbow and 7 FSI elements for the vertical part of the pipe.

Materials:

As in [3,9], a bi-linear constitutive law is used and is based on the following mechanical characteristics: Young's modulus $E = 180 \times 10^3$ MPa, Poisson's ratio $\nu = 0.33$, density $\rho_0 = 7800$ kg/m³, yield stress $\sigma_Y = 155$ MPa, yield strain $\epsilon_Y = 0.086$ %, ultimate stress $\sigma_r = 475$ MPa and deformation at rupture $\epsilon_r = 25$ %.

Boundary Conditions – Loading:

The free end of the vertical pipe is initially closed by a rigid membrane separating the internal volume of the pipe from the external atmosphere. The pipe is rigidly connected to the vessel by means of a flange. The tank and the pipe are filled with hot liquid water at rest, at pressure $p = 166$ bar and at temperature $T = 326^\circ\text{C}$. In the present computation, an instantaneous break of the membrane is assumed and the Moody's critical flow model [10] is used at the free end of the vertical pipe. The vessel at the inlet of the pipe is modeled using a tank boundary condition.

Results:

The computation is performed with the Courant number $C = 0.8$. The numerical results obtained with the 1-D FSI numerical approach are first compared with the experimental data for test 45. The pipe motion is well reproduced in the simulation.

In contrast, the impact load is overestimated. This is due to the reduced kinematics of the beam element, i.e. the variation of the cross-section is not considered. This was also observed in the previous study [11,3] for test 44. In order to circumvent this problem, 3-D elbow modeling could be applied as in [11,3] where the impact zone is modeled using shell elements that enable to naturally take into account the local crushing of the pipe.

References:

[1] Europlexus verification et validation (V&V) report : thermal-hydraulic fluid transients, 3125-1721-2020-03020-FR Technical Report, EDF R&D Paris Saclay, 2020, F. Daude, T. Douillet-Grellier, P.Galon.

[2] J. L. Garcia, P. Chouard, and E. Sermet. Experimental studies of pipe impact on rigid restraints and concrete slabs. Nuclear Engineering and Design, 77:357–368, 1984.

- [3] S. Potapov and P. Galon. Modelling of Aquitaine II pipe whipping test with the plexus fast dynamics code. Nuclear Engineering and Design, 235(17-19):2045–2054, 2005.
- [4] P. Caumette, J.L. Garcia, Experimental studies of PWR primary piping under LOCA conditions. Nuclear Engineering and Design 61 (1980) 197-208.
- [5] J.L. Garcia, P. Caumette, J.L. Huet, A. Martin, Study of pipe whip and impact. SMIRT 1981
- [6] A. Martin, E. Sermet, J.L. Garcia, AQUITAINE II complémentaire : Rapport final d'essais. Note technique DRE/STRE/LMA 84/362
- [7] J.L. Garcia, "AQUITAINE II - Interprétation" : analyses d'essais avec chocs sur structures rigides. Note technique DRE/STRE/LMA 84/373
- [8] A. Ricard, A. Millard, Fouettement des tuyauteries par le programme TEDEL. Rapport DDMT/SMTS/BAMS/81-52
- [9] P. Galon, M. Lepareux, Comportement dynamique des tuyauteries. Analyse d'un essai de fouettement au moyen de PLEXUS. Rapport DDMT/SEMT/DYN/RT/99-028, 1999.
- [10] F. J. Moody. Maximum flow rate of a single component two-phase mixture. Journal of Heat Transfer, 87:134–141, 1965.
- [11] P. Galon, M. Lepareux, Couplage 1D/3D: analyse d'un essai de fouettement au moyen de EUROPLEXUS. Rapport DRN/DMT/SEMT/DYN/ RT/00.015/A

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 35/35

VL_EDF_POPI_MPI

Description

Title:

Pipe-On-Pipe-Impact test

Problem:

This case simulates a pipe-on-pipe impact test with the aim to assess the consequences of a potential pipe whip and to reproduce pipes section deformations due to impact. Numerical results are compared with the experimental ones reported in [1].

The experimental setup is composed of two pipes in a perpendicular position. The first one, referred to as the missile pipe, is a rotating pipe system with a hinged one end extremity while the other end extremity is free. The second pipe, referred to as the target pipe, is clamped at its both ends. An initial angular velocity is given to the missile pipe which rotates about the pivot and strikes the impact point located at the middle of the target pipe where the speed of the missile pipe is maximal.

The experimental results consist of measuring the diameter reductions of the missile and the target pipes at the impact point, as well as deflection of the lowest generator point of the target pipe.

Functionalities involved:

- Finite elements: Q4GS, POUT
- Materials: VMIS ISOT
- Links: TUYA, GLIS, BLOQ

Mesh:

A mixed 1D/3D model is used. The target and the missile pipes are modeled using a combination of POUT beam elements and Q4GS thick plate elements. The plate elements are only used in the area of interest where the contact occurs between the pipes in order to correctly produce the pipe deformations.

Materials:

Elasto-plastic isotropic Von Mises behavior is adopted for both pipes, for both beam and shell parts.

Links:

The continuity constraint at the 1D/3D interfaces is ensured via the directive TUYA which writes the mechanical relations between pipeline meshed by beams and pipeline meshed by thin shells.

Boundary and initial conditions:

To reproduce the rotational motion of the missile pipe with an initial velocity once it is released, the directive ROTA is used by specifying the rotational velocity $\omega = 7$ rad/s and the rotational axis defined by the coordinates of the origin point.

A quasi-static damping is activated after impact in order to obtain a static solution.

Results:

The diameter reduction and the vertical displacement of the central point of the target pipe are compared to experimental values.

Large discrepancy between the numerical and experimental results can be explained by the fact that the real boundary conditions on the target pipe are not known in the experiment, being supposed rigid and fixed in this simulation.

References:

[1] J. Yang, G. Lu, T. Yu, and S. Reid. Experimental study and numerical simulation of pipe-on-pipe impact. International Journal of Impact Engineering, 36(10-11):1259–1268, Oct. 2009.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en MPI.

Qualifications :

OK : 3/3

VL_EDF_ROMANDER_COUDE_1D_VF_FP_E_101

Description*Title:*

Non-linear response of a flexible straight water-filled pipe with an elbow to a pressure pulse (FP-E-101 free surface end).

Problem:

This test aims at verifying the capability of the 1-D approach to predict the non-linear dynamic response of a water-filled pipe to a strong pressure pulse. This experiment was performed by SRI (Stanford Research Institute). The test set-up consists of a water-filled piping system containing a single elbow.

The piping system consists of two flexible 1.5-m-long thin-walled ($\delta = 1.65$ mm) Nickel-200 pipes (with an inner diameter of $d = 72.94$ mm) connected in series through a 90-deg thick-walled stainless steel elbow with a 114.3 mm radius of curvature. The pipe ends and the elbow were fixed to the ground in order to eliminate pipe motion. The pipeline is filled with liquid water at rest, at ambient temperature and atmospheric pressure. Pressure pulses were produced by an explosively driven pulse gun developed at SRI. At one of the closed ends, a pressure pulse of 3 ms duration and 0.2 ms rise time was generated by means of an explosion. The pressure pulse reached values up to 16 MPa, causing local plastic wall deformation in the nickel pipe, which was extensively instrumented with pressure transducers and strain gauges. The test configuration corresponds to elbow test FP-E-101. From left to right, the system consists of the pulse gun, a thick-walled steel pipe, a horizontal nickel-pipe test section, a thick-walled elbow, another vertical nickel-pipe test section, and a heavy flange ending in a free surface. The steel and nickel pipes have the same dimensions as in the straight-pipe test. As also observed in the straight-pipe SRI experiment, due to the high pressure magnitude of the considered fluid transient, the Nickel pipe is plastically deformed. The interaction between the pressure pulse with the plastically deforming Nickel pipe induces a rarefaction wave which causes cavitation.

Mesh:

800 cells (TUVF) are used for the fluid domain in the steel pipe, 80 elements (TYVF) for the fluid/structure domain in the horizontal Nickel pipe, 11 elements (TYVF) for the fluid/structure domain in the elbow and 80 elements (TYVF) for the fluid/structure domain in the vertical Nickel pipe. A junction cell is used for the connection between the steel and the Nickel pipes. In the present computation, the elbow is assumed to be flexible but rigidly connected to the horizontal and vertical Nickel pipes. The computation is performed with the Courant number of 0.95.

Materials:

The stainless steel pipe is supposed to be rigid. The present computation considers the FSI coupling using flexible pipeline elements to represent the Nickel 200 pipe only. In this computation, the variation of the inner diameter of the pipe due to the plastic and elastic deformations of the tube wall is also considered. Note that accounting for the pipe wall elasticity makes it possible to recover the correct pressure wave speed in the internal fluid flowing the flexible pipe.

The mechanical characteristics used for Nickel 200 are: Young's modulus $E = 137.8 \times 10^3$ MPa, Poisson's ratio 0.3, density $\rho_s = 8890$ kg/m³, yield stress 82.7 MPa and yield strain 0.06 %. The piecewise linear strain-stress curve issued from reference is used for the von-Mises-type elastic-plastic behavior of Nickel 200.

The fluid initial conditions are an initial pressure $p = 1$ bar and temperature $T = 20^\circ\text{C}$ which corresponds to the fluid density $\rho = 998.29$ kg/m³.

Boundary Conditions – Loading:

The free surface at the free end of the Nickel pipe is modeled using an absorbing boundary condition whereas its left end is rigidly embedded at the junction with the steel pipe.

The pressure pulse generation was modelled by the direct application of the pressure history measured at P1 (see references) in the experiments following the imposed pressure boundary condition.

Results:

The evolution during 12 ms of the coupled pipe-fluid system is studied. The calculation allows a correct reproduction of the pressure wave attenuation due to deformation of the flexible pipe. There is also a good agreement between the corresponding results and the experimental reference in particular on the time of occurrence of the pressure peak.

The comparison between the measured mean values of the plastic strain and the numerical results is rather good, especially near the bride that interconnects the flexible and rigid pipes.

References:

- Experiments on the response of flexible piping systems to internal pressure pulses. Technical Report SRI Project PYD-1960, Stanford Research Institute, 1976, C. M. Romander and D. J. Cagliostro.
- Response of water-filled thin-walled pipes to pressure pulses: Experiments and analysis. ASME Journal of Pressure Vessel Technology, 102(1):56–61, 1980, C. M. Romander, L. E. Schwer, and D. J. Cagliostro. R.
- Europlexus verification and validation (V&V) report : thermal-hydraulic fluid transients, 3125-1721-2020-03020-FR Technical Report, EDF R&D Paris Saclay, 2020, F. Daude, T. Douillet-Grellier, P.Galon.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 34/34

VL_EDF_ROMANDER_COUDE_1D_VF_FP_E_103

Description*Title:*

Non-linear response of a flexible straight water-filled pipe with an elbow to a pressure pulse (FP-E-103 blind end).

Problem:

This test aims at verifying the capability of the 1-D approach to predict the non-linear dynamic response of a water-filled pipe to a strong pressure pulse. This experiment was performed by SRI (Stanford Research Institute). The test set-up consists of a water-filled piping system containing a single elbow.

The piping system consists of two flexible 1.5-m-long thin-walled ($\delta = 1.65$ mm) Nickel-200 pipes (with an inner diameter of $d = 72.94$ mm) connected in series through a 90-deg thick-walled stainless steel elbow with a 114.3 mm radius of curvature. The pipe ends and the elbow were fixed to the ground in order to eliminate pipe motion. The pipeline is filled with liquid water at rest, at ambient temperature and atmospheric pressure. Pressure pulses were produced by an explosively driven pulse gun developed at SRI. At one of the closed ends, a pressure pulse of 3 ms duration and 0.2 ms rise time was generated by means of an explosion. The pressure pulse reached values up to 16 MPa, causing local plastic wall deformation in the nickel pipe, which was extensively instrumented with pressure transducers and strain gauges. The test configuration corresponds to elbow test FP-E-103. From left to right, the system consists of the pulse gun, a thick-walled steel pipe, a horizontal nickel-pipe test section, a thick-walled elbow, another vertical nickel-pipe test section with a heavy blind flange that terminates the layout. The steel and nickel pipes have the same dimensions as in the straight-pipe test. As also observed in the straight-pipe SRI experiment, due to the high pressure magnitude of the considered fluid transient, the Nickel pipe is plastically deformed. The interaction between the pressure pulse with the plastically deforming Nickel pipe induces a rarefaction wave which causes cavitation.

Mesh:

800 cells (TUVF) are used for the fluid domain in the steel pipe, 80 elements (TYVF) for the fluid/structure domain in the horizontal Nickel pipe, 11 elements (TYVF) for the fluid/structure domain in the elbow and 80 elements (TYVF) for the fluid/structure domain in the vertical Nickel pipe. A junction cell is used for the connection between the steel and the Nickel pipes. In the present computation, the elbow is assumed to be flexible but rigidly connected to the horizontal and vertical Nickel pipes. The computation is performed with the Courant number of 0.95.

Materials:

The stainless steel pipe is supposed to be rigid. The present computation considers the FSI coupling using flexible pipeline elements to represent the Nickel 200 pipe only. In this computation, the variation of the inner diameter of the pipe due to the plastic and elastic deformations of the tube wall is also considered. Note that accounting for the pipe wall elasticity makes it possible to recover the correct pressure wave speed in the internal fluid flowing the flexible pipe.

The mechanical characteristics used for Nickel 200 are: Young's modulus $E = 137.8 \times 10^3$ MPa, Poisson's ratio 0.3, density $\rho_s = 8890$ kg/m³, yield stress 82.7 MPa and yield strain 0.06 %. The piecewise linear strain-stress curve issued from reference is used for the von-Mises-type elastic-plastic behavior of Nickel 200.

The fluid initial conditions are an initial pressure $p = 1$ bar and temperature $T = 20^\circ\text{C}$ which corresponds to the fluid density $\rho = 998.29$ kg/m³.

Boundary Conditions – Loading:

The blind end of the Nickel pipe is modeled using a wall boundary condition whereas its left end is rigidly embedded at the junction with the steel pipe.

The pressure pulse generation was modelled by the direct application of the pressure history measured at P1 (see references) in the experiments following the imposed pressure boundary condition.

Results:

The evolution during 12 ms of the coupled pipe-fluid system is studied. The calculation allows a correct reproduction of the pressure wave attenuation due to deformation of the flexible pipe. There is also a good agreement between the corresponding results and the experimental reference in particular on the time of occurrence of the pressure peak.

The comparison between the measured mean values of the plastic strain and the numerical results is rather good, especially near the bride that interconnects the flexible and rigid pipes.

References:

- Experiments on the response of flexible piping systems to internal pressure pulses. Technical Report SRI Project PYD-1960, Stanford Research Institute, 1976, C. M. Romander and D. J. Cagliostro.
- Response of water-filled thin-walled pipes to pressure pulses: Experiments and analysis. ASME Journal of Pressure Vessel Technology, 102(1):56–61, 1980, C. M. Romander, L. E. Schwer, and D. J. Cagliostro. R.
- Europlexus verification and validation (V&V) report : thermal-hydraulic fluid transients, 3125-1721-2020-03020-FR Technical Report, EDF R&D Paris Saclay, 2020, F. Daude, T. Douillet-Grellier, P.Galon.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 34/34

VL_EDF_ROMANDER_DROIT_1D_VF_ELAS

Description*Title:*

Non-linear response of a flexible straight water-filled pipe to a pressure pulse

Problem:

This test aims at verifying the capability of the 1-D approach to predict the non-linear dynamic response of a water-filled pipe to a strong pressure pulse. This experiment was performed by SRI (Stanford Research Institute). The test set-up consists of a water-filled piping system composed by two co-axial straight pipes rigidly interconnected and clumped to a thick-walled concrete pad at their extremities in order to prevent pipe motion.

The experimental configuration consists of a pulse gun connected to a 3-m-long thick-walled ($\delta = 4.78$ mm) stainless steel pipe, which is connected with a heavy flange to a 1.5-m-long thin-walled ($\delta = 1.65$ mm) Nickel 200 pipe. The piping system is filled with liquid water at rest, at ambient temperature and atmospheric pressure. The internal diameter of both pipes is $d = 72.94$ mm. An explosively driven pulse gun installed near the embedded inlet boundary of the thick-walled steel pipe produces a strong pressure pulse with a pressure peak of 13.8 MPa, a rise time of 0.15 ms and a duration about 3 ms.

The material properties and dimensions of steel and Nickel pipes are such that the steel pipe responds elastically to the pressure pulse whereas the Nickel pipe presents local plastic deformations. As soon as the pressure wave enters the flexible Nickel 200 pipe, its amplitude is suddenly attenuated by the radial pipe expansion with large but local plastic deformations. Those generate a reflected pressure wave propagating back through the rigid part of the pipe. Due to this reflected wave, the pressure drops to the saturation pressure leading to an inception of cavitation.

Mesh:

800 cells (TUVF) are used for the fluid domain in the steel pipe and 80 elements (TYVF) for the fluid/structure domain in the Nickel pipe. A junction cell is used for the connection between the steel and the Nickel pipes. The computation is performed with the Courant number of 0.95.

Materials:

The stainless steel pipe is supposed to be rigid. The present computation considers the FSI coupling using flexible pipeline elements to represent the Nickel 200 pipe only. In this computation, the variation of the inner diameter of the pipe due to the plastic and elastic deformations of the tube wall is also considered. Note that accounting for the pipe wall elasticity makes it possible to recover the correct pressure wave speed in the internal fluid flowing the flexible pipe.

The mechanical characteristics used for Nickel 200 are: Young's modulus $E = 137.8 \times 10^3$ MPa, Poisson's ratio 0.3, density $\rho_s = 8890$ kg/m³, yield stress 82.7 MPa and yield strain 0.06 %. The piecewise linear strain-stress curve issued from reference is used for the von-Mises-type elastic-plastic behavior of Nickel 200.

The fluid initial conditions are an initial pressure $p = 1$ bar and temperature $T = 20^\circ\text{C}$ which corresponds to the fluid density $\rho = 998.29$ kg/m³.

Boundary Conditions – Loading:

The right end of the Nickel pipe is a fixed wall boundary condition whereas its left end is rigidly embedded at the junction with the steel pipe.

The pressure pulse generation was modelled by the direct application of the pressure history measured at P1 (see references) in the experiments following the imposed pressure boundary condition.

Results:

The evolution during 5 ms of the coupled pipe-fluid system is studied. The calculation allows a correct reproduction of the pressure wave attenuation due to deformation of the flexible pipe. There is also a good agreement between the corresponding results and the experimental reference in particular on the time of occurrence of the pressure peak.

The comparison between the measured mean values of the plastic strain and the numerical results is rather good, especially near the bride that interconnects the flexible and rigid pipes.

References:

- Experiments on the response of flexible piping systems to internal pressure pulses. Technical Report SRI Project PYD-1960, Stanford Research Institute, 1976, C. M. Romander and D. J. Cagliostro.
- Response of water-filled thin-walled pipes to pressure pulses: Experiments and analysis. ASME Journal of Pressure Vessel Technology, 102(1):56–61, 1980, C. M. Romander, L. E. Schwer, and D. J. Cagliostro. R.
- Europlexus verification and validation (V&V) report : thermal-hydraulic fluid transients, 3125-1721-2020-03020-FR Technical Report, EDF R&D Paris Saclay, 2020, F. Daude, T. Douillet-Grellier, P.Galon.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 18/18

VL_EDF_PIPE_WHIP_REID_T4_1D

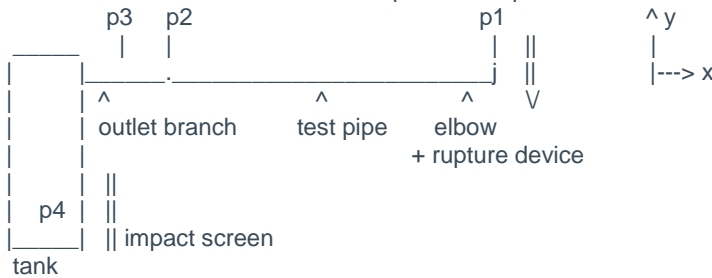
Description

Title:

Large deflection pipe whip test - 1D

Problem:

This test is a full 1D Finite Volume computation. Pipe structural motion is modelled through coupled VF1D/beam TYVF elements.



The experiment consists in a large tank filled with pressurized air (Initial pressure = 62 bar), its outlet branch being connected to the test pipe. The test pipe is a 3m-long and 50.8mm-diameter straight cantilever pipe with a 90° elbow at its extremity. Its free end is sealed with a rupture device. A protective plate is set near the tank where the pipe will impact when whipping.

A 1.8kg tipping mass is added at the extremity. When the rupture happens, air is expelled at the extremity of the pipe, then it bends and impacts the protective screen. 4 pressure sensors are placed along the tube : p1 at the extremity of the test pipe, p2 at its beginning, p3 in the outlet branch and p4 inside the tank.

We consider the test4 configuration with the following parameters :

Pipe length (m)	3.0
Outlet branch length (m)	0.975
Outer diameter (mm)	50.8
Pipe thickness (mm)	2.6
Tipping mass (kg)	1.8
Air pressure (bar)	62.1
Tank volume (m3)	0.28

Tested functionalities:

- VFCC-1D elements: TUVF, TYVF
- bifurcations, cavities: CAVF, BIVF
- Critical mass flow rate outlet: DCRI

Mesh:

The mesh is fully made of VF 1D elements (SEG2 elements in Cast3m) :

- 1 cavity (CAVF) element for the tank
- 32 fixed pipeline (TUVF) elements for the outlet branch
- 95 deformable pipeline (TYVF) elements, superposition of TUVF and POUT elements, 90 in the straight pipe, 5 in the extremity elbow
- 1 bifurcation (BIVF) element to connect the outlet branch and the test pipe
- 1 1D-boundary CLTU element at the extremity of the pipe to set the outflow.

Materials:

The fluid is air, modeled with a perfect gas (GAZP) equation of state. An elasto-plastic isotropic Von Mises behavior (VMIS ISOT) is used for the structure of the test pipe: $E = 200 \text{ GPa}$, $\nu = 0.3$, yield stress = 295 Mpa.

As stress-strain plasticity curve we use the one in the paper, given by the following law: $\sigma(\text{MPa}) = 860 \cdot \epsilon_p^{0.26}$.

Wall laws (PARO) are used in TUVF and TYVF elements to account for regular head losses. The outflow is modeled as a critical mass flow rate (CLVF DCRI). The tipping mass is added with the MASS directive at the end point of the tube.

Links:

All dofs are blocked (LINK COUP BLOQ) at the embedding (first node of the test pipe, connected to the outlet branch).

Initial conditions and computation:

Initial pressure is set at 62 bar in the whole system. Air flows out at the elbow, which causes the bending of the tube. The computation is carried with a Finite Volume formulation, 1st order in time and space. Final time is 100 ms.

Post-treatment:

Coordinates of some nodes along the pipe are recorded at regular time steps in order to plot the x-y profile of the bended pipe and compare it to the experimental results. Fluid pressure at the 4 sensors is plotted and compared to the experimental values.

Results:

For each time stamp, the experimental and the numerical profiles are quite different. Plasticity occurs mostly at the embedding and the curvature along the pipe remains much lower with numerical results. The flexible pipe is modelled as a beam which cannot take into account ovalization of the pipe when bending. A 3D computation will then be necessary to consider this phenomenon and get a more accurate deformation profile.

The Pressure evolution inside the tank (p4) is concordant between numerical and experimental results. The critical mass flow rate outlet and head losses coefficients set along the pipe seem to properly account for the evacuating airflow.

The pressure drop for sensor p1 (rupture point) is consistent with the critical mass flow rate outlet, but it cannot get the experimental values. The oscillation obtained at around 20 ms seems to be a numerical artefact. The pressure profile obtained for sensors p2 and p3 are concordant when the pressure drop reaches them, however pressure gets higher afterwards, contrary to experiments. With the 1D modelling, the section remains constant so excessive ovalization at the embedding cannot explain this behavior.

References:

[1] Reid SR, Yu TX, Yang JL, Corbett GG. Dynamic elastic-plastic behaviour of whipping pipes: experiments and theoretical model. International Journal of Impact Engineering 1996;18(No. 7-8):703-33

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 7/7

VL_EDF_PIPE_WHIP_REID_T4_3D_STRU_MPI

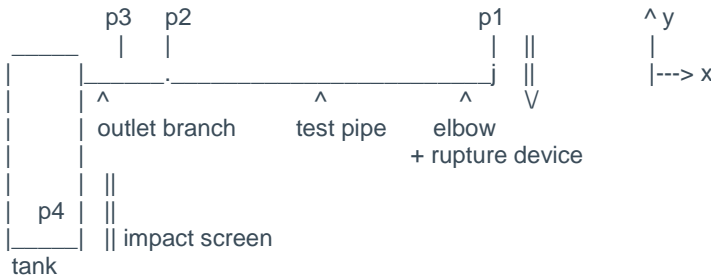
Description

Title:

Large deflection pipe whip test – Structural 3D test

Problem:

This test is a Finite Element structural computation. Fluid motion is not computed and its influence is modelled through a following force at the extremity of the test pipe. The test pipe is modelled in 3D through shell elements.



The experiment consists in a large tank filled with pressurized air (Initial pressure = 62 bar), its outlet branch being connected to the test pipe. The test pipe is a 3m-long and 50.8mm-diameter straight cantilever pipe with a 90° elbow at its extremity. Its free end is sealed with a rupture device. A protective plate is set near the tank where the pipe will impact when whipping.

A 1.8kg tipping mass is added at the extremity. When the rupture happens, air is expelled at the extremity of the pipe, then it bends and impacts the protective screen. 4 pressure sensors are placed along the tube: p1 at the extremity of the test pipe, p2 at its beginning, p3 in the outlet branch and p4 inside the tank.

We consider the test4 configuration with the following parameters:

Pipe length (m)	3.0
Outlet branch length (m)	0.975
Outer diameter (mm)	50.8
Pipe thickness (mm)	2.6
Tipping mass (kg)	1.8
Air pressure (bar)	62.1
Tank volume (m3)	0.28

In this case only the test pipe is modelled, as we are interested in the structural motion of the pipe submitted to a bending force.

Tested functionalities:

- Q4GS shell elements
- Following force FSUI
- TUYA shell-beam connection

Mesh:

The mesh is made of:

- 4320 Q4GS (4-nodes) elements to model the test pipe
- 6 TUYA elements for the elbow
- 1 CLTU element to apply the force

Materials:

An elasto-plastic isotropic Von Mises behavior (VMIS ISOT) is used for the structure of the test pipe: $E = 200$ GPa, $\nu = 0.3$, yield stress = 295 MPa. As stress-strain plasticity curve we use the one in the paper, given by the following law: $\sigma(\text{MPa}) = 860 \cdot \epsilon_p^{0.26}$. The outflow is modeled as a following force (IMPE FSUI). The profile of the force is given in the paper:

$$F(t) = 1000 * \begin{cases} 11.25 - 1171.88 * t & | t \leq 0.004 \\ 6.56 & | 0.004 < t \leq 0.017 \\ 6.56 + 935(t - 0.017) & | 0.017 < t \leq 0.022 \\ 11.26 - 192.3(t - 0.022) & | 0.022 < t \leq 0.035 \\ 8.75 - 15.63(t - 0.035) & | 0.035 < t \end{cases}$$

The tipping mass is added with the MASS directive at the end point of the tube.

Links:

All dofs are blocked (LINK COUP BLOQ) at the embedding (nodes at the left extremity of the pipe). The 7th DDL (fluid) is blocked in TUYA elements. A TUYA connection links the dofs of the extremity of the shell elements and the first node of the 1D elements.

Initial conditions and computation:

The force profile is applied at the extremity of the tube and the computation is run until final time = 60 ms.

Post-treatment:

Coordinates of some nodes along the pipe are recorded at regular time steps in order to plot the x-y profile of the bended pipe and compare it to the experimental results. Fluid pressure at the 4 sensors is plotted and compared to the experimental values.

Results:

The obtained bending profiles show strong similarities. The position of the pipe extremity given by EPX is a good estimate. The curvature is regular along the pipe and gets homogeneously higher with time. However, regarding the behavior at the beginning of the pipe, at the embedding, plasticity occurs at 50 ms with numerical results whereas the pipe remains tangent to the horizontal when whipping.

The way the pipe is embedded is not precisely described in the paper. Test pipes, with various diameters, can be inserted in the device. There may be some strengthening where the outlet branch and the test pipe connect.

References:

[1] Reid SR, Yu TX, Yang JL, Corbett GG. Dynamic elastic-plastic behaviour of whipping pipes: experiments and theoretical model. International Journal of Impact Engineering 1996;18(No. 7-8):703-33

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en MPI

Qualifications :

OK : 10/10

VL_EDF_PIPE_WHIP_REID_T4_3D_IFS_MPI

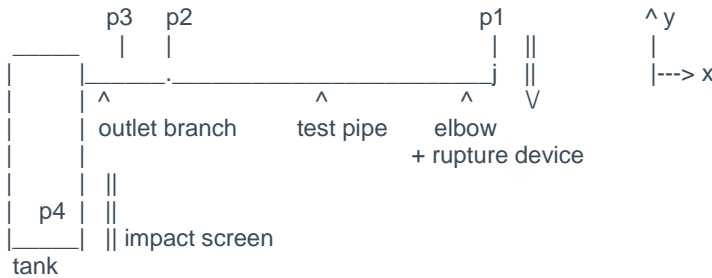
Description

Title:

Large deflection pipe whip test – 3D with IFS test

Problem:

This test is a 3D computation with Fluid-structure interaction. The test pipe is modelled through shell Finite Elements and fluid motion is modelled in Finite Volumes. Fluid and structure meshes are conformal and IFS is done with ALE.



The experiment consists in a large tank filled with pressurized air (Initial pressure = 62 bar), its outlet branch being connected to the test pipe. The test pipe is a 3m-long and 50.8mm-diameter straight cantilever pipe with a 90° elbow at its extremity. Its free end is sealed with a rupture device. A protective plate is set near the tank where the pipe will impact when whipping.

A 1.8kg tipping mass is added at the extremity. When the rupture happens, air is expelled at the extremity of the pipe, then it bends and impacts the protective screen. 4 pressure sensors are placed along the tube: p1 at the extremity of the test pipe, p2 at its beginning, p3 in the outlet branch and p4 inside the tank.

We consider the test4 configuration with the following parameters:

Pipe length (m)	3.0
Outlet branch length (m)	0.975
Outer diameter (mm)	50.8
Pipe thickness (mm)	2.6
Tipping mass (kg)	1.8
Air pressure (bar)	62.1
Tank volume (m3)	0.28

This test is a 1D-3D Finite Volume computation. Tank and fixed pipe are in 1D, and connected to the flexible pipe modelled in 3D

Tested functionalities:

- VFCC-1D elements (TUVF, TYVF)
- Cavities : CAVF
- Shell elements : Q4GS
- Critical mass flow rate outlet : DCRI
- 1D-3D connections : TUBM/TUYM

Mesh:

The mesh is made of :

- 1 cavity (CAVF) element for the tank
- 32 fixed pipeline (TUVF) elements for the outlet branch

- 5 flexible pipeline (TYVF) elements in the extremity elbow
- 1 1D-boundary CLTU element at the extremity of the pipe to set the outflow.
- ~4000 Q4GS elements for the pipe wall
- ~16000 CUVF elements for the interior fluid
- 1 fixed TUBM connection between the fixed pipe and the 3D fluid
- 1 flexible TUYM connection between the 3D flexible pipe and the 1D elbow.

3D fluid nodes are declared ALE and their motion follows the structure motion (CONTOUR directive).

Materials:

The fluid is air, modeled with a perfect gas (GAZP) equation of state.

An elasto-plastic isotropic Von Mises behavior (VMIS ISOT) is used for the structure of the test pipe: $E = 200$ GPa, $\nu = 0.3$, yield stress = 295 MPa. As stress-strain plasticity curve we use the one in the paper, given by the following law: $\sigma(\text{MPa}) = 860 \cdot \epsilon_p^{0.26}$. Wall laws (PARO) are used in TUVF and TYVF elements to account for regular head losses.

The outflow is modeled as a critical mass flow rate (CLVF DCRI). TAU parameter is set at $1e-4$ to reproduce the pressure drop at the extremity obtained experimentally.

The tipping mass is added with the MASS directive at the end point of the tube.

Links:

All dofs are blocked (LINK COUP BLOQ) at the embedding (first node of the test pipe, connected to the outlet branch)

Initial conditions and computation:

Initial pressure is set at 62 bar in the whole system. Air flows out at the elbow, which causes the bending of the tube. The computation is carried with a Finite Volume formulation for the fluid, 1st order in time & space. Final time is 60 ms.

Post-treatment:

Coordinates of some nodes along the pipe are recorded at regular time steps in order to plot the x-y profile of the bended pipe and compare it to the experimental results. Fluid pressure at the 4 sensors is plotted and compared to the experimental values.

Results:

The obtained profiles show that the curvature of the pipe is similar between numerical results and experiments, however EUROPLEXUS obtains an excessive plastic deformation at the embedding. If we "rotate" the profiles around the embedding point, the bending profiles would match. The excessive deformation in this area is much higher with this coupled computation than with the structure only computation. The way the embedding is set is the same in both configuration: all translation and rotation DDLs blocked for all nodes on the extremity circle of the pipe.

The large uncertainties in the design of the embedding system could explain the discrepancies.

The pressure levels for sensors p1, p2, p3 et p4 show a good concordance between Europlexus and the experimental results. However, there are some discrepancies. Numerically, pressure at sensor p1 (pipe outlet) drops right at the beginning of the computation whereas there is a slight shift experimentally. That can be explained as we don't know the exact position of the sensor in the pipe. For EPX, pressure is taken for the last element, near the critical flow rate outlet. The initial pressure drop obtained numerically is then lower than the experimental one. This is mostly dependent on the setting of the critical mass flow rate outlet (DCRI), where there are some parameters hard to set precisely, such as the section ratio and the head loss. Furthermore the DCRI impedance has been implemented for water models, and has not been tested for gas materials.

The pressure drops obtained for sensors p2 and p3 (before and after the connection between the fixed and flexible pipe) is concordant. Pressure gets then higher when approaching 60 ms, which can be explained with the bending profiles. The high plastic deformation of the pipe right after the embedding causes ovalization and probably restricts the air flow.

References:

[1] Reid SR, Yu TX, Yang JL, Corbett GG. Dynamic elastic-plastic behaviour of whipping pipes: experiments and theoretical model. International Journal of Impact Engineering 1996;18(No. 7-8):703-33

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en MPI

Qualifications :

OK : 14/14

VL_EDF_VTT-X1_GLRC_MPI

Description*Title:*

VTT-X1 impact test studied with GLRC DAMA material

Problem:

The VTT impact tests are RC slabs which are representative of existing NPP structures with a missile which can be assimilated to a small aircraft. The steel tube-like projectile has a length of 1.81m and a diameter of 0.256m. Its mass is around 50 kg and the impact velocity is 165.9 m/s.

The target is a RC slab of total dimensions 2.087m x 2.087m x 0.25m with:

- longitudinal reinforcement ratio of 8.7 cm²/m²,
- shear reinforcement ratio of 17.1 cm²/m².

Displacement measurements are used as reference solution.

Mesh:

The finite element mesh is composed of quadrangular shells (Q4GS).

Materials:

Global non-linear GLRC DAMA material law is used.

Boundary conditions and loading:

The slab is simply supported by rigid line-type supports restraining out-of-plane displacements of the slab.

An impact force is uniformly applied to a disc of 0.4m radius at the center of the slab. The Riera's method is used to define the impact force evolution trend.

No structural damping is added.

Results:

At the time $t = 1.0E-2$ s, the displacement of point D1 in the impact direction is compared with experimental value.

Reference:

This validation test is fully described in the EDF R&D report 6125-1721-2022-02038-EN entitled "Verification & Validation report for the AirPlane Crash calculations".

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en MPI

Qualifications :

OK : 2/2

VL_EDF_VTT-X2_GLRC_MPI

Description*Title:*

VTT-X2 impact test studied with GLRC DAMA material

Problem:

The VTT impact tests are RC slabs which are representative of existing NPP structures with a missile which can be assimilated to a small aircraft. The steel tube-like projectile has a length of 1.81m and a diameter of 0.256m. Its mass is around 50 kg and the impact velocity is 165.9 m/s.

The target is a RC slab of total dimensions 2.087m x 2.087m x 0.25m with:

- longitudinal reinforcement ratio of 8.7 cm²/m²,
- shear reinforcement ratio of 11.7 cm²/m².

Displacement measurements are used as reference solution.

Mesh:

The finite element mesh is composed of quadrangular shells (Q4GS).

Materials:

Global non-linear GLRC DAMA material law is used.

Boundary conditions and loading:

The slab is simply supported by rigid line-type supports restraining out-of-plane displacements of the slab.

An impact force is uniformly applied to a disc of 0.4m radius at the center of the slab. The Riera's method is used to define the impact force evolution trend.

No structural damping is added.

Results:

At the time $t = 1.0E-2$ s, the displacement of point D1 in the impact direction is compared with experimental value.

Reference:

This validation test is fully described in the EDF R&D report 6125-1721-2022-02038-EN entitled "Verification & Validation report for the AirPlane Crash calculations".

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en MPI

Qualifications :

OK : 2/2

VL_EDF_VTT-X3_GLRC_MPI

Description*Title:*

VTT-X3 impact test studied with GLRC DAMA material

Problem:

The VTT impact tests are RC slabs which are representative of existing NPP structures with a missile which can be assimilated to a small aircraft. The steel tube-like projectile has a length of 1.3 m and a diameter of 0.22 m. Its mass is around 50 kg and the impact velocity is 165.9 m/s.

The target is a RC slab of total dimensions 2.087m x 2.087m x 0.25m with:

- longitudinal reinforcement ratio of 8.7 cm²/m²,
- shear reinforcement ratio of 17.1 cm²/m².

Displacement measurements are used as reference solution.

Mesh:

The finite element mesh is composed of quadrangular shells (Q4GS).

Materials:

Global non-linear GLRC DAMA material law is used.

Boundary conditions and loading:

The slab is simply supported by rigid line-type supports restraining out-of-plane displacements of the slab.

An impact force is uniformly applied to a disc of 0.4m radius at the center of the slab. The Riera's method is used to define the impact force evolution trend.

No structural damping is added.

Results:

At the time $t = 1.0E-2$ s, the displacement of point D1 in the impact direction is compared with experimental value.

Reference:

This validation test is fully described in the EDF R&D report 6125-1721-2022-02038-EN entitled "Verification & Validation report for the AirPlane Crash calculations".

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en MPI

Qualifications :

OK : 2/2

VL_FRA_1D_SUPERCANON30

Description*Title*

Supercanon experiments - 30%

Object

The SuperCanon experiments are considered to assess the capability of EUROPLEXUS to model membrane break and open boundary conditions, in addition to wave propagation and reflection in pressurized piping. The experimental setup consists of 4.389 m straight pipe (internal diameter of 102.3 mm), initially heated and pressurized at 300C and 150bar. One end of the pipe is closed while the other end is blocked by a membrane that is broken at a given instant, triggering the transient phenomenon.

Geometry

The experimental setup consists of 4.389 m straight pipe (internal diameter of 102.3 mm)

Mesh

The pipe is modeled by 1D elements TUVF. Number of elements : 1000

Material

The material EAU is used.

Boundary Conditions

The membrane break is modeled using the DCRI directive with the following parameters:

Boundary flow model : Athis-like

Outlet pressure : 1 Bar

Section Ratio : 0.3

Results-Qualification

The computation is performed with the Courant number $C = 0.25$. The numerical results are compared to the experimental data. Good agreement is obtained for the pressure history at different locations along the pipe (P1 to P6). Good agreement is obtained for the void fraction at one location (PA).

Reference

B. Riegel, Contribution à l'étude de la décompression d'une capacité en régime diphasique, *Université Scientifique et Médicale et Institut National Polytechnique de Grenoble*, 1978.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en séquentiel

Qualifications :

OK : 35/35

VL_FRA_1D_SUPERCANON50

Description*Title*

Supercanon experiments - 50%

Object

The SuperCanon experiments are considered to assess the capability of EUROPLEXUS to model membrane break and open boundary conditions, in addition to wave propagation and reflection in pressurized piping. The experimental setup consists of 4.389 m straight pipe (internal diameter of 102.3 mm), initially heated and pressurized at 300C and 150bar. One end of the pipe is closed while the other end is blocked by a membrane that is broken at a given instant, triggering the transient phenomenon.

Geometry

The experimental setup consists of 4.389 m straight pipe (internal diameter of 102.3 mm)

Mesh

The pipe is modeled by 1D elements TUVF. Number of elements : 1000

Material

The material EAU is used.

Boundary Conditions

The membrane break is modeled using the DCRI directive with the following parameters:

Boundary flow model : Athis-like

Outlet pressure : 1 Bar

Section Ratio : 0.5

Results-Qualification

The computation is performed with the Courant number $C = 0.5$. The numerical results are compared to the experimental data. Good agreement is obtained for the pressure history at different locations along the pipe (P1 to P6). Good agreement is obtained for the void fraction at one location (PA).

Reference

B. Riegel, Contribution à l'étude de la décompression d'une capacité en régime diphasique, *Université Scientifique et Médicale et Institut National Polytechnique de Grenoble*, 1978.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en séquentiel

Qualifications :

OK : 35/35

VL_FRA_1D_SUPERCANON70

Description*Title*

Supercanon experiments - 70%

Object

The SuperCanon experiments are considered to assess the capability of EUROPLEXUS to model membrane break and open boundary conditions, in addition to wave propagation and reflection in pressurized piping. The experimental setup consists of 4.389 m straight pipe (internal diameter of 102.3 mm), initially heated and pressurized at 300C and 150bar. One end of the pipe is closed while the other end is blocked by a membrane that is broken at a given instant, triggering the transient phenomenon.

Geometry

The experimental setup consists of 4.389 m straight pipe (internal diameter of 102.3 mm)

Mesh

The pipe is modeled by 1D elements TUVF. Number of elements : 1000

Material

The material EAU is used.

Boundary Conditions

The membrane break is modeled using the DCRI directive with the following parameters:

Boundary flow model : Athis-like

Outlet pressure : 1 Bar

Section Ratio : 0.7

Results-Qualification

The computation is performed with the Courant number $C = 0.5$. The numerical results are compared to the experimental data. Good agreement is obtained for the pressure history at different locations along the pipe (P1 to P6). Good agreement is obtained for the void fraction at one location (PA).

Reference

B. Riegel, Contribution à l'étude de la décompression d'une capacité en régime diphasique, *Université Scientifique et Médicale et Institut National Polytechnique de Grenoble*, 1978.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en séquentiel

Qualifications :

OK : 35/35

VL_FRA_1D_SUPERCANON100

Description*Title*

Supercanon experiments - 100%

Object

The SuperCanon experiments are considered to assess the capability of EUROPLEXUS to model membrane break and open boundary conditions, in addition to wave propagation and reflection in pressurized piping. The experimental setup consists of 4.389 m straight pipe (internal diameter of 102.3 mm), initially heated and pressurized at 300C and 150bar. One end of the pipe is closed while the other end is blocked by a membrane that is broken at a given instant, triggering the transient phenomenon.

Geometry

The experimental setup consists of 4.389 m straight pipe (internal diameter of 102.3 mm)

Mesh

The pipe is modeled by 1D elements TUVF. Number of elements : 1000

Material

The material EAU is used.

Boundary Conditions

The membrane break is modeled using the DCRI directive with the following parameters:

Boundary flow model : Athis-like

Outlet pressure : 1 Bar

Section Ratio : 1.0

Results-Qualification

The computation is performed with the Courant number $C = 0.5$. The numerical results are compared to the experimental data. Good agreement is obtained for the pressure history at different locations along the pipe (P1 to P6). Good agreement is obtained for the void fraction at one location (PA).

Reference

B. Riegel, Contribution à l'étude de la décompression d'une capacité en régime diphasique, *Université Scientifique et Médicale et Institut National Polytechnique de Grenoble*, 1978.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en séquentiel

Qualifications :

OK : 35/35

VL_FRA_LAME_FLUIDE

Description*Title*

Fluid-Structure Interaction (FSI) in Finite Volumes and EQBM liaison

Object

The objective of this test case is to validate the transmission of Fluid-Structure Interaction (FSI) forces in finite volumes via « EQBM » liaison (Equivalent beam kinematics for continuum elements). The validation is carried out on a representative model of a reactor downcomer (fluid in the annular space between the Reactor Pressure Vessel and the Core Barrel). The FSI forces are compared to those of the analytical formulas of Fritz's theory [1].

Geometry

The model considered consists of two cylindrical shells (length 9 m) separated by a fluid blade (outside diameter 4.39 m and thickness 0.26 m). The fluid, which is modeled in finite volumes, is coupled with the cylindrical shells. Each cylindrical shell is connected to a master beam via « EQBM » directive. The beam connected to the outer cylinder will be called external beam and the beam connected to the inner cylinder will be called internal beam.

Mesh

The beams are modeled by « POUT » elements and are divided into 9 elements each. The beams are aligned according to the Y axis. The cylindrical shells are meshed with « Q4GS » elements with an axial discretization (Y) identical to that of the beams. The fluid volume is meshed with « CUVF » with an axial discretization (Y) four times finer than the shells and the beams. The angular discretization is identical between the fluid and the cylindrical shells. The fluid-shell interface is a non-conforming interface declared via « NCFS » keyword option.

Material

A phantom material « FANT » (without mass) is considered for beams and shells in order to obtain only forces due to the fluid-structure interaction without being parasitized by the inertia forces of the structures. The sections of the beams and the thicknesses of the shells are chosen arbitrarily since rigid body movements are imposed to the overall structure (no structure deformation). The material used to model the fluid is the « FLUID » material with a density $\rho = 1000 \text{ kg/m}^3$ and a speed of sound $c = 1500 \text{ m/s}$.

Boundary Conditions - Loading

All degrees of freedom of the beams are blocked except the UX displacement which is imposed on the external beam. The internal beam is thus blocked and a sinusoidal UX displacement is imposed on the external beam with a frequency of 20 Hz and an amplitude of $10\text{E-}8 \text{ m}$ for 6 seconds. An initial velocity is imposed on the moving beam in order to have zero initial acceleration. At both ends of the fluid domain ($Y = 0$ and $Y = 9 \text{ m}$), a zero normal speed in the Y direction is imposed (rigid wall conditions) in order to be consistent with the assumptions of the Fritz formula (flow perpendicular to the axis of the cylinders).

Results-Qualification

The FSI forces are analyzed for the last oscillations once the permanent regime is established. These forces correspond to the internal forces « FINT » of the shells which are transmitted to the beams via « EQBM » liaison. Thus, these forces correspond also to the liaisons forces « FLIA ». The amplitudes of the forces are compared to those given by analytical Fritz formula (Formula 8 of Table 1 of [1]). The differences between EUROPLEXUS and theoretical forces are less than 3% which is a good agreement.

Reference

- [1] Fritz R.J, The effects of liquids on the dynamic motions of immersed solids, Journal of Engineering for Industry, 94, 167-173, Feb. 1972

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en séquentiel

Qualifications : OK : 20/20

VL_FRA_LAME_FLUIDE_OPPPOSITION_PHASE

Description

Title

Fluid-Structure Interaction (FSI) in Finite Volumes and EQBM liaison

Object

The objective of this test case is to validate the transmission of Fluid-Structure Interaction (FSI) forces in finite volumes via « EQBM » liaison (Equivalent beam kinematics for continuum elements). The validation is carried out on a representative model of a reactor downcomer (fluid in the annular space between the Reactor Pressure Vessel and the Core Barrel). The FSI forces are compared to those of the analytical formulas of Fritz's theory [1].

Geometry

The model considered consists of two cylindrical shells (length 9 m) separated by a fluid blade (outside diameter 4.39 m and thickness 0.26 m). The fluid, which is modeled in finite volumes, is coupled with the cylindrical shells. Each cylindrical shell is connected to a master beam via « EQBM » directive. The beam connected to the outer cylinder will be called external beam and the beam connected to the inner cylinder will be called internal beam.

Mesh

The beams are modeled by « POUT » elements and are divided into 9 elements each. The beams are aligned according to the Y axis. The cylindrical shells are meshed with « Q4GS » elements with an axial discretization (Y) identical to that of the beams. The fluid volume is meshed with « CUVF » with an axial discretization (Y) four times finer than the shells and the beams. The angular discretization is identical between the fluid and the cylindrical shells. The fluid-shell interface is a non-conforming interface declared via « NCFS » keyword option.

Material

A phantom material « FANT » (without mass) is considered for beams and shells in order to obtain only forces due to the fluid-structure interaction without being parasitized by the inertia forces of the structures. The sections of the beams and the thicknesses of the shells are chosen arbitrarily since rigid body movements are imposed to the overall structure (no structure deformation). The material used to model the fluid is the « FLUID » material with a density $\rho = 1000 \text{ kg/m}^3$ and a speed of sound $c = 1500 \text{ m/s}$.

Boundary Conditions - Loading

All degrees of freedom of the beams are blocked except the UX displacement which is imposed on each beam. A sinusoidal UX displacement is imposed on each internal and external beam in phase opposition with a frequency of 20 Hz and an amplitude of $10E-8 \text{ m}$ for 6 seconds. An initial velocity is imposed on the moving beams in order to have zero initial acceleration. At both ends of the fluid domain ($Y = 0$ and $Y = 9 \text{ m}$), a zero normal speed in the Y direction is imposed (rigid wall conditions) in order to be consistent with the assumptions of the Fritz formula (flow perpendicular to the axis of the cylinders).

Results-Qualification

The FSI forces are analyzed for the last oscillations once the permanent regime is established. These forces correspond to the internal forces « FINT » of the shells which are transmitted to the beams via « EQBM » liaison. Thus, these forces correspond also to the liaisons forces « FLIA ». The amplitudes of the forces are compared to those given by analytical Fritz formula (Formula 8 of Table 1 of [1]). The differences between Europlexus and theoretical forces are less than 3% which is a good agreement.

Reference

- [1] Fritz R.J, The effects of liquids on the dynamic motions of immersed solids, Journal of Engineering for Industry, 94, 167-173, Feb. 1972

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en séquentiel

Qualifications : OK : 20/20

VL_FRA_POUTRE_EPAISSE_COQUES

Description*Title*

Modelling « thick » beams using Q4GS shell elements

Object

The objective of this test case is to validate the modelling of so-called «thick» beams (that is beams for which the shear deformation is no longer negligible w.r.t the bending deformation) using rows of shell elements (Q4GS). EUROPLEXUS "POUT" beam elements do not take into account transverse shear stiffness, as Timoshenko beams would. In order to take into account the shear deformation, one solution is to use rows of "Q4GS" shell elements. The first four frequencies of the beam are compared to those of an equivalent ANSYS model, modeled with Timoshenko beam elements.

Geometry

A beam of length $L = 7.17$ m and rectangular section (height $h = 7.79$ m and width $b = 0.23$ m) is considered. Given these dimensions, it is clear that shear deformation of the section can not be neglected.

Mesh

The beam is modeled by Q4GS shell elements in the YZ plane. The length of the beam ($L = 7.17$ m) is according to the Y axis and the width ($b = 0.23$ m) is according to the Z axis. The height of the beam (h) is modelled by the thickness of the shell elements.

Material

The beam material is standard steel with a density of $\rho = 7500$ kg/m³, a Young modulus of $E = 189$ GPa and a Poisson coefficient of $\nu = 0.3$.

Boundary Conditions - Loading

Only the degrees of freedom UX and RZ of the beam nodes are free, except at one end where $RZ = 0$ and UX is imposed. At this end, a UX displacement is imposed as « sine sweep » with a frequency that varies from 50 Hz to 500 Hz in 1s (frequency range that covers the first four natural frequencies of the beam).

Results-Qualification

The FFT (Fast Fourier Transform) of the displacement of the free end of the beam is calculated. The frequencies of the first four modes are identified and are compared with those obtained by modal analysis performed with ANSYS of a beam with the same properties modeled using Timoshenko beam elements. The differences between EUROPLEXUS and ANSYS frequencies are less than 2% which is a good agreement. The qualification is carried out on the time history results since the FFT is not directly available in EUROPLEXUS.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Ce test est exécuté uniquement en séquentiel

Qualifications : **OK : 18/18**

VL_JRC_CAVI51

Description

Title:

Cavi51: steam explosion in a 3D cavity.

Problem:

This problem was suggested by ISMES (I) in the mid-nineties as a check of the code capability to model an extremely violent phenomenon, the steam explosion caused by sudden drop of hot corium material (following an HCDA) into a pool of liquid water situated in a cavity between the primary reactor vessel (supposed to be imperforated!) and the rigid cylindrical walls. The problem is 3D because of a lateral corridor that opens in the cavity and puts it in communication with an external room. All walls are considered rigid, but the deformations (plasticity) of the primary vessel must be modeled.

Mesh:

The model is 3D and uses 7664 elements FL38 for the fluid and 3072 shell elements COQI for the vessel. The calculation is ALE and starts with a hot and high-pressure steam bubble at the center of the liquid pool.

Materials:

Because of the huge distortions and the thinness of the cavity the only way to simulate this test is by means of the multi-phase multi-component fluid material model (FLMP). This consists of three components: the liquid water (WT0 model) for the pool, a perfect gas for the air and another perfect gas, initially at high pressure and temperature, for the steam bubble. The structure uses the VM23 elasto-plastic material.

Boundary Conditions:

The vessel is entirely blocked along the top circle. The fluid-structure interaction is automatically modeled by the FSA directive along the fluid-vessel interface, and by FSR along the external surface of the fluid (the cavity walls are supposed rigid).

Loading:

The system is initially at rest, but not in equilibrium.

Calculation:

The calculation is performed up to 50 ms. At the final time, the vessel has undergone large plastifications at the bottom and also near the top (water-hammer effect), and the liquid water has invaded both the cavity and the external room.

Results:

Although no comparison with experiments can be done on this problem, the reference results appear physically correct.

Post-treatment:

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

Reference:

Generalization of the Finite Element Model for Compressible Fluids in PLEXIS-3C to Multi-Phase Flows, F. Casadei, Technical Note N. I.97.33, March 1997 (available on the EUROPLEXUS Consortium Web site).

Exécution

Modifications du(des) jeu(x) de données : aucune.

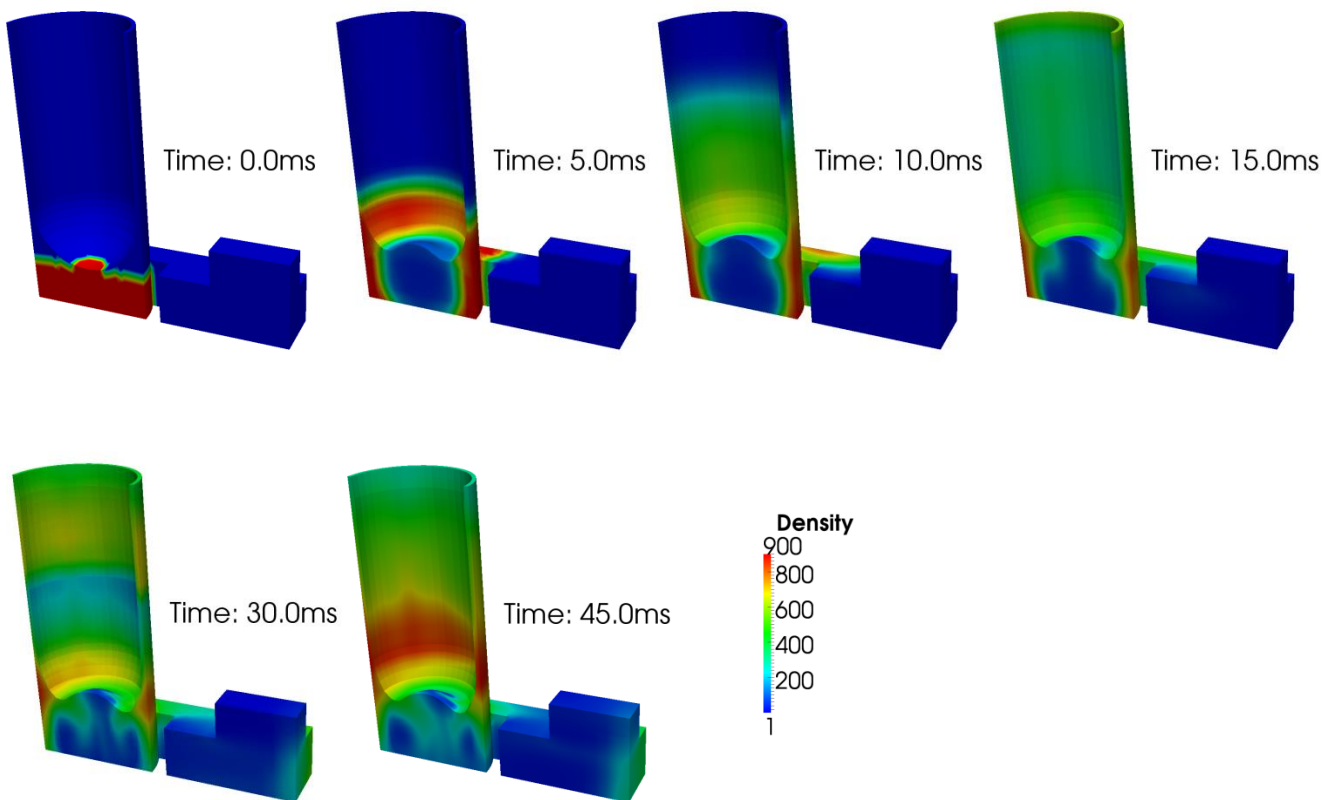
Remarque :

L'exécution des fichiers de post-traitement *vl_jrc_cavi51_press.epx*, *vl_jrc_cavi51_stru.epx*, *vl_jrc_cavi51_strub.epx*, *vl_jrc_cavi51_struc.epx*, *vl_jrc_cavi51_velo.epx*, *vl_jrc_cavi51_water.epx* et *vl_jrc_cavi51_water_surf.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

OK : 2/2

Illustrations



VL_JRC_CONT01

Description*Title:*

Cont01: Hypothetical Core Disruptive Accident (HCDA) in a Liquid-Metal Cooled Fast Breeder Reactor (LMFBR).

Problem:

This is the well-known CONT problem, a benchmark used from the 1970s to validate nuclear safety codes. Following an HCDA, a corium bubble at high pressure and temperature is formed in the core region of a LMFBR. The bubble expands and pushes the liquid sodium pool upwards. The inner shield is deformed and the liquid sodium level raises, compressing the inert cover gas. Towards the end of the transient, a water-hammer effect against the reactor roof (assumed rigid here) causes plastic deformation of the upper part of the primary vessel. Finally, the bubble starts to collapse and instabilities appear on its surface. The calculation is ALE and uses the LAGC parameter for the roof impact.

Mesh:

The model is 2D axisymmetric and uses 267 FLU1 quadrilateral/triangular specialized fluid elements (recently replaced by FL24/FL23) for the fluid domain, and 33 conical shell elements ED41 for the structures.

Materials:

The vessel and shield are made of two different steels, modeled by the VM23 elasto-plastic material. The corium bubble, liquid sodium and cover gas are represented by three different FLUT materials. These may not mix up with one another, so all fluid/fluid interfaces are treated as Lagrangian in this simulation.

Boundary Conditions:

The reactor roof is rigid, while the primary vessel is clamped at the upper part and the inner shield is clamped at its bottom. Fluid-structure interaction is imposed by the (now obsolete) FSS ALE and LAGR directives. Lagrangian sliding takes part in the upper part of the interface between liquid sodium and primary vessel. The slug impact against the roof is modeled by the IMPA directive.

Loading:

The system is initially at rest, but not in equilibrium since the pressure in the corium bubble is very high.

Calculation:

The calculation is performed up to 250 ms. At the final time, the inner shield and the primary vessel have both undergone some plastifications.

Results:

These results have been found in good agreement both with our computer codes and with experiments.

Post-treatment:

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

References:

- 1) Comparison of Different LMFBR Primary Containment Codes Applied to a Benchmark Problem, A. Benuzzi, Nuclear Engineering and Design, Vol. 100, pp. 239-249, 1987.
- 2) Use of PLEXUS as a LMFBR primary containment code for the CONT benchmark problem, F. Casadei, A. Daneri, G. Toselli, 10th International Conference on Structural Mechanics in Reactor Technology, Anaheim, U.S.A., August 14-18, 1989.
- 3) Further Development and Validation of Multi-Phase Multi-Component Compressible Fluid Models in PLEXIS-3C, F. Casadei, Technical Note N. I.99.28, February 1999.

These documents are available on the EUROPLEXUS Consortium Web site.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque :

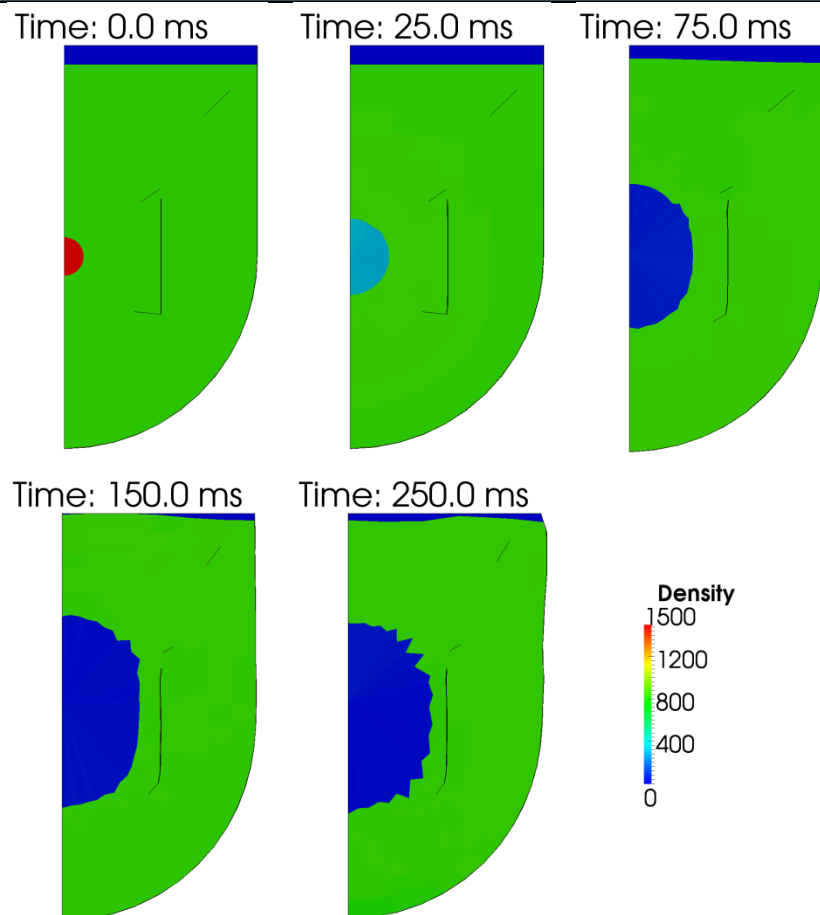
L'exécution des fichiers de post-traitement *vl_jrc_cont01_mate.epx*, *vl_jrc_cont01_press.epx*, *vl_jrc_cont01_stru.epx*, *vl_jrc_cont01_struv.epx* et *vl_jrc_cont01_velo.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Plusieurs messages ATTENTION sont obtenus concernant une possible incompatibilité entre certains éléments. Cela n'affecte pas les résultats.

Qualifications :

OK : 2/2

Illustrations



VL_JRC_CONT02

Description*Title:*

Cont02: Hypothetical Core Disruptive Accident (HCDA) in a Liquid-Metal Cooled Fast Breeder Reactor (LMFBR). Version with multi-phase multi-component fluid material.

Problem:

This is the well-known CONT problem, a benchmark used from the 1970s to validate nuclear safety codes. Following an HCDA, a corium bubble at high pressure and temperature is formed in the core region of a LMFBR. The bubble expands and pushes the liquid sodium pool upwards. The inner shield is deformed and the liquid sodium level raises, compressing the inert cover gas. Towards the end of the transient, a water-hammer effect against the reactor roof (assumed rigid here) causes plastic deformation of the upper part of the primary vessel. Finally, the bubble starts to collapse and instabilities appear on its surface. The calculation is ALE and uses the FLMP multi-phase multi-component fluid material to represent the entire fluid domain. In this way, Lagrangian fluid-fluid interfaces are avoided.

Mesh:

The model is 2D axisymmetric and uses 230 FL24 and FL23 quadrilateral/triangular specialized fluid elements (recently replaced by FL24/FL23) for the fluid domain, and 33 conical shell elements ED01 for the structures.

Materials:

The vessel and shield are made of two different steels, modeled by the VM23 elasto-plastic material. The corium bubble, liquid sodium and cover gas are represented by three different FLUT materials, which may mix up with one another (FLMP material model).

Boundary Conditions:

The reactor roof is rigid, while the primary vessel is clamped at the upper part and the inner shield is clamped at its bottom. Fluid-structure interaction is imposed by the FSA directive. Due to the absence of fluid-fluid interfaces, all FS interactions are of the ALE type so the FSA directive is sufficient.

Loading:

The system is initially at rest, but not in equilibrium since the pressure in the corium bubble is very high.

Calculation:

The calculation is performed up to 250 ms. At the final time, the inner shield and the primary vessel have both undergone some plastifications.

Results:

These results have been found in good agreement both with our computer codes and with experiments.

Post-treatment:

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

References:

- 1) Comparison of Different LMFBR Primary Containment Codes Applied to a Benchmark Problem, A. Benuzzi, Nuclear Engineering and Design, Vol. 100, pp. 239-249, 1987.
- 2) Use of PLEXUS as a LMFBR primary containment code for the CONT benchmark problem, F. Casadei, A. Daneri, G. Toselli, 10th International Conference on Structural Mechanics in Reactor Technology, Anaheim, U.S.A., August 14-18, 1989.
- 3) Further Development and Validation of Multi-Phase Multi-Component Compressible Fluid Models in PLEXIS-3C, F. Casadei, Technical Note N. I.99.28, February 1999.

These documents are available on the EUROPLEXUS Consortium Web site.

Exécution

Modifications du(des) jeu(x) de données : aucune.

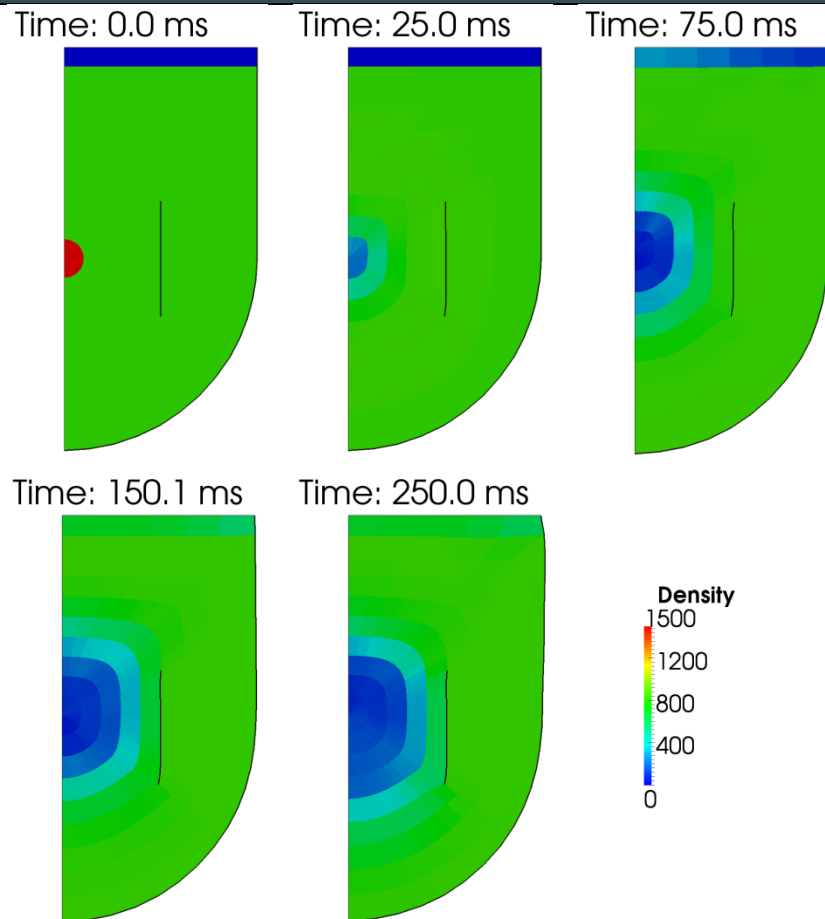
Remarque :

L'exécution des fichiers de post-traitement *vl_jrc_cont02_mate.epx*, *vl_jrc_cont02_press.epx*, *vl_jrc_cont02_stru.epx*, *vl_jrc_cont02_struv.epx* et *vl_jrc_cont02_velo.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

OK : 1/1

Illustrations



VL_JRC_HONEYCOMB

Description

Title:

Honeycomb: 4-point bending of honeycomb structure for train carriages.

Problem:

This problem has been taken from the paper listed below in the references and reproduces an experiment of 4-point bending of an extruded aluminum panel used in railway carriages. Loading is very slow (nominally static) in the experiment. In the numerical simulation, we use a somewhat faster imposed displacement for efficiency of the explicit calculation, but still in relatively slow regime.

Mesh:

The model is 3D and uses 3712 elements Q4GS for the structure. A localized "defect" is introduced near the center of the panel (see mesh generation file) to induce the right startup of the dynamic buckling mode.

Materials:

An elasto-plastic VM23 material is used to simulate the aluminum. A failure criterion is introduced.

Boundary Conditions:

The panel is held fixed along two lines, and an imposed displacement is applied along other two lines (4-point bending). This induces some localized buckling of the panel.

Loading:

The applied "load" is rather an applied displacement. The resulting reaction force (as computed by the code) is monitored and compared with the experimentally measured value.

Calculation:

The calculation is performed up to 100 ms. At the final time, the panel has undergone large motions and some localized buckling in its central part.

Results:

Despite the use of a rather coarse mesh (much coarser than the meshes used in the calculations contained in the cited paper), reference results compare well with the experiment (see cited paper) both as concerns the buckling mode and the measured force. The experimental load curves are given in Figure 10 of the mentioned paper, while the obtained final shape of the panel is shown in Figure 25 of the paper.

Post-treatment:

An animation of the computed results from this calculation is available on the EUROPLEXUS Consortium Web site.

Reference:

Fracture prediction in 4-point bending of an extruded aluminium Panel, L. Zheng, D. Petry, T. Wierzbicki, H. Rapp, Thin-Walled Structures 43 (2005), 565-590.

Exécution

Modifications du(des) jeu(x) de données : aucune.

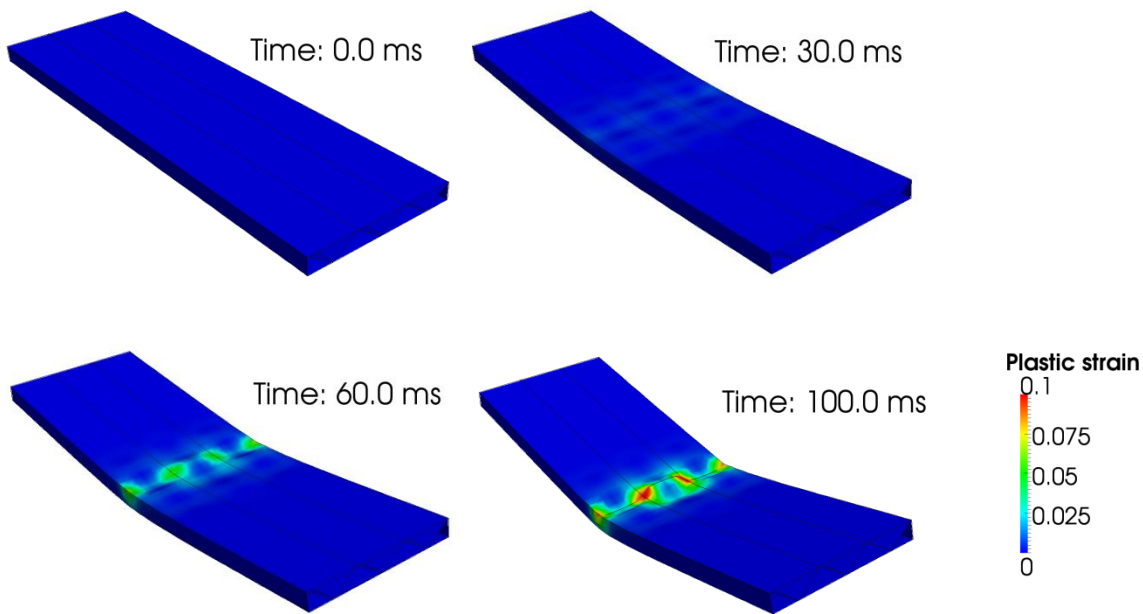
Remarque :

L'exécution des fichiers de post-traitement *vl_jrc_honeycomb_XX.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

OK : 1/1

Illustrations



VL_JRC_HOPBARUR

Description

Title:

Hopbarur: Hopkinson bar unconfined rigid

Problem:

This is a simulation of an experiment performed at JRC on the Large Hopkinson Bar device. A cylindrical specimen of Aluminium is subjected to dynamic compressive loading.

Mesh:

The model is 3D and uses 2240 elements CUBE, 6914 PRIS and 8319 TETR to model the central part of the experimental loading device. The input and output bars are modeled by 3756 BR3D bar elements.

Materials:

The experimental apparatus uses the LINE material while the specimen uses the VMIS DYNA elasto-plastic material law.

Boundary Conditions:

The RIGI directive is used to connect the 1-D bars with the 3-D model and the extremities of the specimen with the machine.

Loading:

The system is initially at rest, but not in equilibrium. A pre-stress simulating traction of 86 tons is applied to the input bar, whose far extremity is blocked.

Calculation:

The calculation is performed up to 10 ms. At the final time, the specimen has undergone large plastification.

Results:

Comparison of the reference solution with the JRC experiment is done, showing good agreement.

Post-treatment:

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

Reference:

LDTF calculations with EUROPLEXUS, C. Giry, July 2008 (available on the EUROPLEXUS Consortium Web site).

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque :

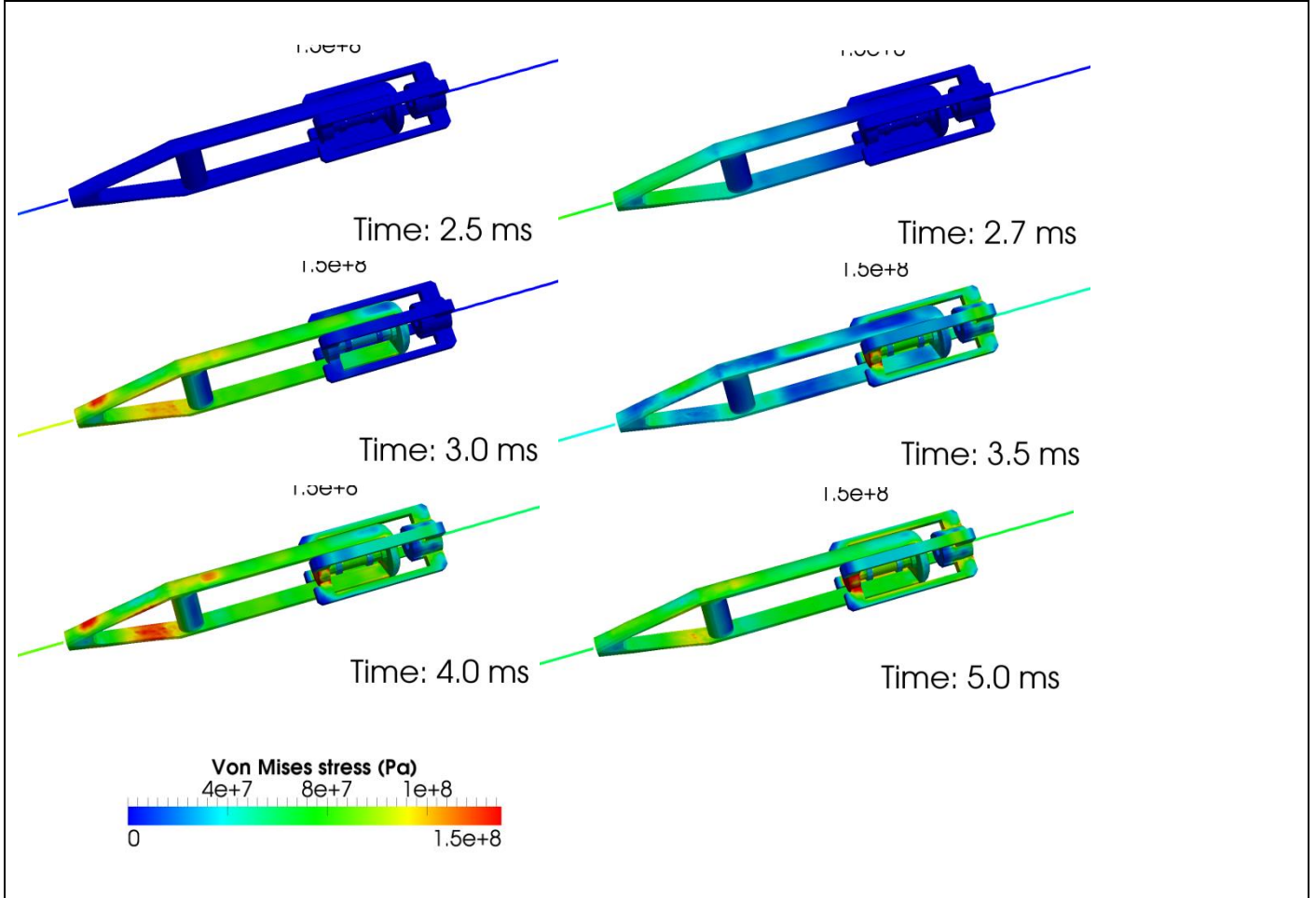
Le cas test est composé de deux jeux de données, vl_jrc_hopbarur.epx et vl_jrc_hopbarur.epx_post.epx, séparant le calcul du post-traitement.

L'exécution des fichiers de post-traitement *results.epx* et *hopbarura.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

6/6

Illustrations



VL_JRC_INDE10

Description*Title:*

Inde10: indentation problem, 2D axisymmetric solution.

Problem:

This problem was suggested by EDF as a check of the code capability to model a classical contact problem by the contact models available, in particular by the pinballs method. The ideal problem is a static one. A rigid spherical indenter is forced to penetrate into an elastic perfectly plastic half-space. The computed result is the resultant of the contact forces in the indentation direction as a function of time. An approximated analytical solution exists, due to Johnson.

Mesh:

The model is 2D axisymmetric and uses 1694 elements Q42L for the piece (half-space represented by a square region of side equal to 3 times the indenter radius), and one PMAT element for the indenter.

Materials:

The VM23 material is assigned to the piece, with zero plastic hardening. The MASS element is assigned to the indenter.

Boundary Conditions:

All nodes on the axis of symmetry are blocked in the radial direction. The base of the piece is blocked in all directions. Contact is represented by the pinball model. A single pinball with radius equivalent to that of the indenter is associated with the PMAT element. The region of the piece likely to come in contact is filled by parent pinballs with a hierarchy level of 4 to get accurate contact resolution.

Loading:

The indenter is pushed into the piece at constant imposed speed (linear displacement in time), until it reaches an indentation depth equal to the radius of the indenter.

Initial Conditions:

The indenter has an initial velocity equal to the imposed indentation velocity (to avoid an initial error in the energy balance).

Calculation:

The calculation is performed up to 50 ms. At the final time, the indenter has reached a depth equal to its radius.

Results:

The computed resultant contact force is in good agreement with the approximate analytical solution up to an indentation depth equal to approximately 1/2 of the indenter radius. For larger indentations, the analytical (linear) solution is no longer valid.

Post-treatment:

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

Référence:

- 1) Indentation élasto-plastique d'un bloc par un indenteur sphérique élastique, N. Tardieu, B. Serre, Code_Aster, Manuel de Validation, Report V6.04.506, décembre 2002.
- 2) Validation of the EUROPLEXUS Pinball Impact-Contact Model on an Indentation Problem, F. Casadei, Technical Note, July 2007 (available on the EUROPLEXUS Consortium Web site).

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque :

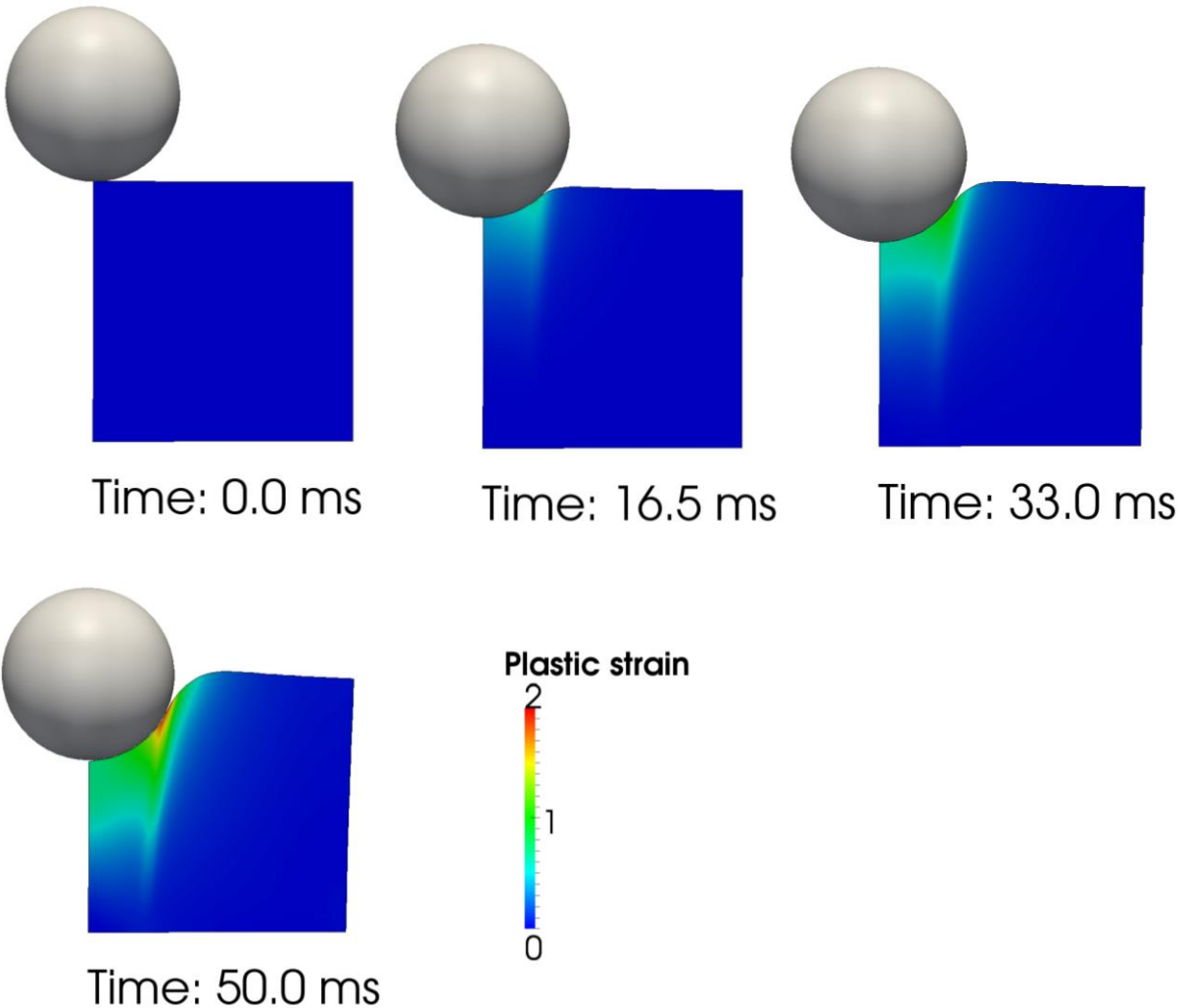
Le cas test est composé de deux jeux de données, vl_jrc_inde10.epx et vl_jrc_inde10_post.exp, séparant le calcul du post-traitement.

L'exécution des fichiers de post-traitement *rend.epx*, *rend_dnorm.epx* et *rend_fe.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

1/1

Illustrations



VL_JRC_INDE13

Description

Title:

Inde13: indentation problem, 3D non-axisymmetric solution.

Problem:

This problem was suggested by EDF as a check of the code capability to model a classical contact problem by the contact models available, in particular by the pinballs method. The ideal problem is a static one. A rigid spherical indenter is forced to penetrate into an elastic perfectly plastic half-space. The computed result is the resultant of the contact forces in the indentation direction as a function of time. An approximated analytical solution exists, due to Johnson.

Mesh:

The model is 3D non-axisymmetric and uses 9361 elements CUB8 for the piece (half-space represented by a square region of side equal to 3 times the indenter radius), and one PMAT element for the indenter.

Materials:

The VMIS PARF material is assigned to the piece, with zero plastic hardening. The MASS element is assigned to the indenter.

Boundary Conditions:

All nodes on the two planes of symmetry are blocked in the suitable direction. The base of the piece is blocked in all directions. Contact is represented by the pinball model. A single pinball with radius equivalent to that of the indenter is associated with the PMAT element. The region of the piece likely to come in contact is filled by parent pinballs with a hierarchy level of 4 to get accurate contact resolution.

Loading:

The indenter is pushed into the piece at constant imposed speed (linear displacement in time), until it reaches an indentation depth equal to the radius of the indenter.

Initial Conditions:

The indenter has an initial velocity equal to the imposed indentation velocity (to avoid an initial error in the energy balance).

Calculation:

The calculation is performed up to 50 ms. At the final time, the indenter has reached a depth equal to its radius.

Results:

The computed resultant contact force is in good agreement with the approximate analytical solution up to an indentation depth equal to approximately 1/2 of the indenter radius. For larger indentations, the analytical (linear) solution is no longer valid.

Post-treatment:

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

Références:

- 1) Indentation élasto-plastique d'un bloc par un indenteur sphérique élastique, N. Tardieu, B. Serre, Code_Aster, Manuel de Validation, Report V6.04.506, décembre 2002.
- 2) Validation of the EUROPLEXUS Pinball Impact-Contact Model on an Indentation Problem, F. Casadei, Technical Note, July 2007 (available on the EUROPLEXUS Consortium Web site).

Exécution

Modifications du(des) jeu(x) de données : aucune.

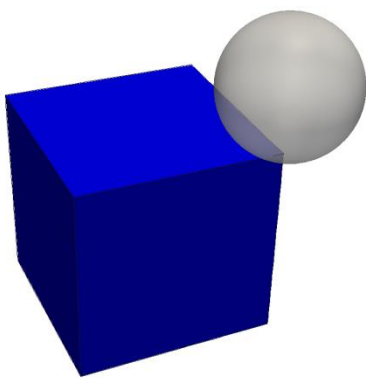
Remarque :

L'exécution des fichiers de post-traitement *pinb.epx*, *rend.epx*, *rend_dnorm.epx* et *rend_surf.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

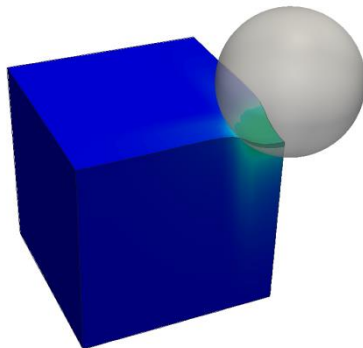
Qualifications :

Qualification manquante. Solution correcte par comparaison visuelle avec la référence 2D-axi (test vl_jrc_inde10).

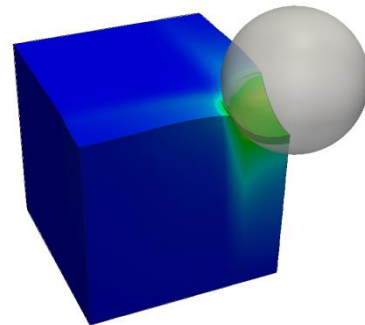
Illustrations



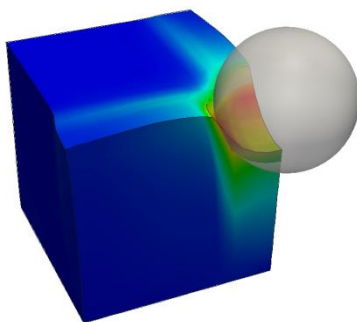
Time: 0.0 ms



Time: 16.5 ms



Time: 33.0 ms



Time: 50.0 ms

Plastic strain



VL_JRC_REAC05

Description

Title:

Reac05: propagation of steam explosion pressure waves in the secondary containment of a nuclear reactor. Similar to Stea05 but with reinforced concrete material model for the building.

Problem:

This problem was suggested by ISMES (I) in the early-nineties. It represents the effects of a steam or gas explosion on the secondary containment building of a nuclear reactor. The explosion takes place in a liquid water pool located immediately below the primary containment (reactor vessel) and produces strong pressure waves, which propagate across the atmosphere in the reactor building, interacting with its walls and with some schematized internal structures. The explosive products are schematized by a high-pressure gas.

Mesh:

The model is 2D axisymmetric and uses 24 elements FL23 and 957 elements FL24 to represent the fluid, and 88 shell elements ED01 for the building walls and internals. Shell elements are made of 5 layers each, 3 of which representing the concrete and 2 for the steel reinforcements. The calculation is ALE and starts with a hot and high-pressure steam bubble in the lower part of the building.

Materials:

The bottom of the building is assumed to be rigid, while the walls and internals, which are made of reinforced concrete material, are represented by the DPSF model, a Drucker-Prager constitutive law with softening and viscoplastic regularization, for the concrete, and by the VMSF (Von Mises elasto-plastic material with softening) for the steel reinforcement.

Boundary Conditions:

The fluid-structure interaction along the deformable building walls is automatically modeled by the FSA directive.

Loading:

The system is initially at rest, but not in equilibrium.

Calculation:

The calculation is performed up to 250 ms. This time is sufficient to see propagation of the pressure waves to the top of the building and several reflections.

Results:

Although no comparison with experiments can be done on this problem, the results appear physically correct.

Post-treatment:

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

References:

- 1) Simulation of a Gas Explosion in a Reactor Containment by PLEXIS-3C, F. Casadei, Technical Note N. I.93.148, November 1993.
- 2) Fast Transient Analysis of Thin Shell Reinforced Concrete Structures with Drucker-Prager Model, J.J. Lopez Cela, F. Casadei and P. Pegon, SMiRT-14 Conference, Lyon (F), August 17-22, 1997.

(both available on the EUROPLEXUS Consortium Web site).

Exécution

Modifications du(des) jeu(x) de données : aucune.

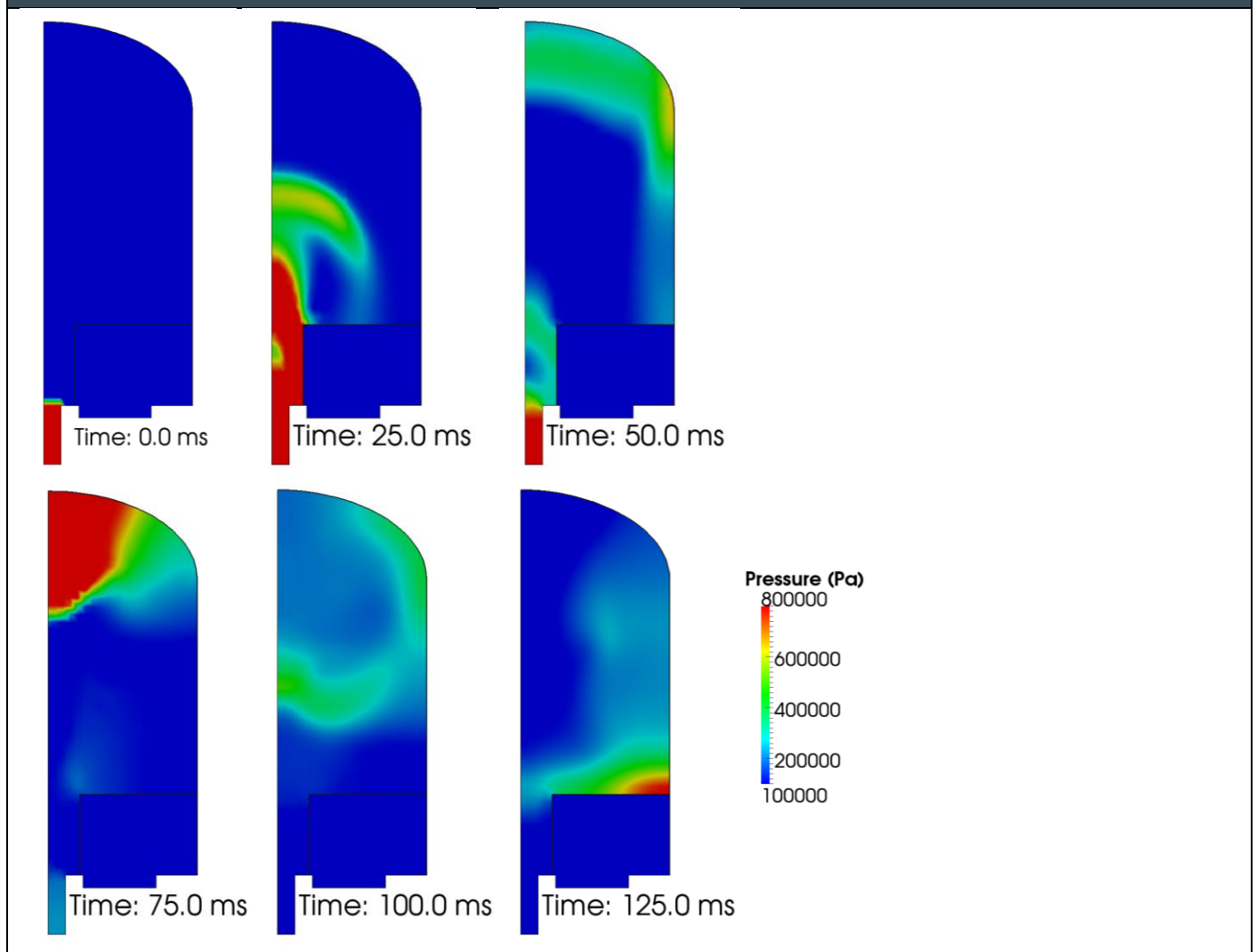
Remarque :

L'exécution des fichiers de post-traitement *vl_jrc_reac05_press.epx*, *vl_jrc_reac05_press_surf.epx*, *vl_jrc_reac05_stru.epx*, *vl_jrc_reac05_struv.epx* et *vl_jrc_reac05_velo.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

OK : 2/2

Illustrations



VL_JRC_SONA01

Description

Title: Bird strike on a simplified aircraft wing section.

Problem:

This test verifies a bird strike on a simplified aircraft wing section. The bird is modelled by SPH.

Mesh:

The following mesh is used:

- 1 element PMAT
- 2719 elements BILL
- 110 elements DST3
- 4361 elements Q4GS

Calculation:

The calculation is performed up to 3 ms.

Exécution

Modifications du(des) jeu(x) de données : aucune.

Qualifications :

OK : 3/3

VL_JRC_STEA05

Description*Title:*

Stea05: propagation of steam explosion pressure waves in the secondary containment of a nuclear reactor.

Problem:

This problem was suggested by ISMES (I) in the early-nineties. It represents the effects of a steam or gas explosion on the secondary containment building of a nuclear reactor. The explosion takes place in a liquid water pool located immediately below the primary containment (reactor vessel) and produces strong pressure waves, which propagate across the atmosphere in the reactor building, interacting with its walls and with some schematized internal structures. The explosive products are schematized by a high-pressure gas.

Mesh:

The model is 2D axisymmetric and uses 24 elements FL23 and 957 elements FL24 to represent the fluid, and 88 shell elements ED01 for the building walls and internals. The calculation is ALE and starts with a hot and high-pressure steam bubble in the lower part of the building.

Materials:

The bottom of the building is assumed to be rigid, while the walls and internals are represented as linear elastic, in the hypothesis that the reinforced concrete does not reach its yield limit.

Boundary Conditions:

The fluid-structure interaction along the deformable building walls is automatically modeled by the FSA directive.

Loading:

The system is initially at rest, but not in equilibrium.

Calculation:

The calculation is performed up to 250 ms. This time is sufficient to see propagation of the pressure waves to the top of the building and several reflections.

Results:

Although no comparison with experiments can be done on this problem, the results appear physically correct.

Post-treatment:

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

References:

1) Simulation of a Gas Explosion in a Reactor Containment by PLEXIS-3C, F. Casadei, Technical Note N. I.93.148, November 1993.

2) Fast Transient Analysis of Thin Shell Reinforced Concrete Structures with Drucker-Prager Model, J.J. Lopez Cela, F. Casadei and P. Pegon, SMiRT-14 Conference, Lyon (F), August 17-22, 1997.

(both available on the EUROPLEXUS Consortium Web site).

Exécution

Modifications du(des) jeu(x) de données : aucune.

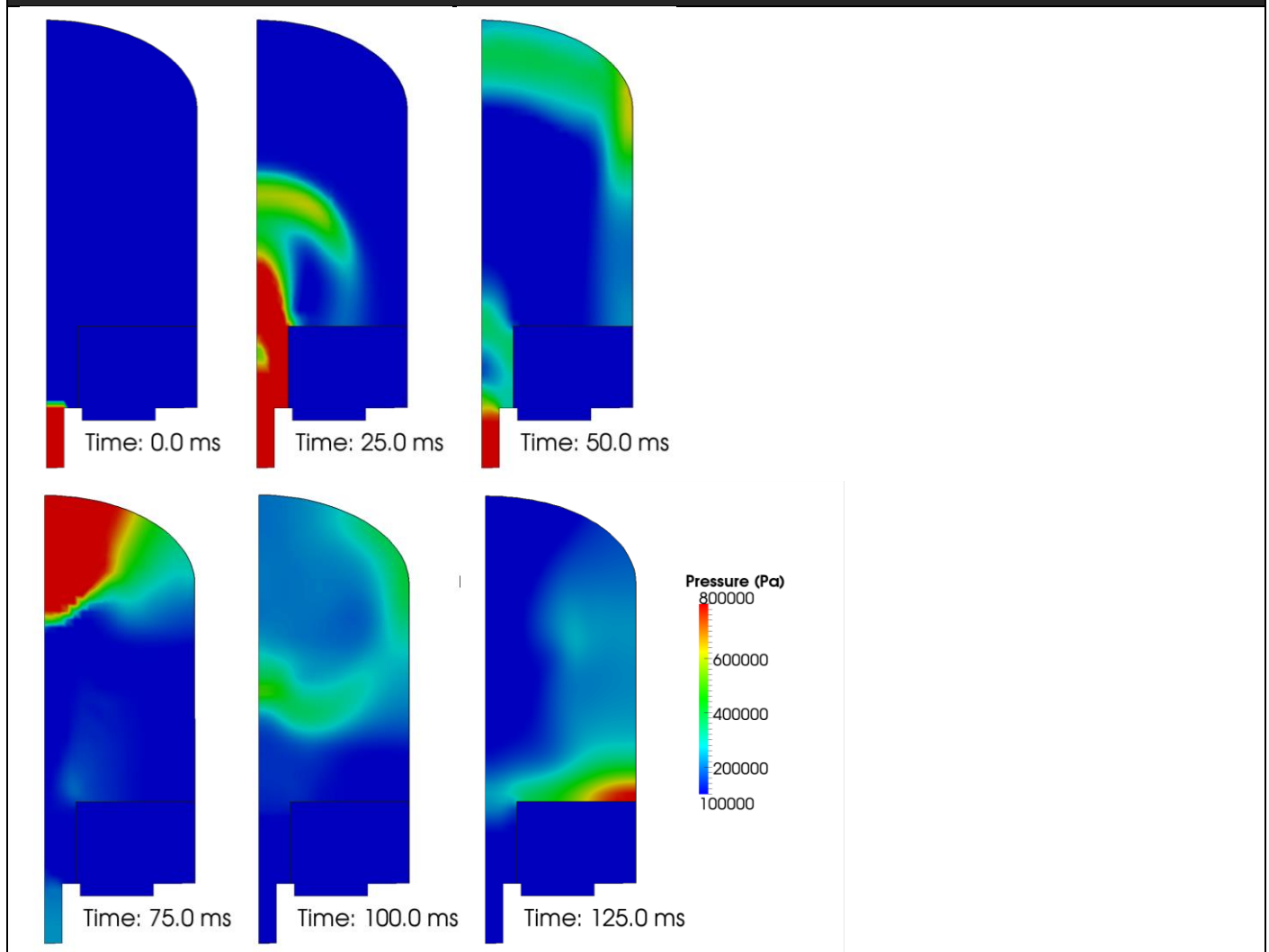
Remarque :

L'exécution des fichiers de post-traitement *vl_jrc_stea05_press.epx*, *vl_jrc_stea05_press_surf.epx*, *vl_jrc_stea05_stru.epx*, *vl_jrc_stea05_struv.epx* et *vl_jrc_stea05_velo.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

OK : 2/2

Illustrations



VL_JRC_TOWE02

Description*Title:*

Towe02: impact against a cooling tower

Problem:

This problem is a very schematic simulation of an impact against a cooling tower. The parabolic tower is 123.9 m high and has a ground diameter of 95.8 m, a minimum diameter of 65.72 m (at a height of 88.44 m) and a top diameter of 68.86 m. It is schematized (very roughly) by a shell of constant thickness, equal to 0.25 m, made of reinforced concrete. The geometry is axisymmetric, but the loading is not so the problem has to be treated in 3D. The applied load is a time-dependent pressure in a localized zone near the top, simulating an impact of a projectile or flying object. The base of the tower is assumed completely blocked and thanks to the presence of a vertical symmetry plane, only one half of the tower needs to be modeled.

Mesh:

The model is 3D and uses 800 triangular plate/shell elements COQI.

Materials:

The tower is made of reinforced concrete, modeled by the DPSF material for the concrete and VMSF for the steel reinforcement. The structure is composed of 5 layers, 3 of which being concrete and 2 reinforcements. The respective thickness fractions are 0.1, 0.0025, 0.795, 0.0025 and 0.1.

Boundary Conditions:

The tower is entirely blocked along the bottom basis. A symmetry plane is imposed.

Loading:

The system is initially at rest, and starting at the initial time a step-wise external pressure is applied onto a square region near the top of the tower, simulating the impact zone. The pressure stays constant for a duration of 500 ms, then goes linearly to 0 in 100 ms.

Calculation:

The calculation is performed up to 2000 ms. At the final time, the tower has undergone large plastifications in the impact zone and to some extent in the surrounding area.

Results:

Although no comparison with experiments can be done on this problem, the results appear physically correct.

Post-treatment:

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

References:

- 1) Fast Transient Analysis of Thin Shell Reinforced Concrete Structures with Drucker-Prager Model, J.J. Lopez Cela, F. Casadei and P. Pegon, SMiRT-14 Conference, Lyon (F), August 17-22, 1997.
- 2) Brittle Material Law with Drucker Prager Yield Surface and Softening Behaviour, J.J. Lopez Cela, P. Pegon, F. Casadei, Technical Note N. I.96.34, February 1996.
- 3) A Multilayer Formulation for Shell Elements in PLEXIS-3C, F. Casadei, J.J. Lopez Cela, Technical Note N. I.96.14, February 1996.

(all these documents are available on the EUROPLEXUS Consortium Web site).

Exécution

Modifications du(des) jeu(x) de données : aucune.

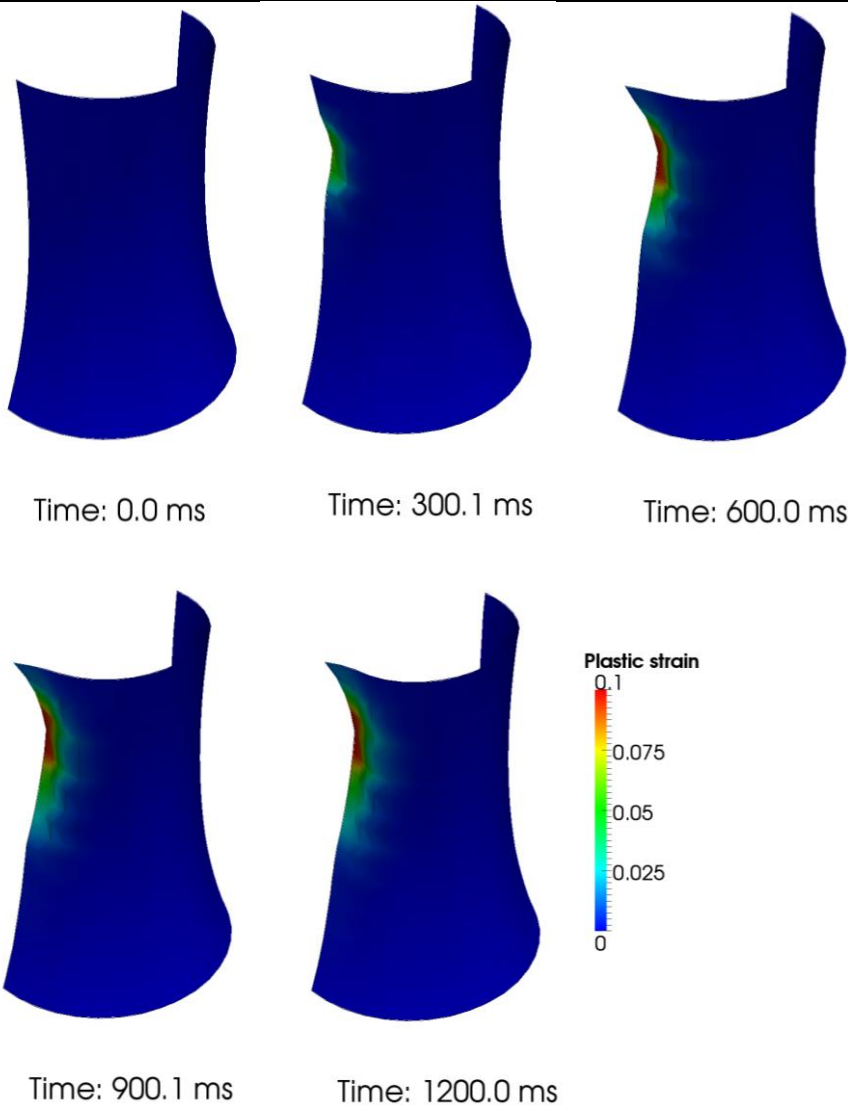
Remarque :

L'exécution des fichiers de post-traitement *vl_jrc_towe02_stru.epx*, *vl_jrc_towe02_struv.epx* et *vl_jrc_towe02_struv2.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

OK : 2/2

Illustrations



VL_JRC_WOCO2D

Description*Title:*

Woco2d: the well-known Woodward-Colella test in 2D, revisited by using a deformable structure (FSI).
Supersonic flow over a forward-facing deformable step.

Problem:

This is the well-known Woodward-Colella problem, a benchmark used to validate CFD codes. A channel is filled by gas flowing at Mach 3, initially in uniform conditions. The channel is 3 m long, 1 m high and 0.4 m wide (but the problem may be considered as 2-D plane). A deformable step is introduced at time 0 at 0.6 m downstream from the channel inlet. The step is 0.2 m high and its thickness and material law are chosen so as to undergo relatively large deformations during the transient solution. An ALE calculation with fluid-structure interaction is performed. Mesh rezoning is done automatically by Giuliani's algorithm.

Mesh:

The model is 2D plane deformation and uses 8064 MC34 triangular finite volumes (4193 nodes) for the fluid domain, and 104 beam/shell elements for the step.

Materials:

The step is made of elasto-plastic metallic material (VM23), while the fluid is a mixture of perfect gases (MCGP). A special MCFE material is used to model the inlet and outlet conditions (far field).

Boundary Conditions:

The step is entirely blocked at the outlet, and blocked in the vertical direction only in the front tip. At the inlet and outlet suitable boundary conditions are prescribed in the fluid. The boundary conditions at the channel inlet also correspond to the initial conditions. The same boundary conditions are assumed at the channel outlet, although they will not be taken into consideration by the numerical scheme, since they correspond to a supersonic outlet where all the characteristic lines leave the computational domain.

Loading:

The system is initially at rest, but not in equilibrium since the pressure in the fluid will tend to deform the step, which is disturbing the initially uniform flow field.

Calculation:

The calculation is performed up to 3.5 ms. At the final time, the shock detached from the step has hit the upper part of the channel being reflected again towards the downstream portion of the step.

Results:

These reference results have been found in good agreement with those reported in the literature for the case with rigid step.

Post-treatment :

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

References:

- 1) The numerical simulation of Two-Dimensional Fluid Flow with Strong Shocks, P. Woodward and P. Colella, J. Comp. Phys., 54, pp. 115-173 (1984).
- 2) Modelling of Arbitrary Lagrangian-Eulerian Multicomponent Flow with Fluid-Structure Interaction in PLEXIS-3C, A. Soria, F. Casadei, Special Publication N. I.95.01, Jan. 1995.
- 3) Arbitrary Lagrangian-Eulerian Multicomponent Compressible Flow with Fluid-Structure Interaction, A. Soria, F. Casadei, International Journal for Numerical Methods in Fluids, Vol. 25, pp. 1263--1284, December 1997. (last two are available in the EUROPLEXUS Consortium Web site)

Exécution

Modifications du(des) jeu(x) de données : aucune.

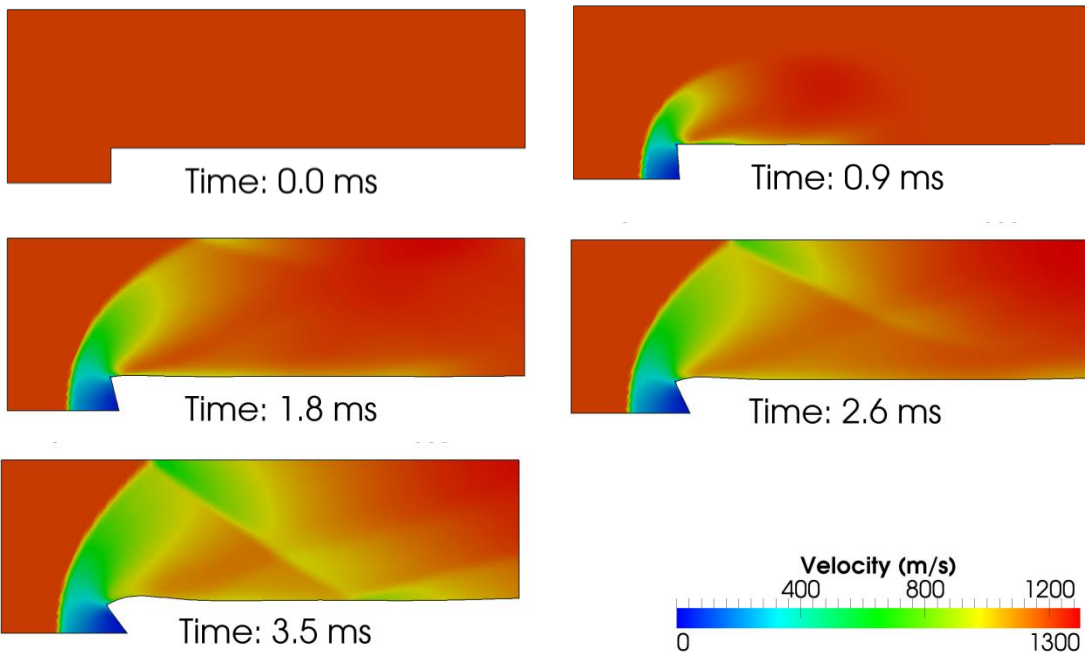
Remarque :

L'exécution des fichiers de post-traitement *vl_jrc_woco2d_rezo.epx* et *vl_jrc_woco2dc.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

OK : 8/8

Illustrations



VL_JRC_WOCO3D

Description*Title:*

Woco3d: the well-known Woodward-Colella test in 3D, revisited by using a deformable structure (FSI).
Supersonic flow over a forward-facing deformable step.

Problem:

This is the well-known Woodward-Colella problem, a benchmark used to validate CFD codes. A channel is filled by gas flowing at Mach 3, initially in uniform conditions. The channel is 3 m long, 1 m high and 0.4 m wide (but the problem may be considered as 2-D plane). A deformable step is introduced at time 0 at 0.6 m downstream from the channel inlet. The step is 0.2 m high and its thickness and material law are chosen so as to undergo relatively large deformations during the transient solution. An ALE calculation with fluid-structure interaction is performed. Mesh rezoning is done automatically by Giuliani's algorithm.

Mesh:

The model is 3D and uses 8064 MC36 triangular prism finite volumes (8386 nodes) for the fluid domain, and 104 beam/shell elements Q4GS for the step.

Materials:

The step is made of elasto-plastic metallic material (VM23), while the fluid is a mixture of perfect gases (MCGP). A special MCFP material is used to model the inlet and outlet conditions (far field).

Boundary Conditions:

The step is entirely blocked at the outlet, and blocked in the vertical direction only in the front tip. At the inlet and outlet suitable boundary conditions are prescribed in the fluid. The boundary conditions at the channel inlet also correspond to the initial conditions. The same boundary conditions are assumed at the channel outlet, although they will not be taken into consideration by the numerical scheme, since they correspond to a supersonic outlet where all the characteristic lines leave the computational domain.

Loading:

The system is initially at rest, but not in equilibrium since the pressure in the fluid will tend to deform the step, which is disturbing the initially uniform flow field.

Calculation:

The calculation is performed up to 3.5 ms. At the final time, the shock detached from the step has hit the upper part of the channel being reflected again towards the downstream portion of the step.

Results:

These reference results have been found in good agreement with those reported in the literature for the case with rigid step.

Post-treatment :

Several animations of the computed results from this calculation are available on the EUROPLEXUS Consortium Web site.

References:

- 1) The numerical simulation of Two-Dimensional Fluid Flow with Strong Shocks, P. Woodward and P. Colella, J. Comp. Phys., 54, pp. 115-173 (1984).
- 2) Modelling of Arbitrary Lagrangian-Eulerian Multicomponent Flow with Fluid-Structure Interaction in PLEXIS-3C, A. Soria, F. Casadei, Special Publication N. I.95.01, Jan. 1995.
- 3) Arbitrary Lagrangian-Eulerian Multicomponent Compressible Flow with Fluid-Structure Interaction, A. Soria, F. Casadei, International Journal for Numerical Methods in Fluids, Vol. 25, pp. 1263--1284, December 1997.
(last two are available in the EUROPLEXUS Consortium Web site)

Exécution

Modifications du(des) jeu(x) de données : aucune.

Remarque :

L'exécution des fichiers de post-traitement *vl_jrc_woco3d_rezo.epx* et *vl_jrc_woco3dc.epx* est ignorée, car ils sont destinés à une version d'EPX utilisant la visualisation OpenGL interne, ce qui n'est pas le cas de la version concernée par ce document. Ils n'apportent aucune information de qualification supplémentaire.

Qualifications :

OK : 8/8

Illustrations

